

Circuit Systems with MATLAB[®] and PSpice[®]

Circuit Systems with MATLAB[®] and PSpice[®]

Won Y. Yang and Seung C. Lee

Chung-Ang University, South Korea



John Wiley & Sons(Asia) Pte Ltd

Copyright © 2007

John Wiley & Sons (Asia) Pte Ltd, 2 Clementi Loop, #02-01,
Singapore 129809

Visit our Home Page on www.wiley.com

MATLAB[®] and Simulink[®] are trademarks of The MathWorks, Inc. and are used with permission. The MathWorks does not warrant the accuracy of the text or exercises in this book. This book's use or discussion of MATLAB[®] and Simulink[®] software or related products does not constitute endorsement or sponsorship by The MathWorks of a particular pedagogical approach or particular use of the MATLAB[®] and Simulink[®] software.

All Rights Reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, scanning, or otherwise, except as expressly permitted by law, without either the prior written permission of the publisher, or authorization through payment of the appropriate photocopy fee to the Copyright Clearance Center. Requests for permission should be addressed to the Publisher, John Wiley & Sons (Asia) Pte Ltd, 2 Clementi Loop, #02-01, Singapore 129809, tel: 65-64632400, Fax: 65-64646912, email: enquiry@wiley.com.sg

Designations used by companies to distinguish their products are often claimed as trademarks. All brand names and product names used in this book are trade names, service marks, trademarks or registered trademarks of their respective owners. The Publisher is not associated with any product or vendor mentioned in this book.

This publication is designed to provide accurate and authoritative information in regard to the subject matter covered. It is sold on the understanding that the Publisher is not engaged in rendering professional services. If professional advice or other expert assistance is required, the services of a competent professional should be sought.

Other Wiley Editorial Offices

John Wiley & Sons Ltd, The Atrium, Southern Gate, Chichester, West Sussex PO19 8SQ, UK

John Wiley & Sons, Inc., 111 River Street, Hoboken, NJ 07030, USA

Jossey-Bass, 989 Market Street, San Francisco, CA 94103-1741, USA

Wiley-VCH Verlag GmbH, Boschstr. 12, D-69469 Weinheim, Germany

John Wiley & Sons Australia Ltd, 42 McDougall Street, Milton, Queensland 4064, Australia

John Wiley & Sons Canada Ltd, 6045 Freemont Blvd, Mississauga, ONT, L5R 4J3, Canada

Wiley also publishes its books in a variety of electronic formats. Some content that appears in print may not be available in electronic books.

Anniversary Logo Design: Richard J. Pacifico

ISBN 978-0-470-82232-6 (HB)

Typeset in 9/11pt Times by Thomson Digital Noida, India

Printed and bound in Singapore by Markono Print Media Pte Ltd, Singapore.

This book is printed on acid-free paper responsibly manufactured from sustainable forestry in which at least two trees are planted for each one used for paper production.

*To our parents and families
who love and support us
and
to our teachers and students
who enriched our knowledge*

‘Why, I haven’t failed, I’ve just found 10,000 ways that won’t work’, replied Thomas Alva Edison (1847–1931), a great inventor, asked once if he wasn’t discouraged because his work was not going well. When Edison was old and close to death, he said that the biggest mistake he had made was that he never respected Tesla or his work. After Edison died, Tesla was quoted as saying, ‘I was almost a sorry witness of his doings, knowing that just a little theory and calculation would have saved him 90 per cent of the labor. But he had a veritable contempt for book learning and mathematical knowledge, trusting himself entirely to his inventor’s instinct and practical American sense.’

Nikola Tesla (1856–1943), another great inventor, had more than 700 patents on the AC induction motor, Tesla coil transformer, AC power transmission, wireless transmission, radio, fluorescent light, etc. He sailed from Europe for America in 1884, arriving in New York, with a few cents and a letter of recommendation to Edison from Charles Batchelor, his previous manager, saying, ‘I know two great men and you are one of them; the other is this young man.’ He joined Edison’s company, but walked off the job when Edison reneged on his promise of a bonus, and then established his own laboratory in 1886. He virtually put an end to ‘War of Currents’ between the DC forces headed by Edison and the AC forces led by Westinghouse in favor of the latter by selling his patent rights to the polyphase system of AC motors, dynamos, and transformers. In 1893 Westinghouse used Tesla’s AC power system to light the World’s Columbian Exposition in Chicago.

Contents

Preface	xiii
Limits of Liability and Disclaimer of Warranty of Software	xv
1 Basic Concepts on Electric Circuits	1
1.1 Symbols and Units	1
1.2 Network Variables	1
1.2.1 Voltage and Current	1
1.2.2 Electric Power and Energy	3
1.2.3 Reference Polarity and Direction of Voltage/Current	3
1.2.4 Passive Sign Convention	4
1.3 Circuit Elements	5
1.3.1 Passive Elements	5
1.3.2 Active Elements	9
1.3.3 Operational Amplifier	10
1.3.4 Transistor	13
1.4 Kirchhoff's Laws	13
1.4.1 Nodes, Branches, and Meshes/Loops	14
1.4.2 Kirchhoff's Current Law (KCL)	15
1.4.3 Kirchhoff's Voltage Law (KVL)	16
1.4.4 The Number of KCL/KVL Equations	18
1.5 Equivalent Transformation of Sources	19
1.5.1 Combination of Several Sources	19
1.5.2 Voltage–Current Source Transformation	21
1.5.3 Examples of Source Transformation	23
1.6 Series and Parallel Connections	25
Problems	25
2 Resistor Circuits	35
2.1 Combination of Resistors	35
2.1.1 Series Combination of Resistors	35
2.1.2 Parallel Combination of Resistors	36
2.2 Voltage/Current Divider	37
2.2.1 Voltage Divider	37
2.2.2 Current Divider	38
2.3 Δ -Y(II-T) Transformation	38
2.3.1 Δ -Y(II-T) Conversion Formula	39
2.3.2 Y- Δ (T-II) Conversion Formula	39
2.4 Node Analysis	40
2.4.1 Circuits Having No Dependent Sources	42

2.4.2 Circuits Having Dependent Sources	45
2.5 Mesh (Loop) Analysis	48
2.5.1 Circuits Having No Dependent Sources	49
2.5.2 Circuits Having Dependent Sources	53
2.6 Comparison of Node Analysis and Mesh Analysis	56
2.7 Thevenin/Norton Equivalent Circuits	63
2.8 Superposition Principle and Linearity	71
2.9 OP Amp Circuits with Resistors	72
2.9.1 Inverting OP Amp Circuit	72
2.9.2 Noninverting OP Amp Circuit	74
2.9.3 Voltage Follower	76
2.9.4 More Exact Analysis of OP Amp Circuits	77
2.9.5 OP Amp Circuits with Positive Feedback	78
2.10 Transistor Circuits	81
2.11 Loading Effect and Input/Output Resistance	81
2.12 Load Line Analysis of Nonlinear Resistor Circuits	82
2.13 More Examples of Resistor Circuits	86
Problems	95
3 First-Order Circuits	111
3.1 Characteristics of Inductors and Capacitors	111
3.1.1 Inductor	111
3.1.2 Capacitor	113
3.2 Series–Parallel Combination of Inductors/Capacitors	115
3.2.1 Series–Parallel Combination of Inductors	115
3.2.2 Series–Parallel Combination of Capacitors	116
3.3 Circuit Analysis Using the Laplace Transform	117
3.3.1 The Laplace Transform for a First-Order Differential Equation	118
3.3.2 Transformed Equivalent Circuits for R , L , and C	119
3.4 Analysis of First-Order Circuits	120
3.4.1 DC-Excited RL Circuits	120
3.4.2 DC-Excited RC Circuits	123
3.4.3 Time-Constant and Natural Responses of First-Order Circuits	125
3.4.4 Sequential Switching	133
3.4.5 AC-Excited First-Order Circuits	136
3.5 Analysis of First-Order OP Amp Circuits	138
3.5.1 First-Order OP Amp Circuits with Negative Feedback	138
3.5.2 First-Order OP Amp Circuits with Positive Feedback	140
3.6 LRL Circuits and CRC Circuits	144
3.6.1 An LRL Circuit	144
3.6.2 A CRC Circuit	146
3.6.3 Conservation of Flux Linkage and Charge	148
3.6.4 A Measure Against Violation of the Continuity Rule on the Inductor Current	148
3.7 Simulation Using PSpice and MATLAB	149
3.7.1 An RC Circuit with Sequential Switching	149
3.7.2 An AC-Excited RL Circuit	151
3.8 Application and Design of First-Order Circuits	152
Problems	159

4 Second-Order Circuits	177
4.1 The Laplace Transform For Second-Order Differential Equations	177
4.1.1 Overdamped Case with Two Distinct Real Characteristic Roots	178
4.1.2 Critically Damped Case with Double Real Characteristic Roots	179
4.1.3 Underdamped Case with Two Distinct Complex Characteristic Roots	179
4.1.4 Stability of a System and Location of its Characteristic Roots	180
4.2 Analysis of Second-Order Circuits	181
4.2.1 A Series <i>RLC</i> Circuit	181
4.2.2 A Parallel <i>RLC</i> Circuit	192
4.2.3 Two-Mesh/Node Circuit	198
4.2.4 Circuits Having Dependent Sources	200
4.2.5 Thevenin Equivalent Circuit	202
4.3 Second-Order OP Amp Circuits	203
4.4 Analogy and Duality	205
4.4.1 Analogy	205
4.4.2 Duality	206
4.5 Transfer Function, Impulse Response, and Convolution	207
4.5.1 Linear Systems	208
4.5.2 Time-Invariant Systems	208
4.5.3 The Pulse Response of a Linear Time-Invariant System	208
4.5.4 The Input–Output Relationship of a Linear Time-Invariant System	209
4.6 The Steady-State Response to a Sinusoidal Input	211
4.7 An Example of MATLAB Analysis and PSpice Simulation	213
Problems	214
5 Magnetically Coupled Circuits	223
5.1 Self-Inductance	223
5.2 Mutual Inductance	225
5.3 Relative Polarity of Induced Voltages and Dot Convention	226
5.3.1 Dot Convention and Sign of Mutual Inductance Terms	226
5.3.2 Measurement of the Relative Winding Direction	226
5.3.3 Measurement of Mutual Inductance	227
5.3.4 Energy in Magnetically Coupled Coils	228
5.4 Equivalent Models of Magnetically Coupled Coils	228
5.4.1 T-Equivalent Circuit	229
5.4.2 II-Equivalent Circuit	234
5.5 Ideal Transformer	237
5.6 Linear Transformer	240
5.7 Autotransformers	241
Problems	243
6 AC Circuits	255
6.1 Sinusoidal Sources	255
6.2 Phasor and AC Analysis	256
6.3 AC Impedance of Passive Elements	261
6.3.1 Resistor	261
6.3.2 Inductor	261
6.3.3 Capacitor	262

6.4 AC Circuit Examples	263
6.5 Instantaneous, Active, Reactive, and Complex Power	275
6.6 Power Factor	278
6.7 Maximum Power Transfer – Impedance Matching	283
6.8 Load Flow Calculation	285
6.9 Design and Simulation for Maximum Power Transfer	286
Problems	289
7 Three-Phase AC Circuits	299
7.1 Balanced Three-Phase Voltages	299
7.2 Power of Balanced Three-Phase Loads	302
7.3 Measurement of Three-Phase Power	303
7.4 Three-Phase Power System	304
7.5 Electric Shock and Grounding	310
Problems	313
8 Frequency Selective Circuit – Filter	319
8.1 Lowpass Filter (LPF)	319
8.1.1 Series LR Circuit	319
8.1.2 Series RC Circuit	320
8.2 Highpass Filter (HPF)	321
8.2.1 Series CR Circuit	321
8.2.2 Series RL Circuit	321
8.3 Bandpass Filter (BPF)	322
8.3.1 Series RLC Circuit and Series Resonance	322
8.3.2 Parallel RLC Circuit and Parallel Resonance	326
8.4 Bandstop Filter (BSF)	329
8.4.1 Series RLC Circuit	329
8.4.2 Parallel RLC Circuit	332
8.5 Active Filter	333
8.5.1 First-Order Active Filter	333
8.5.2 Second-Order Active LPF/HPF	334
8.5.3 Second-Order Active BPF	336
8.5.4 Second-Order Active BSF	337
8.6 Analog Filter Design	341
Problems	354
9 Circuits Analysis Using Fourier Series	373
9.1 Fourier Series	373
9.2 Computation of Fourier Coefficients Using Symmetry	375
9.3 Circuit Analysis Using Fourier Series	379
9.4 Fourier Series and Laplace Transform	387
9.5 RMS Value and Power of a Nonsinusoidal Periodic Signal	393
9.5.1 RMS Value and Distortion Factor of a Nonsinusoidal Periodic Signal	393
9.5.2 Power and Power Factor of a Nonsinusoidal Periodic Signal	394
Problems	395

10 Two-Port Networks	401
10.1 Definitions of Two-Port Parameters	401
10.2 Relationships Among Two-Port Parameters	406
10.2.1 The z -Parameters and a -Parameters	406
10.2.2 The a -Parameters and h -Parameters	407
10.2.3 The z -Parameters and h -Parameters	408
10.3 Reciprocity of a Two-Port Network	411
10.4 Interconnection of Two-Port Networks	413
10.4.1 Series Connection and z -Parameters	413
10.4.2 Parallel (Shunt) Connection and y -Parameters	414
10.4.3 Series–Parallel (Shunt) Connection and h -Parameters	415
10.4.4 Parallel (Shunt)–Series Connection and g -Parameters	415
10.4.5 Cascade Connection and a -Parameters	416
10.4.6 Curse of the Port Condition (Current Requirement)	416
10.5 Two-Port Networks Having Source/Load	420
10.5.1 Input Impedance	422
10.5.2 Voltage Gain	423
10.5.3 Current Gain	423
10.5.4 (Thevenin) Equivalent Impedance Seen from the Output	424
10.5.5 (Thevenin) Equivalent Source Seen from the Output	424
10.5.6 The Parameters of an Overall Two-Port Network	425
10.6 Feedback Amplifiers as Two-Port Networks	430
10.6.1 Series–Parallel (Shunt) Feedback Amplifier	431
10.6.2 Series–Series Feedback Amplifier	431
10.6.3 Parallel–Parallel Feedback Amplifier	432
10.6.4 Parallel (Shunt)–Series Feedback Amplifier	433
10.6.5 General Feedback Structure	434
10.7 Circuit Models with Given Parameters	438
10.7.1 Circuit Model with Given z -Parameters	438
10.7.2 Circuit Model with Given y -Parameters	438
10.7.3 Circuit Model with Given h and g -Parameters	438
10.7.4 Circuit Model with Given a and b -Parameters	438
Problems	440
Appendices	451
Appendix A: Laplace Transform	451
Appendix B: Matrix Operations with MATLAB	461
Appendix C: Complex Number Operations with MATLAB	466
Appendix D: Nonlinear/Differential Equations with MATLAB	468
Appendix E: Symbolic Computations with MATLAB	471
Appendix F: Useful Formulas (Reference [K-2])	474
Appendix G: The Standard Values of Resistors, Capacitors, and Inductors	476
Appendix H: OrCAD/PSpice (References [K-1] and [R-2])	481
Appendix I: MATLAB Introduction (Reference [K-2])	511
Appendix J: Solutions to Problems	514
References	525
Index	527

Preface

Knowledge in electric circuits is crucial to students majoring in Electrical Engineering since it provides them with not only the very basics of Electricity but also the key concepts about the general system theory. As with most subjects in the field of Engineering, the ultimate objective of the subject ‘Electric Circuits’ is to equip students with the capability of designing electric circuits in order to meet given specifications as well as the ability of modeling and analyzing electric circuits. In this context, the book tries to bring the reader’s attention to the circuit designer’s point of view, while allocating most of the pages to explaining how to analyze electric circuits and find their voltages and currents. This book also presents how OrCAD/PSpice can be used for simulating circuit systems. The features of this book can be summarized as follows:

1. Instead of the conventional method of using general/particular solutions, this book lays emphasis on the Laplace transform method for solving the differential equations for electric circuits. We recommend taking the Laplace transform of electric circuits (containing inductors/capacitors) and setting up the transformed circuit equations directly in the unified framework (as if they were just made of resistors and sources) rather than setting up the circuit equations in the form of differential equations and then taking their Laplace transforms to solve them. The Laplace transform and the inverse Laplace transform are introduced in the Appendix.
2. This book presents several MATLAB programs that can be used to get the Laplace transformed solutions, take their inverse Laplace transforms, and plot the solutions along the time or frequency axis. The MATLAB programs can save a lot of time and effort when solving the equations so that readers can concentrate on establishing circuit equations, gaining insights, and making interpretations.
3. This book also introduces step by step how to use OrCAD/PSpice (version 10.0) for circuit simulations. For circuit problems that take much time to solve by hand, readers are recommended to use MATLAB and PSpice. This approach gives the readers not only information about the state of the art but also self-confidence on the condition that the graphical solutions obtained by using the two software tools agree with each other. OrCAD/PSpice is introduced in the Appendix. However, the amount of MATLAB and PSpice used has been restricted to avoid readers becoming too dependent on the software in case they are tempted to neglect the importance of basic circuit theory.
4. Each example shows something different from other examples so that readers can acquire the essential circuit analysis techniques efficiently and gain insights into the various types of circuits. On the other hand, instead of repeating similar exercise problems, most exercise problems are designed to arouse readers’ interest in practical applications or help form a view for circuit application and design.
5. For representative examples, we present the analytical solution together with the results of MATLAB analysis (close to the theory) and PSpice simulation (close to the experiment) in the form of a trinity. I am sure this style of presentation will interest many students, attracting their attention to the topics on circuits efficiently.

The contents of this book are derived from the works of many (known or unknown) great scientists, scholars, and researchers, all of whom are deeply appreciated. We would like to thank the reviewers for their valuable comments and suggestions, which contributed to enriching this book:

Professor Byung-Suhl Suh

Department of Electricity, Control, and Biomedical Engineering, Hanyang University

Professor Jang-Myung Lee

School of Electronic, Electrical, and Communication Engineering, Pusan University

Professor Sungwon Cho

Department of Electronics and Electrical Engineering, Hong-Ik University

Professor Dongwook Park

Department of Electronics and Electrical Engineering, Hong-Ik University

Professor Jun Heo

Department of Electronics, Information and Communication Engineering, Konkuk University

We also thank the people of the School of Electronic and Electrical Engineering, Chung-Ang University, for giving us an academic environment. Without the affection and support of our families and friends, this book could not have been written. Special thanks should be given to Senior Researcher Yong-Suk Park at KETI (Korea Electronics Technology Institute) for his invaluable help in correction. We also thank the editorial and production staff at Wiley including Copyeditor Mrs. Pat Bateson, Content Editor Mr. Brett Wells, and Project Editor Ms. Sarah Hinton, who contributed to the production of this book. We express our special gratitude to Commissioning Editor James Murphy for his kind, efficient, and encouraging guidance.

Any questions, comments, and suggestions regarding this book are welcome. They should be sent to wyyang53@hanmail.net.

Won Young Yang and Seung Chul Lee

Limits of Liability and Disclaimer of Warranty of Software

The reader is expressly warned to consider and adopt all safety precautions that might be indicated by the activities herein and to avoid all potential hazards. By following the instructions given, the reader willingly assumes all risks in connection with such instructions.

The authors and publisher of this book have used their best efforts and knowledge in preparing this book as well as developing the computer programs in it. However, they make no warranty of any kind, expressed or implied, with regard to the programs or the documentation contained in this book. Accordingly, they will not be liable for any incidental or consequential damages in connection with, or arising out of, the readers' use of, or reliance upon, the material in this book.

Questions about the contents of this book can be mailed to wyyang@cau.ac.kr.

Program files can be downloaded from the following website:

(http://www.wiley.com/go/yang_circuit)

MATLAB is a registered trademark of The MathWorks, Inc. For MATLAB and Simulink product information, please contact:

The MathWorks, Inc.
3 Apple Hill Drive
Natick, MA 01760-2098, USA
Tel: (+1)508-647-7000, Fax: (+1)508-647-7001
E-mail: info@mathworks.com
Website: www.mathworks.com

OrCAD, PSpice, and Cadence are registered trademarks of the Cadence Design Systems, Inc. The authors gratefully acknowledge Cadence Design Systems, Inc. for making OrCAD and PSpice available.

1

Basic Concepts on Electric Circuits

Electricity means physical phenomena that are caused by the presence or movement of electric charge and can be measured in the form of voltages, currents, and charges. Electric circuit means a physical system composed of electric elements such as resistors, inductors, capacitors, sources, etc., in which electric charges can move, or its model given in the form of circuit diagrams. This chapter begins with network variables such as voltage, current, and power involved in electric circuits. Then it introduces several circuit elements, each with its voltage–current relationship (VCR) and Kirchhoff’s laws, which present the basis for circuit equations. This chapter ends with the equivalent transformation of voltage and current sources that are very useful for the analysis of electric circuits.

1.1 Symbols and Units

Throughout this book SI (International System) units are used, summarized in Table 1.1.

1.2 Network Variables

1.2.1 Voltage and Current

In the gravitational field with gravitational acceleration $g[\text{m/s}^2 = \text{N/kg}]$ a point of height $h[\text{m}]$ measured from a point of reference like the ground in the direction against the gravity has a *gravitational potential*

Table 1.1 SI units

Quantity/ symbol	Unit	Quantity/ symbol	Unit	Quantity/ symbol	Unit
Length l	m (meter)	Voltage v	V (volt)	Resistance R	Ω (ohm)
Mass m	kg (kilogram)	Charge q	C (coulomb)	Resistivity ρ	$\Omega \text{ m}$
Time t	s (second)	Current i	A (ampere)	Conductance G	S (siemens)
Force f	N (newton)	Electric field E	V/m or N/C	Conductivity σ	S/m
Energy w	J (joule)	Magnetic flux ϕ	Wb (weber)	Capacitance C	F (farad)
Power p	W (watt)	Magnetic flux density B	Wb/m ²	Permittivity ε	F/m = C ² /N m ²
Velocity v	m/s			Inductance L	H (henry) = Wb/A
Acceleration a	m/s ²			Permeability μ	N/A ² = H/m

$gh[\text{m}^2/\text{s}^2 = \text{J/kg}]$. A mass $m[\text{kg}]$ positioned at that point has a gravitational potential (mechanical) energy $W_g = mgh[\text{kg} \cdot \text{m}^2/\text{s}^2 = \text{N} \cdot \text{m} = \text{J}]$ and is forced to move toward another point of lower gravitational potential by the *gravitational force* $F_g = mg[\text{kg} \cdot \text{m}/\text{s}^2 = \text{N}]$ (see Figure 1.1(a)). Likewise in the electric field with constant intensity $E[\text{V}/\text{m} = \text{N}/\text{C}]$, a point of distance $d[\text{m}]$ measured from a point of reference like the electrical ground in the direction against the electric field has an *electric potential* or *voltage* $V = Ed[\text{V} = \text{J}/\text{C}]$. A charge $q[\text{C}]$ positioned at that point has an electric potential energy $W_e = qV[\text{C} \cdot \text{V} = \text{J}]$ and is forced to move toward another point of lower electric potential by the *electric force* $F_e = qE[\text{N}]$ (see Figure 1.1(b)).

It is rather the (relative) *voltage* or *potential difference* between points than the (absolute) voltage or potential at each point (or node) that really matters in most electric circuits. However, for convenience, the ground is often made the reference when describing the potential V in an electric field, just as the sea level is set as the reference when mentioning the height in a gravitational field. The potential difference produced by a voltage source such as a generator or a battery, which causes the current to flow in a circuit, is referred to as an *electromotive force* (emf), while the potential difference that results from the current flowing through a passive element such as a resistor is referred to as a *voltage drop*.

Just as the gravitational potential at a point is the energy that would be required to move a unit mass from the reference point to that point, the electric potential or voltage at a point is the energy that would be required to move a unit of positive charge from the reference point to that point. This definition of voltage can be described by the following equations:

$$v[\text{V}] = \frac{dw}{dq}[\text{J}/\text{C}] \quad (1.1a)$$

and

$$V[\text{V}] = \frac{W}{Q}[\text{J}/\text{C}] \quad (1.1b)$$

where the lower-case letters v , w , and q and the upper-case letters V , W , and Q represent possibly varying and nonvarying voltage, energy, and charge, respectively. If the voltage varies with the movement of a charge, only Equation (1.1a) can apply; otherwise, either of these two definitions will do since they conform with each other. If some amount of work is not required but is performed by the charge movement from the reference point to a point, the sign of the voltage at that point should be negative.

The *current* is a stream of charges, whose magnitude is defined to be the rate of charge flow as

$$i[\text{A}] = \frac{dq}{dt}[\text{C}/\text{s}] \quad (1.2a)$$

and

$$I[\text{A}] = \frac{Q}{t}[\text{C}/\text{s}] \quad (1.2b)$$

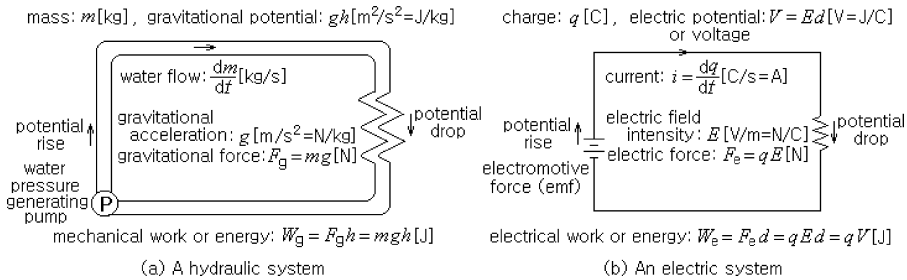


Figure 1.1 A hydraulic system and an electric system

where the lower-case letter i and the upper-case letter I represent possibly varying and nonvarying current, respectively. If the current varies with time, only Equation (1.2a) can apply; otherwise, either of these two definitions will do since they conform with each other. The direction of current is defined to be that of positive charges (holes) or opposite to that of electrons.

1.2.2 Electric Power and Energy

Electric power is defined to be the time rate of absorbing or supplying (electric) energy as

$$p [\text{W}] = \frac{dw}{dt} [\text{J/s}] \quad (1.3a)$$

and

$$P[\text{W}] = \frac{W}{t} [\text{J/s}] \quad (1.3b)$$

where the lower-case letter p and the upper-case letter P represent possibly varying and nonvarying power, respectively. If the power varies with time, only Equation (1.3a) can apply; otherwise, either of these two definitions will do since they conform with each other. Note that the (electric) power can be obtained from the product of the associated voltage and current:

$$p(t) = \frac{dw}{dt} = \frac{dw}{dq} \frac{dq}{dt} \stackrel{(1.1a),(1.2a)}{=} v(t) i(t) \quad (1.4a)$$

and

$$P = \frac{W}{t} = \frac{W}{Q} \frac{Q}{t} \stackrel{(1.1b),(1.2b)}{=} VI \quad (1.4b)$$

1.2.3 Reference Polarity and Direction of Voltage/Current

For the voltages of the elements in Figures 1.2.1(a) and (b), it could be said that ‘The voltage at the left terminal (node a) is 5 V higher than that at the right terminal (node b)’ and ‘The voltage at the right terminal (node b) is 3 V higher than that at the left terminal (node a)’, respectively. It would, however, be convenient to say and easy to understand if the *reference polarity* is fixed arbitrarily, as denoted by a plus–minus sign pair in Figure 1.2.2, and say that the voltages across the element are $v_{ab} = v_a - v_b = +5 \text{ V}$ and $v_{ab} = v_a - v_b = -3 \text{ V}$, respectively. It does not matter which one of the two polarities is selected as the reference one. In fixing the reference polarity, there is no need to be concerned about the polarity of the real voltage since it depends on the surrounding condition of the element. However, if the polarity of the real voltage can be guessed, it is often fixed as the reference polarity.

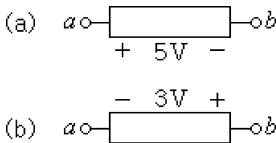


Figure 1.2.1 No reference polarity for voltages

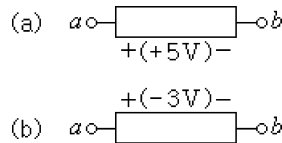


Figure 1.2.2 With reference polarity for voltages

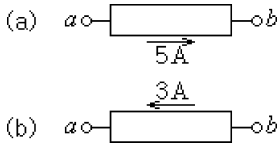


Figure 1.3.1 No reference direction for currents

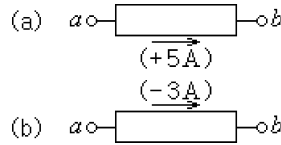


Figure 1.3.2 With reference direction for currents

For the currents of the elements in Figures 1.3.1(a) and (b), it could be said that ‘The current of 5 A flows from the left terminal (node a) to the right terminal (node b)’ and ‘The current of 3 A flows from the right terminal (node b) to the left terminal (node a)’, respectively. It would, however, be convenient to say and easy to understand if the *reference direction* is fixed arbitrarily, as denoted by an arrow in Figure 1.3.2, and say that the currents through the element are $i_{ab} = +5$ A and $i_{ab} = -3$ A, respectively. It does not matter which one of the two directions is selected as the reference one. In fixing the reference direction, there is no need to be concerned about the direction of the real current since it depends on the condition surrounding the element. However, if the direction of the real current can be guessed, it is often fixed as the reference direction.

1.2.4 Passive Sign Convention

If the reference polarity of the voltage across an element is associated with the reference direction of the current through it in such a way that the current is directed from the positive polarity (plus-signed) terminal to the negative polarity (minus-signed) terminal, as depicted in Figure 1.4.1, the sign of the power obtained as the product of the voltage and the current (each with the sign in view of the reference polarity/direction) turns out to be positive or negative depending on whether the element absorbs (passively) from or supplies (actively) to the rest part of the circuit. This is called the *passive sign convention*.

Consider the circuit shown in Figure 1.4.2, where the voltage source $V_s = 10$ V is applied across the resistor $R_L = 5 \Omega$ so that the current of $10 \text{ V} / 5 \Omega = 2$ A flows in the clockwise direction. Fixing the voltage reference polarities and current reference directions of V_s and R_L as depicted in the circuit diagram, the currents, voltages, and powers of the two elements can be written as follows:

$$\text{For } R_L : I_L = I = 2 \text{ A}, V_L = R_L I_L = 5 \times 2 = 10 \text{ V}; \quad P_L \stackrel{(1.4b)}{=} V_L I_L = 10 \times 2 = 20 \text{ W} \quad (1.5a)$$

$$\text{For } V_s : I_s = -I = -2 \text{ A}, V_s = 10 \text{ V}; \quad P_s \stackrel{(1.4b)}{=} V_s I_s = 10 \times (-2) = -20 \text{ W} \quad (1.5b)$$

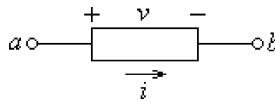


Figure 1.4.1 Reference polarity of voltage and reference direction of current associated by the passive sign convention

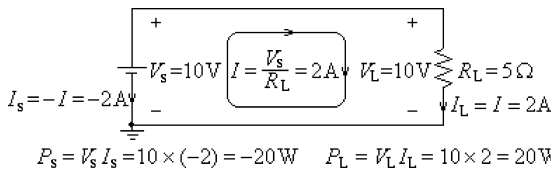


Figure 1.4.2 Passive sign convention and the sign of power

Here, $I_s = -I$ since the current I flows through V_s against its reference direction. The positive power $P_L = 20 \text{ W}$ implies that the resistor absorbs (expends) a power of 20 W (from the source), while the negative power $P_s = -20 \text{ W}$ implies that the voltage source supplies a power of 20 W (to the load).

1.3 Circuit Elements

There are two types of circuit elements used for building circuits: passive elements and active elements. If a circuit element is capable of delivering more energy than has been supplied, it is said to be *active*; otherwise, it is said to be *passive*. Resistors, inductors, and capacitors are passive elements, while generators, batteries, and operational amplifiers are active elements.

1.3.1 Passive Elements

1.3.1.1 Resistor

Figure 1.5(a) shows the symbol representing a resistor in circuit diagrams and Figure 1.5(b) shows its typical appearance. The external characteristic of a resistor as a two-terminal device can be written as its voltage–current relationship based on Ohm’s law:

$$v = Ri : \quad \text{Ohm's law} \quad (1.6a)$$

$$i = Gv \quad (1.6b)$$

This indicates that the voltage across a resistor is proportional to the current flowing through it, where the proportionality constants R and G are called the *resistance* and *conductance* of the resistor, respectively:

$$R [\Omega] = \frac{v}{i} [\text{V/A}] \quad (1.7a)$$

$$G [\text{S}] = \frac{i}{v} [\text{A/V}] \quad (1.7b)$$

The internal characteristic of a resistor is that its resistance is obtained from its length $l[\text{m}]$ and cross-sectional area $A[\text{m}^2]$ and the *resistivity* $\rho [\Omega \cdot \text{m}]$ of the material (Figure 1.5(c)) as

$$R = \rho \frac{l}{A} = \frac{l}{\sigma A} [\Omega] \quad \text{with} \quad \sigma = \frac{1}{\rho} [1/(\Omega \cdot \text{m})] : \quad \text{conductivity} \quad (1.8)$$

The power dissipated by a resistance R is

$$p \stackrel{(1.4a)}{=} vi \stackrel{(1.6a)}{=} Ri^2 \stackrel{(1.6b)}{=} Gv^2 [\text{W}] \quad (1.9)$$

A resistor is said to be *linear* if its voltage–current relationship can be represented by a straight line passing through the origin on the i – v plane, as shown in Figure 1.5(d), where the slope of the $v(i)$ line, which is the resistance, does not vary with the voltage or current. Most real resistors can be reasonably and practicably assumed modeled to be linear. However, some very useful elements such as a diode exhibit a nonlinear voltage–current relationship that can be described by a curve on the i – v plane, as illustrated in Figure 1.5(e).

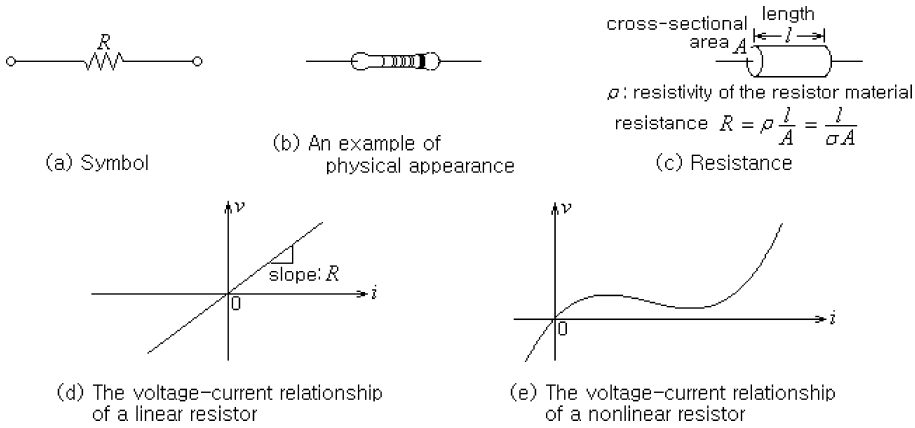


Figure 1.5 Resistor

1.3.1.2 Inductor

Figure 1.6(a) shows the symbol representing an inductor in circuit diagrams. As seen from the typical appearance illustrated in Figure 1.6(b), an *inductor* consists of a coil of insulated conducting wire wound around some magnetic or nonmagnetic material (core). For this reason, an inductor is often called a *coil*. The flux ϕ [Wb] produced by its current i [A] and linked with it is proportional to the *magnetomotive force* (mmf), i.e. the product of the number of turns N [turns] and the current, with its *permeance* $\mathcal{P}$ [Wb/A · turns] as the proportionality constant, which can be written as

$$\phi = \mathcal{P} N i \quad (1.10)$$

The product of the flux ϕ and the number of turns N [turns] is referred to as the *flux linkage*, denoted by

$$\lambda = N\phi \stackrel{(1.10)}{=} \mathcal{P} N^2 i = L i \quad (1.11)$$

where $L = \mathcal{P} N^2$ is called the (self)-*inductance* of the coil and has the dimension of Wb · turns/A, denoted by H (henry).

Faraday's law states that the change in the flux linkage induces a voltage across the coil linked with the flux, which equals the time rate of change of the flux linkage, yielding the external characteristic of an inductor as

$$v(t)[V] = \frac{d\lambda(t)}{dt} = N \frac{d\phi(t)}{dt} = L \frac{di(t)}{dt} [\text{H} \cdot \text{A/s}] \quad (1.12a)$$

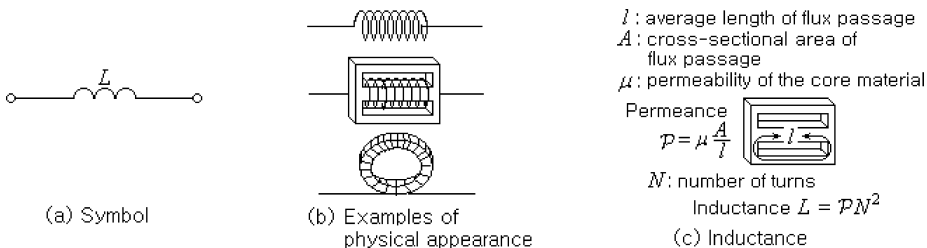


Figure 1.6 Inductor

This indicates that the voltage across an inductor is proportional to the time rate of the current flowing through it, where the proportionality constant L is the inductance of the coil. This voltage–current relationship of an inductor can also be written in integral form as

$$i(t) = \frac{1}{L} \int_{-\infty}^t v(\tau) d\tau \quad (1.12b)$$

The internal characteristic of an inductor is that its inductance is proportional to the squared number of turns:

$$L[\text{henry}] = \mathcal{P}N^2 \quad (1.13)$$

where the permeance $\mathcal{P}$ of an inductor is obtained from the (average) length l [m] and cross-sectional area A [m²] of the flux path through the core and the (magnetic) permeability μ [H/m] of the core material as follows:

$$\mathcal{P} = \frac{1}{\mathcal{R}} = \mu \frac{A}{l} [\text{Wb/A} \cdot \text{turns}] : \text{ permeance} \quad (1.14)$$

Note that the reciprocal of the permeance, denoted by $\mathcal{R}$, is called the *magnetic reluctance*.

The power transferred from or to an inductor L is

$$p(t) \stackrel{(1.4a)}{=} v(t) i(t) \stackrel{(1.12a)}{=} L i(t) \frac{di(t)}{dt} [\text{W}] \quad (1.15)$$

and this power can be integrated to obtain the *magnetic field energy* of an inductor with the current $i(t)$ as

$$w_L(t) = \int_{-\infty}^t p(\tau) d\tau \stackrel{(1.4a)}{=} \int_{-\infty}^t v(\tau) i(\tau) d\tau \stackrel{(1.12a)}{=} \int_{-\infty}^t L i(\tau) \frac{di(\tau)}{d\tau} d\tau = \int_{i(-\infty)=0}^{i(t)} L i di \stackrel{(F.32)}{=} \frac{1}{2} L i^2(t) \quad (1.16)$$

Note: Equation (F.32) is in Appendix F. The first letter or number shows which appendix or chapter the equation is from.

1.3.1.3 Capacitor

Figure 1.7(a) shows the symbol representing a capacitor in circuit diagrams. As seen from the typical appearance illustrated in Figure 1.7(b), a *capacitor* consists of two conductive plates or electrodes separated by an insulator or dielectric. It is occasionally called a *condenser*. The two plates in a pair are charged with equal but opposite electric charges if the capacitor is connected to a source. The charge on the capacitor is proportional to the voltage across it as

$$q(t) [\text{C}] = C v(t) [\text{F} \cdot \text{V}] \quad (1.17)$$

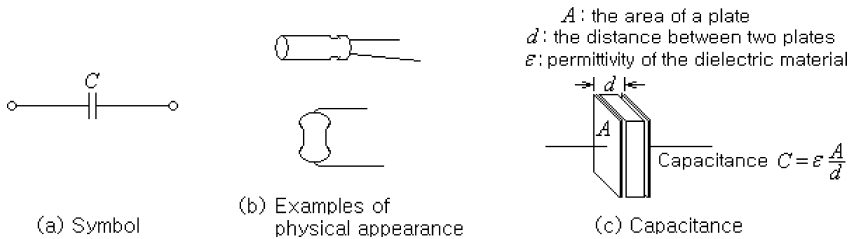


Figure 1.7 Capacitor

where the proportionality constant C is called the *capacitance* of the capacitor and has the dimension of C(coulomb)/V(volt), denoted by F(farad). Since the current is defined as the time rate of charge (by Equation (1.2a)), the charge equation (1.17) can be differentiated w.r.t. time t to get the external characteristic of a capacitor as

$$i(t) [\text{A}] = \frac{dq(t)}{dt} = C \frac{dv(t)}{dt} [\text{F} \cdot \text{V/s}] \quad (1.18a)$$

This indicates that the current flowing through a capacitor is proportional to the time rate of change of the voltage across it, where the proportionality constant C is the capacitance of the capacitor. This voltage–current relationship of a capacitor can also be written in integral form as

$$v(t) = \frac{1}{C} \int_{-\infty}^t i(\tau) d\tau \quad (1.18b)$$

The internal characteristic of a capacitor is that its *capacitance* is obtained from the distance $d[\text{m}]$ between two parallel plates, the area $A[\text{m}^2]$ of one plate, and the *permittivity* or *dielectric constant* $\varepsilon [\text{F/m}]$ of the dielectric between the plates as follows:

$$C [\text{F}] = \varepsilon \frac{A}{d} \quad (1.19)$$

The power transferred from or to a capacitor C is

$$p(t) \stackrel{(1.4a)}{=} v(t) i(t) \stackrel{(1.18a)}{=} C v(t) \frac{dv(t)}{dt} [\text{W}] \quad (1.20)$$

This power can be integrated to obtain the *electric field energy* of a capacitor with the voltage $v(t)$ as

$$w_C(t) = \int_{-\infty}^t p(\tau) d\tau \stackrel{(1.4a)}{=} \int_{-\infty}^t v(\tau) i(\tau) d\tau \stackrel{(1.18a)}{=} \int_{-\infty}^t C v(\tau) \frac{dv(\tau)}{d\tau} d\tau = \int_{v(-\infty)=0}^{v(t)} C v dv \stackrel{(F.32)}{=} \frac{1}{2} C v^2(t) \quad (1.21)$$

(Question) What does $t = -\infty$ mean as the lower limit of the integration interval in Equations (1.12b), (1.16), (1.18b), and (1.21)? Is it the beginning of the human history, the world, or the universe?

(Answer) It is reasonable to think of $t = -\infty$ as the time at which the inductor or capacitor becomes connected to a source for the first time and so is supposed to have no stored energy.

[Remark 1.1] Linear/Nonlinear, Time-Invariant/Time-Varying, and Lumped/Distributed

1. In reality, all physical resistors, inductors, and capacitors exhibit *nonlinear* and *time-varying* behaviour in their voltage–current, flux–current, or charge–voltage relationship because the resistance, inductance, and capacitance are affected by conditions such as temperature, which may vary more or less with the heating caused by current flow. However, they are often modeled to be *linear* and *time-invariant* on the assumption that such a modeling yields practicably accurate solutions.
2. In reality, resistance, inductance, and capacitance are *distributed* all over the circuits. For example, conducting wires have some amount of resistance as long as they are not made of superconducting material. Sources and switches also have some internal or contact resistance. Two conducting wires in parallel and long transmission lines or cables with earth have some capacitance. Twisted wires have some inductance. Inductors and capacitors also have some stray or leakage resistance. However, they are modeled as explicitly shown elements like R , L , and C as if they were *lumped* at the circuit elements drawn there.

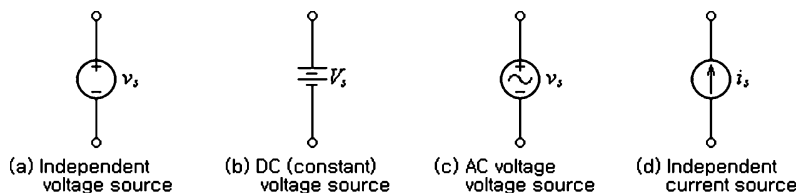


Figure 1.8.1 Symbols for independent sources

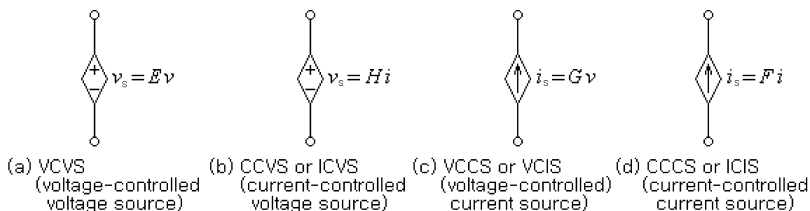
3. If a model is thought to represent a physical system exactly, it must be a blind belief. It is a matter of degree how closely a model describes the behavior of the real system represented by it. From a practical point of view, the time and effort required to obtain solutions are valued above the mathematical rigor of the solution process and the accuracy of the solutions. That is why engineers prefer to have models as simple as possible, requiring less time and effort, as long as they yield solutions with reasonable accuracies. It is not only the analysis but also the physical interpretation that can be made easy with a simple model.
4. Readers should keep in mind that the circuits appearing throughout this book are not real ones, but just circuit diagrams modeling the real circuits, and that models are valid only for certain operation ranges and under some assumptions, specified explicitly or not.

1.3.2 Active Elements

While it is natural to classify the voltage/current sources such as batteries and generators as being *active*, it may seem strange that operational amplifiers (OP Amps) and transistors with no physical source inside themselves are classified as active elements. The justification is that they are always used with power supplies, functioning as dependent voltage/current sources whose values are determined by their own characteristics and the controlling variable, independently of their power sources. That is why they are modeled as dependent sources with no source explicitly shown, as will be introduced in subsequent sections. Figures 1.8.1 and 1.8.2 show the symbols standing for independent and dependent voltage/current sources that will be discussed below.

1.3.2.1 Independent versus Dependent Voltage/Current Sources

If a source maintains a prescribed value with no dependence on voltages or currents elsewhere in the circuit, it is said to be an *independent source*. In contrast, a source whose value depends on the value of a (controlling) voltage or current at some specified location other than itself in the circuit is said to be a dependent or controlled source. A *dependent (controlled) voltage/current source* is referred to as a voltage-controlled or current-controlled source depending on whether the controlling variable is a voltage or a current. As a consequence, there are four types of dependent (controlled) sources: VCVS



Note For the four types of dependent sources, the gains (proportionality coefficients) E , H , G , and F multiplied by the controlling variables are used to denote their PSpice part names, respectively.

Figure 1.8.2 Symbols for dependent (controlled) sources

(voltage-controlled voltage source), VCCS (voltage-controlled current source), CCVS (current-controlled voltage source), and CCCS (current-controlled current source).

Figure 1.8.1 shows the symbols for independent voltage/current sources and Figure 1.8.2 shows the (diamond) symbols for dependent voltage/current sources. Especially, the symbol in Figure 1.8.1(a) can represent both (ideal independent) DC (*direct* or *constant current*) and AC (*alternating* or *sinusoidal current*) voltage sources, while the ones in Figures 1.8.1(b) and (c) can represent only (ideal independent) DC and AC voltage sources, respectively.

1.3.2.2 Ideal versus Practical Source Models

A voltage source is said to be *ideal* if its voltage is not affected by the current flowing through it. Likewise, a current source is said to be *ideal* if its current is not affected by the voltage across it. Although all the sources appearing in the circuit diagrams throughout this book are ideal ones, such sources do not exist in the real world. They are just idealized models of *practical* (*nonideal*) voltage/current sources, whose values are affected more or less by its current/voltage. In the case of the nonideal voltage/current source, which is affected by its current/voltage to such a degree that the ideal source assumption may result in a solution with significant error, a model should be made using an ideal voltage/current source with a resistor in series/parallel, as depicted in Figures 1.9(a) and (b). In such models, the value of a nonideal voltage source is expressed as

$$v = v_s - r_s i \quad \text{with a presumably small series resistance } r_s$$

and the value of a nonideal current source is expressed as

$$i = i_s - \frac{1}{R_p} v \quad \text{with a presumably large parallel resistance } R_p$$

Note the following:

1. The resistance in series/parallel with the ideal voltage/current source can be regarded as modeling internal energy dissipation as well as voltage/current droop of the nonideal source.
2. The practical voltage source with zero series resistance $r_s = 0$ corresponds to an ideal one and the practical current source with infinite parallel resistance $R_p = \infty$ corresponds to an ideal one.

1.3.3 Operational Amplifier

The operational amplifier (OP Amp) is probably the most versatile chip available. It was originally designed to be used for analog computers that perform mathematical operations such as addition,

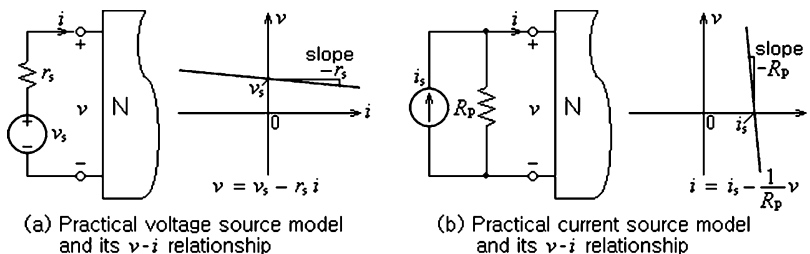


Figure 1.9 Modeling of practical sources

subtraction, integration, differentiation, and so on. As it was widely used for many other applications, early OP Amps constructed from discrete components (such as vacuum tubes and then transistors and resistors) have been superseded by integrated-circuit (IC) OP Amps made of a large number of transistors, resistors, and (sometimes) one capacitor. With only a few external components, it can be made to perform a wide variety of analog signal processing tasks. One of the most common and famous OP Amps is the $\mu A741$, which was introduced by Fairchild in 1968 and is now available at less than a dollar.

Note. A handbook of OP Amp applications is available at focus.ti.com/lit/an/sboa092a/sboa092a.pdf.

The OP Amp usually comes in the form of an eight-pin DIP (dual in-line package) IC as depicted in Figures 1.10(a) and (b). It is basically a differential amplifier having a large open-loop voltage gain A , a very high input impedance R_i , and a low output impedance R_o (*impedance* is a generalized concept of resistance). It has an ‘inverting’ or negative ($-$) input v_- (through pin 2), a ‘noninverting’ or positive ($+$) input v_+ (through pin 3), and a single output v_o available through pin 6. It is powered by a dual-polarity power supply $\pm V_{CC}$ in the range of ± 5 V to ± 15 V through pins 7 and 4. However, it is customary to omit the two power supply pins from the OP Amp symbol as depicted in Figure 1.10(c) since there is no need to be concerned about the power supplies in the circuit analysis.

Its major features are as follows:

1. Its output voltage v_o is A times as large as the differential input voltage ($v_+ - v_-$), which is the difference between the two positive/negative input voltages, with the limitation of the upper/lower bounds $\pm V_{om}$ (slightly smaller than the power supply voltages $\pm V_{CC}$), where the open-loop gain A of an OP Amp is typically in the order of 10^4 – 10^6 . This differential input–output relationship is described by the graph in Figure 1.10(d) and can be written as

$$-V_{om} \leq v_o = A(v_+ - v_-) \leq +V_{om} \quad (1.22)$$

or, more specifically,

$$v_o = \begin{cases} +V_{om} & \text{for } v_+ - v_- > (+V_{om}/A): \text{ nonlinear(saturation) region} \\ A(v_+ - v_-) & \text{for } (-V_{om}/A) \leq v_+ - v_- \leq (+V_{om}/A): \text{ linear region} \\ -V_{om} & \text{for } v_+ - v_- < (-V_{om}/A): \text{ nonlinear(saturation) region} \end{cases} \quad (1.23)$$

2. It has very high input resistance $R_i \approx 2 \text{ M}\Omega$ between the two positive/negative input terminals so that the currents flowing into or out of the input terminals are almost negligible, being normally in the order of μA .
3. It has a very low output resistance $R_o \approx 75 \Omega$ between the output terminal and the ground (see Figure 1.10(e) for a practical OP Amp model).

Referring to Figure 1.10(d), the differential input–output relationship (1.22) or (1.23) indicates that if the magnitude of the differential input voltage $|v_+ - v_-|$ is larger than V_{om}/A , the OP Amp operates in the positive or negative *saturation (nonlinear) region* with the output voltage saturated at $+V_{om}$ or $-V_{om}$; otherwise, it operates in the *linear region* with the output voltage proportional to the differential input as $v_o = A(v_+ - v_-)$. Note that $+V_{om}/-V_{om}$, determining the *output voltage swing*, are called the *positive/negative saturation or maximum output voltages*, respectively.

Here is a question that often arises: What is the condition for an OP Amp to operate in the linear region and how can it be satisfied? The condition is obtained by dividing both sides of Equation (1.22) by the open-loop voltage gain A as

$$-\frac{V_{om}}{A} \leq v_+ - v_- \leq +\frac{V_{om}}{A} \quad (1.24)$$

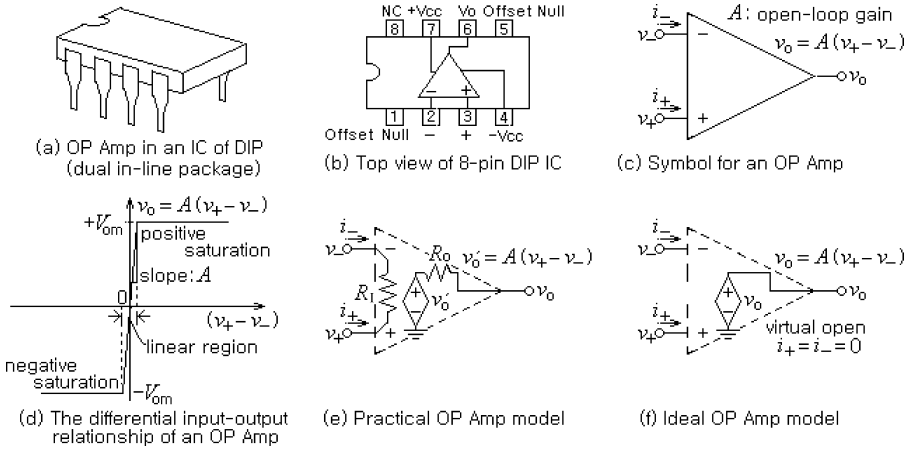


Figure 1.10 Symbol, model, and differential input-output relationship of an OP Amp

This implies that the two voltages at the positive(noninverting)/negative(inverting) input terminals should be ‘almost’ equal, but they must not be ‘exactly’ equal. Why? If the two input voltages are exactly equal so that $v_+ - v_- = 0$, the output voltage will be zero:

$$v_o \stackrel{(1.22)}{=} A(v_+ - v_-) = 0$$

In order for the output voltage v_o to be at some nominal value V_o between $-V_{om}$ and $+V_{om}$, the differential input voltage should be $v_+ - v_- = V_o/A$, which is not exactly zero, but close to zero. Appearing not so easy to satisfy, this condition can be satisfied by simply using a *negative feedback path*, which is a connection between the output terminal and the negative input terminal, where it does not matter whether the connection is simply a shorted path or via any circuit element.

Another question might come to mind: How can the negative feedback make the two voltages at the positive/negative input terminals so close as to satisfy Equation (1.24)? Suppose the differential input $(v_+ - v_-)$ becomes lower/higher than some nominal value close to zero. Then the output voltage v_o determined by Equation (1.22) will be lower/higher than its nominal value V_o , which affects the voltage v_- (at the negative input terminal) via the negative feedback path in such a way that v_- becomes lower/higher than its nominal value. This is expected to make $(v_+ - v_-)$ higher/lower so that the differential input can go back to its nominal value near zero. This so-called *stabilization* effect of negative feedback forms the basis of the *virtual short* or *imaginary short* principle, where the two positive/negative input terminals of an OP Amp circuit with negative feedback are at almost equal voltage levels as if they were shorted in terms of their voltages.

Together with the infinitely large input resistance and negligibly small output resistance of an OP Amp, two important properties of an idealized OP Amp model can be summarized (in Fig. 1.10(f)), which are very useful for the analysis of OP Amp circuits.

[Remark 1.2] Conditions of the Ideal OP Amp Model

1. The input resistance between the two (positive/negative) input terminals and the output resistance between the output terminal and the ground of the ideal OP Amp are assumed to be infinity and zero, respectively. On the assumption of an infinite input resistance, the two input

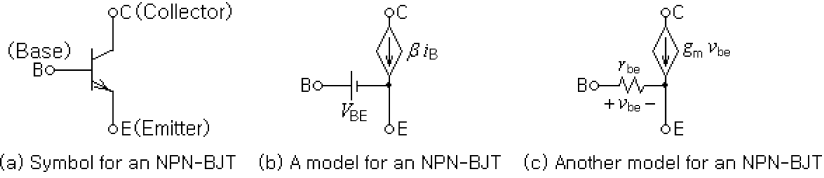


Figure 1.11.1 Symbol and model for an NPN-type BJT (bipolar junction transistor)

terminals can be regarded as being open in terms of current in the sense that no current flows into or out of them, i.e.

$$i_+ = 0 \quad \text{and} \quad i_- = 0 \quad (1.25)$$

which is referred to as the ‘virtual open’ principle.

- The two (positive/negative) input terminals of an OP Amp with a negative feedback path (between the output terminal and the negative input terminal) can be regarded as being short in terms of voltages in the sense that the voltage levels at the two input terminals are almost equal, i.e.

$$v_+ \cong v_- \quad (1.26)$$

which is referred to as the ‘virtual short’ principle. However, this principle may not hold if the OP Amp circuit has also a *positive feedback path*, which is a connection between the output terminal and the positive input terminal. This will be made clear in Section 3.5.2.1, where an OP Amp circuit with positive feedback as well as negative feedback is analyzed in Figure 3.15(a).

1.3.4 Transistor

Figures 1.11.1(a), (b), and (c) show the symbol for an NPN type of BJT (bipolar junction transistor) and two slightly different models of the BJT operating in the forward active region. Figures 1.11.2(a) and (b) show the symbols for the JFET (junction field-effect transistor) and the MOSFET (metal-oxide semiconductor field-effect transistor), respectively, and Figure 1.11.2(c) shows a model for an FET operating in the saturation (constant-current) region.

Note. Refer to Reference [S-1] for details about transistors.

1.4 Kirchhoff's Laws

Kirchhoff's current and voltage laws, each referred to as KCL and KVL, respectively, were introduced by the German physicist Gustav Kirchhoff (1824–1887) in 1845. Together with VCRs (voltage–current relationships) of circuit elements, KCL and KVL enable a set of circuit equations describing the behaviour of a circuit to be formulated in terms of voltages/currents.

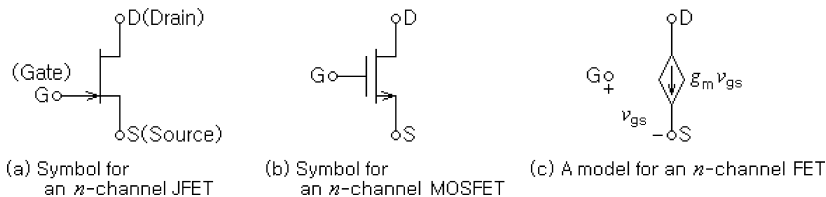


Figure 1.11.2 Symbol and model for an NPN-type BJT (bipolar junction transistor)

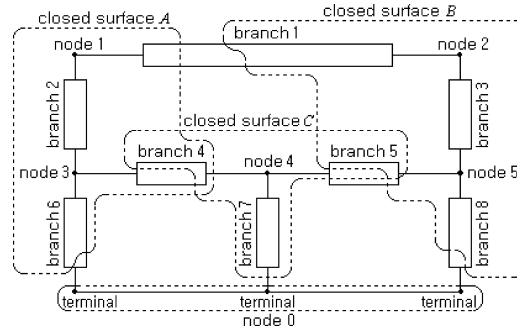


Figure 1.12.1 Branches, nodes, and closed surfaces

1.4.1 Nodes, Branches, and Meshes/Loops

To state and understand KCL and KVL properly, there needs to be an understanding of nodes, branches, and meshes/loops. A *node* or a vertex is a point of connection of two or more circuit elements, which is usually indicated by a dot in a circuit, as illustrated in Figure 1.12.1. It differs from a terminal (connector) in the sense that two or more terminals connected by a short path (a perfect conducting wire) constitute a single node, as illustrated by the bottom node consisting of three terminals. In contrast, a group of neighboring nodes connected by some circuit element(s) amounts to a closed surface. For example, nodes 1 and 3 form the closed surface A and nodes 2 and 5 the closed surface B, while the closed surface C corresponds to node 4 in Figure 1.12.1. If a group of neighboring nodes are connected by only voltage source(s), it is called a *supernode*. A *branch* is a portion of a circuit containing a single element and the nodes at each end of the element, as illustrated in Figures 1.12.1 and 1.12.2. A *mesh* is a closed path that looks like a mesh of a net or a windowpane, as illustrated in Figure 1.12.2. A group of neighboring meshes sharing branch(es) forms a *loop*, which is formally defined to be a closed path through nodes/branches in which no node/branch is encountered more than once. For example, meshes 1, 2, and 3 form the loop A and meshes 1 and 2 form the loop B in Fig. 1.12.2. If a group of neighboring meshes shares current sources pairwise, it is granted the title *supermesh*. A mesh amounts to a special kind of loop containing a single mesh, just as a node corresponds to a special kind of closed surface containing a single node.

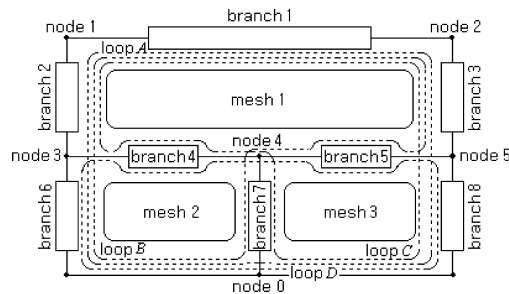


Figure 1.12.2 Branches, meshes, and loops

Note that a set of branches hanging on to a closed surface such as {branches 1,4,6}, {branches 1,5,8}, {branches 4,5,7}, etc., is called a *cutset*, which is defined to be a minimal set of branches that, when cut (removed), will divide the circuit into two separate parts.

1.4.2 Kirchhoff's Current Law (KCL)

KCL states that 'the algebraic sum of all the currents leaving/entering any node or any closed surface is zero'; in other words, the total current entering a node must equal the total current leaving the node. It can be described by

$$\sum_{n=1}^N i_n = 0 \quad (1.27)$$

where N is the number of branches connected to the node and i_n is the n th current leaving (+) or entering (−) the node through branch n . Assigning negative/positive signs to leaving/entering currents results in multiplying both sides of this KCL Equation (1.27) by -1 , making no difference. This law is based on the fact that no charges can simply disappear or become created, so current cannot accumulate any net charge at a node: what goes in must come out instantly.

For example, KCL can be applied to nodes 1, 2, 3, 4, and the closed surface A in the network of Figure 1.13 to write the following equations:

Node 1: $-i_1 - i_2 + i_4 + i_5 = 0$;

$$i_4 + i_5 (\text{the total leaving currents}) = i_1 + i_2 (\text{the total entering currents})$$

Node 2: $-i_3 - i_4 + i_6 + i_7 = 0$;

$$i_6 + i_7 (\text{the total leaving currents}) = i_3 + i_4 (\text{the total entering currents})$$

Node 3: $-i_5 - i_6 + i_8 = 0$; $i_8 (\text{the total leaving currents}) = i_5 + i_6 (\text{the total entering currents})$

Node 4: $-i_7 - i_8 + i_9 = 0$; $i_9 (\text{the total leaving currents}) = i_7 + i_8 (\text{the total entering currents})$

Closed surface A: $-i_3 - i_4 - i_5 + i_9 = 0$;

$$i_9 (\text{the total leaving currents}) = i_3 + i_4 + i_5 (\text{the total entering currents})$$

The following example shows how KCL is applied to the nodes of a circuit.

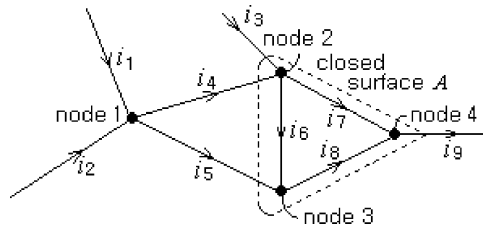


Figure 1.13 Applying KCL to a node or a closed surface

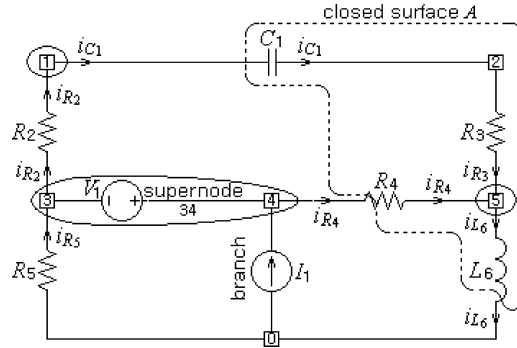


Figure 1.14 Applying KCL to a node, a supernode, and a closed surface

(Example 1.1) Applying KCL to Nodes and Closed Surface

Consider the circuit of Figure 1.14.

- (a) Noting that the nodes are labeled from number 0 to 5 and the nodes 3 and 4 connected by a voltage source constitute a supernode 34, KCL is applied to node 1, node 2, supernode 34, node 5, node 0, and the closed surface A:

$$\text{Node 1: } i_{C_1} - i_{R_2} = 0 \quad (\text{E1.1.1})$$

$$\text{Node 2: } i_{R_3} - i_{C_1} = 0 \quad (\text{E1.1.2})$$

$$\text{Supernode 34: } i_{R_2} + i_{R_4} - i_{R_3} - I_1 = 0 \quad (\text{E1.1.3})$$

$$\text{Node 5: } i_{L_6} - i_{R_3} - i_{R_4} = 0 \quad (\text{E1.1.4})$$

$$\text{Node 0: } i_{R_5} + I_1 - i_{L_6} = 0 \quad (\text{E1.1.5})$$

$$\text{Closed surface A: } -i_{C_1} - i_{R_4} + i_{L_6} = 0 \quad (\text{E1.1.6})$$

- (b) The dependence among the above KCL equations is checked. Adding Equations (E1.1.1) through (E1.1.4) yields Equation (E1.1.5), while adding Equations (E1.1.1), (E1.1.3), and (E1.1.5) yields Equation (E1.1.6). This indicates that only four of the six equations can be independent.

(Question) Can KCL be applied to node 3 or node 4?

(Answer) No, because the current through the voltage source V_1 (connected to those nodes) is not specified in terms of its voltage.

1.4.3 Kirchhoff's Voltage Law (KVL)

KVL states that 'the algebraic sum of all voltage drops/rises (in the clockwise or counterclockwise direction) around any loop (including mesh) is zero'; in other words, the total voltage drops must equal the total voltage rises around a loop. It can be described by

$$\sum_{m=1}^M v_m = 0 \quad (1.28)$$

where M is the number of branch voltages around the loop and v_m is the m th branch voltage falling (+) or rising (−) (in the clockwise or counterclockwise direction) around the loop. Assigning negative/positive signs to voltage drops/rises results in multiplying both sides of this KVL Equation (1.28) by -1 , making

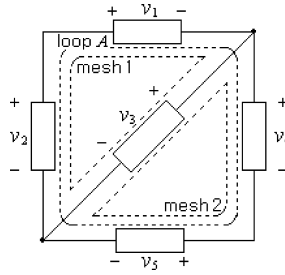


Figure 1.15 Applying KVL to meshes (loops)

no difference. This law is based on the principle of conservation of energy that the total work performed in moving a test charge around a loop is zero.

For example, KVL can be applied to meshes 1, 2, and the loop A in the circuit of Figure 1.15 to write the following equations:

Mesh 1: $v_1 + v_3 - v_2 = 0$; $v_1 + v_3$ (the sum of voltage drops) = v_2 (the sum of voltage rises)

Mesh 2: $-v_3 + v_4 + v_5 = 0$; $v_4 + v_5$ (the sum of voltage drops) = v_3 (the sum of voltage rises)

Loop A: $v_1 + v_4 + v_5 - v_2 = 0$; $v_1 + v_4 + v_5$ (the sum of voltage drops) = v_2 (the sum of voltage rises)

(Example 1.2) Applying KVL to Meshes (Loops)

Consider the circuit of Figure 1.16.

- (a) Noting that the meshes 2 and 3 sharing the current source I_1 constitute a supermesh 23, KVL is applied to mesh 1, supermesh 23, and the loop A:

$$\text{Mesh 1: } v_{C1} + v_{R3} - v_{R4} + V_1 + v_{R2} = 0 \quad (\text{E1.2.1})$$

$$\text{Supermesh 23: } -V_1 + v_{R4} + v_{L6} + v_{R5} = 0 \quad (\text{E1.2.2})$$

$$\text{Loop A: } v_{C1} + v_{R3} + v_{L6} + v_{R5} + v_{R2} = 0 \quad (\text{E1.2.3})$$

- (b) The dependence among the above KCL equations is checked. Adding Equations (E1.2.1) and (E1.2.2) yields Equation (E1.2.3). This indicates that only two of the three equations are independent.

(Question) Can KVL be applied to mesh 2 or mesh 3?

(Answer) No, because the voltage across the current source I_1 (hanging on to those meshes) is not specified in terms of its current.

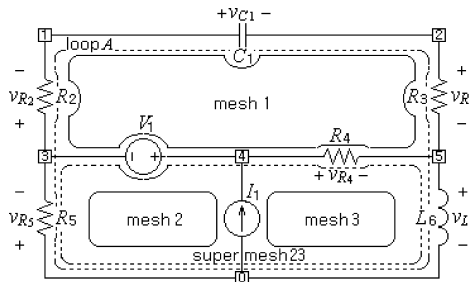


Figure 1.16 Applying KVL to a mesh, a supermesh, and a loop

1.4.4 The Number of KCL/KVL Equations

To find the voltages and currents of a circuit, a set of circuit equations should be set up in the unknown voltages/currents and solved. Aside from how to set it up and solve it, the solvability of a system of circuit equations should be considered. In order for a set of equations to be solved, it should consist of the same number of independent equations as that of unknowns. Thus the independence of circuit equations must be secured as well as the necessary and sufficient number of equations.

Although it may be helpful for understanding some useful theorems on graph theory to see their derivations or proofs, only the basic results about the number of independent KCL/KVL equations are introduced, because they are not only beyond the scope of this book but are also unnecessary for studying the rest of the book. To count the necessary and sufficient number of independent KCL/KVL equations, every source is first removed by short-circuiting voltage sources and open-circuiting current sources, and then the numbers of branches and nodes are denoted by b and n , respectively. Referring to Figures 1.17.1 and 1.17.2, note the following facts:

1. Starting from a single branch having two nodes, one additional node can be covered by connecting one more branch in such a way that no loop is formed.
2. All of the n nodes can be covered by connecting $(n - 1)$ branches in such a way that no loop is formed. Such a set of branches is called a *tree*. The remaining $b - (n - 1)$ branches, called *links* (or *chords*), form the complement set of branches, which is called the *co-tree* (or *chord-set*) with respect to the tree.
3. For every one of the $(n - 1)$ tree branches, a new cutset can be made having the branch exclusively, called a *fundamental cutset*. Applying KCL to every one of the $(n - 1)$ fundamental cutsets yields a set of independent KCL equations.
4. Every time one of the remaining $b - (n - 1)$ branches (called links) is added to a tree, a new loop having the branch exclusively, called a *fundamental loop*, is formed. Applying KVL to every one of the $b - (n - 1)$ fundamental loops yields a set of independent KVL equations.

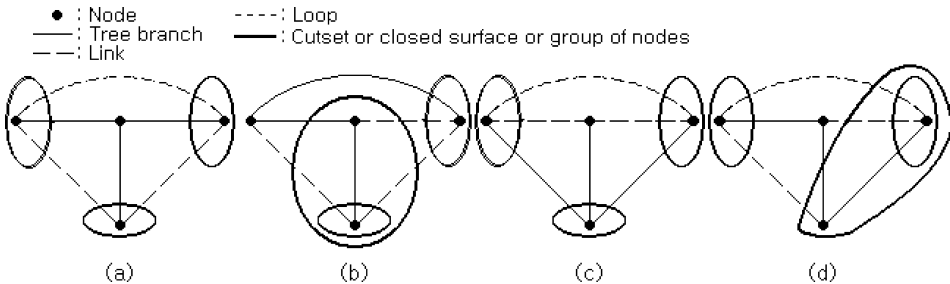


Figure 1.17.1 Example graph to illustrate fundamental cutsets formed in such a way that every cutset has a distinct tree branch exclusively

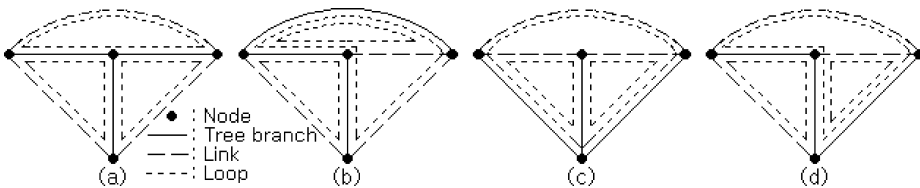


Figure 1.17.2 Example graph to illustrate fundamental loops formed by adding a branch (link) to a set of tree branches

1.5 Equivalent Transformation of Sources

In the process of analyzing circuits, it can be very useful to reduce several sources into an equivalent source and to convert a voltage source into a current source or vice versa.

1.5.1 Combination of Several Sources

1.5.1.1 Series Combination of Voltage Sources

Two or more voltage sources connected in series (as in Figure 1.18.1) can be added algebraically (with their polarities denoted by signs) to yield a single equivalent voltage source.

1.5.1.2 Parallel Combination of Current Sources

Two or more current sources connected in parallel (as in Figure 1.18.2) can be added algebraically (with their directions denoted by signs) to yield a single equivalent current source.

1.5.1.3 Parallel Combination of Voltage Sources

A parallel combination of identical voltage sources (as in Figure 1.18.3) is equivalent to a single voltage source of the same value. This fact presents the basis on which a single voltage source can be converted into multiple ones in parallel for the purpose of transforming a voltage source (with no element in series) into a current source, which is called 'the parallel duplication of the V-source'.

(Question) What about a parallel combination of voltage sources of different values?

(Answer) It is infeasible because it does not satisfy KVL as depicted in Figure 1.18.4. If two batteries of different values, say 3 V and 5 V, happened to be connected in parallel, the difference between the two voltages ($5 - 3 = 2$ V) is applied to the connecting wire of negligibly low resistance r so that an infinitely large circulating current amounting to $2/r$ [A] flows. This situation cannot last due to an infinitely large amount of power dissipation ($4/r$ [W]), resulting in one of two cases: the conducting wire will melt away or the 5 V battery will be exhausted.

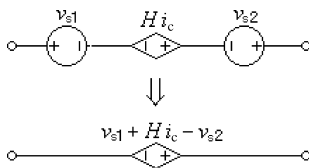


Figure 1.18.1 Voltage sources in series

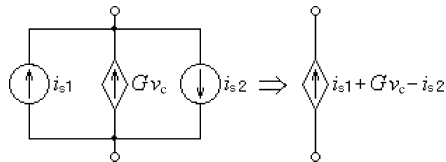


Figure 1.18.2 Current sources in parallel

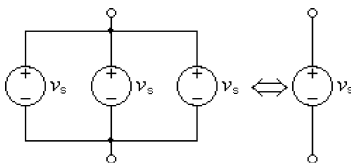


Figure 1.18.3 Voltage sources in parallel

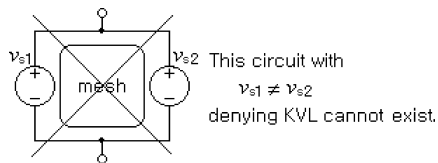


Figure 1.18.4 Infeasible connection of V-sources

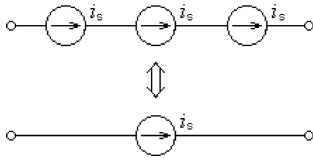


Figure 1.18.5 Current sources in series

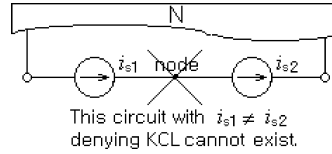


Figure 1.18.6 Infeasible connection of I-sources

1.5.1.4 Series Combination of Current Sources

A series combination of identical current sources (as in Figure 1.18.5) is equivalent to a single current source of the same value. This fact presents the basis on which a single current source can be converted into multiple ones in series for the purpose of transforming a current source (with no element in parallel) into a voltage source, which is called ‘the series duplication of the I-source’.

(Question) What about a series combination of current sources of different values?

(Answer) It is infeasible because it does not satisfy KCL as depicted in Figure 1.18.6.

1.5.1.5 Series Combination of Voltage Source and Current Source

If a voltage source is connected in series with a current source in a branch (as depicted in Figure 1.18.7), it cannot play its inborn role of regulating the voltage of the branch while the current of the branch is determined solely by the current source. Consequently, a voltage source connected in series with a current source can be deactivated or removed (by short-circuiting it) without making any difference in the analysis of the rest of the circuit.

Note. How can the voltage source not be pessimistic after comparing itself (idling around) with the neighboring current source playing its legitimate role? It can be thought of as an exit of the voltage source having no reason for being. As a matter of fact, any element connected in series with a current source has the same destiny of being short-circuited to death for the simplification of circuit analysis. This seems to imply that any being as well as humans needs a *raison d'être*. It is a wish of the authors that every reader of this book will find his/her own role to play in this world in order to be happy (instead of being ignored) and that this book helps even a bit in achieving such an objective.

1.5.1.6 Parallel Combination of Voltage Source and Current Source

If a current source is connected in parallel with a voltage source between two nodes (as depicted in Figure 1.18.8), it cannot play its inborn role of regulating the current between the two nodes while the voltage across the two nodes rests solely on the voltage source. Consequently, a current source connected in parallel with a voltage source can be deactivated or removed (by open-circuiting it) without disturbing the analysis of the rest of the circuit.

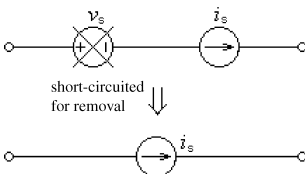


Figure 1.18.7 Voltage and current sources in series

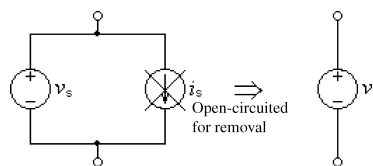


Figure 1.18.8 Voltage and current sources in parallel

Note. It can also be thought of as an exit of the current source having no reason for being. As a matter of fact, any element connected in parallel with a voltage source has the same destiny of being open-circuited (for deactivation) to simplify the circuit analysis.

[Remark 1.3] Element in Series/Parallel with a Current/Voltage Source

Any element connected in series with a current source can be short-circuited and any element in parallel with a voltage source can be open-circuited for removal, without making any difference to the analysis of the rest of the circuit.

[Remark 1.4] Deactivation or Removal of Sources

1. In order to remove (deactivate/paralyze) a voltage source for any reason, it should be made short so that the voltage across its two terminals will be zero.
2. In order to remove (deactivate/paralyze) a current source for any reason, it should be made open so that the current through the branch (having the current source) will be zero.

1.5.2 Voltage–Current Source Transformation

Two electric circuits are said to be *externally equivalent* with respect to a pair of terminals if their terminal voltage–current relationships are identical so that they are indistinguishable from outside. The *source transformation* refers to the conversion of a voltage source in series with an element like a resistor (Figure 1.19(a1)) to a current source in parallel with the element (Figure 1.19(b1)), or vice versa, in such a way that the two circuits are (externally) equivalent w.r.t. their terminal characteristics. What is the relationship among the values of the voltage source V_s , the current source I_s , the series resistor R_s , and the parallel resistor R_p required for the external equivalence of the two source models? To find it out, the voltage–current relationship of each circuit can be written as

$$v = R_s i + V_s \quad (1.29a)$$

$$v = R_p(i + I_s) = R_p i + R_p I_s \quad (1.29b)$$

where the current through R_p is found to be $(i + I_s)$ by applying KCL to the top node of the resistor R_p in Figure 1.19(b1). In order for these two polynomial equations (in i) to be identical for any value of v and i , their coefficients (including the constant term) should be the same:

$$R_p = R_s = R \quad (1.30a)$$

$$V_s = R_p I_s \text{ or, equivalently, } I_s = \frac{V_s}{R_s} \quad (1.30b)$$

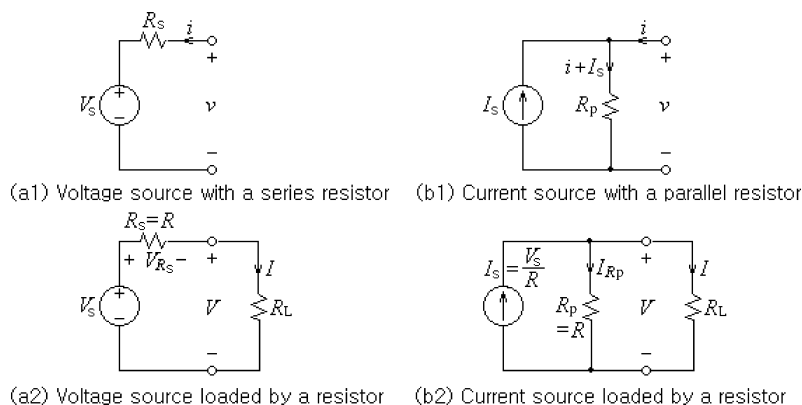


Figure 1.19 Equivalence of voltage and current sources

This *source equivalence condition* is used for a voltage-to-current or current-to-voltage source transformation.

Now, not only to check the validity of the equivalence of the two source models in Figures 1.19(a1) and (b1) but also to realize the meaning of *external equivalence*, a load resistor R_L is connected to their terminals as depicted in Figures 1.19(a2) and (b2) and the current I flowing through R_L is found for each of them:

$$\text{KVL for (a2): } V_{R_s} + V = V_s \xrightarrow{\text{Ohm's law (1.6)}} RI + R_L I = V_s; \quad I = \frac{V_s}{R + R_L} \quad (1.31a)$$

$$\begin{aligned} \text{KCL for (b2): } I_{R_p} + I = I_s &= \frac{V_s}{R} \xrightarrow{\text{Ohm's law (1.6)}} \frac{V}{R} + \frac{V}{R_L} = \frac{V_s}{R}; \quad V = \frac{V_s/R}{1/R + 1/R_L} = \frac{R_L V_s}{R + R_L}; \\ I &= \frac{V}{R_L} = \frac{V_s}{R + R_L} \end{aligned} \quad (1.31b)$$

Since the value of the load resistor R_L is arbitrary, this result indicates that the two source models in Figures 1.19(a1) and (b1) present any resistor with the same voltage and current, reflecting their external equivalence.

One thing to note is that the two externally equivalent source models are not expected to be *internally equivalent*, i.e. equivalent in terms of their internal characteristic. To understand this, the powers dissipated in the two source models are compared as follows:

$$P_a \stackrel{(1.9)}{=} RI^2 \stackrel{(1.31a)}{=} \frac{RV_s^2}{(R + R_L)^2} \neq P_b \stackrel{(1.9)}{=} \frac{V^2}{R} \stackrel{(1.31b)}{=} \frac{R_L^2 V_s^2}{R(R + R_L)^2} \quad (1.32)$$

At this point, it is possible to transform a voltage source with a resistor in series into a current source with the resistor in parallel or vice versa by using Equations (1.30a) and (1.30b). What about a voltage source having no resistor in series and a current source having no resistor in parallel? Some tricks are needed to treat such cases. First the voltage source in Figure 1.20.1(a) is considered, which has no element in series, while having two elements on its left side and three elements on the right side. There are two choices. One is to make two copies of the V_s source as depicted in Figure 1.20.1(b1) and remove the connection between the terminals a and b , knowing that the voltage between the right ends (a and b) of elements A and B and the left ends (c , d , and e) of elements C , D , and E still do not change. This yields two voltage sources each having an element in series, as depicted in Figure 1.20.1(b2). The other is to make three copies of the V_s source as depicted in

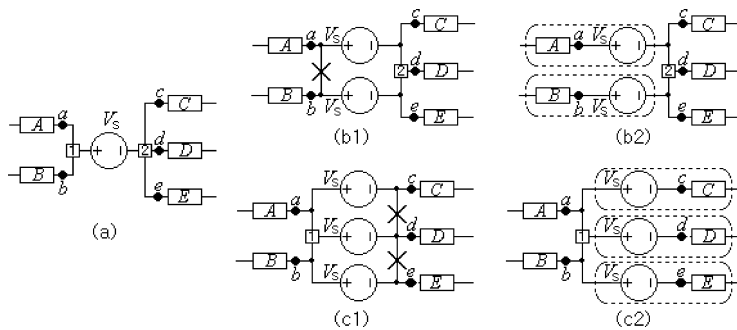


Figure 1.20.1 Parallel duplication of a voltage source for voltage-to-current transformation

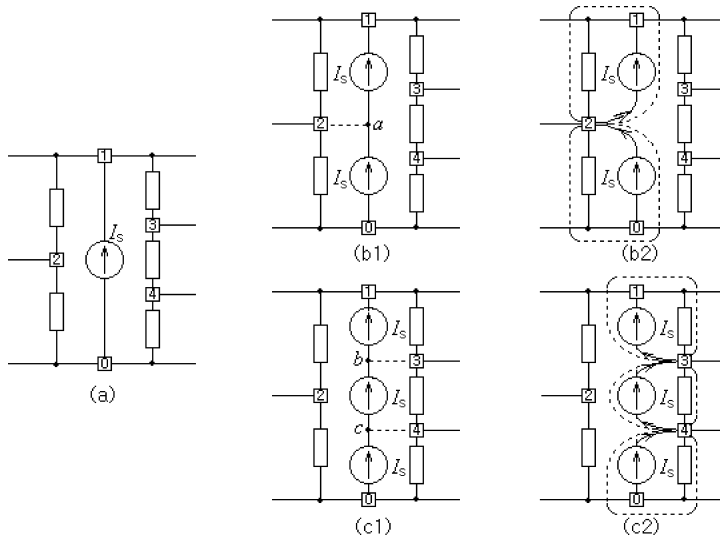


Figure 1.20.2 Series duplication of a current source for current-to-voltage transformation

Figure 1.20.1(c1) and associate each of the three voltage sources with one of the three elements C , D , and E , as depicted in Figure 1.20.1(c2).

Now a current source is considered having no resistor in parallel, such as the one in Figure 1.20.2(a). It has two elements on its left side and three elements on the right side. There are two choices. One is to make two copies of the current source in series, as depicted in Figure 1.20.2(b1), and connect the newly created node a to the left node 2, knowing that it gives a current of I_s to node 2 and get it back instantly, affecting no node. This yields two current sources each having an element in parallel, as depicted in Figure 1.20.2(b2). The other is to make three copies of the current source in series, as depicted in Figure 1.20.2(c1), and associate each of the three current sources with one of the three elements on the right side, as depicted in Figure 1.20.2(c2).

1.5.3 Examples of Source Transformation

To practice source transformation, let us make the source transformations of the three sources in the circuit of Figure 1.21.1.

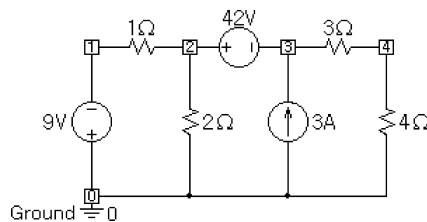


Figure 1.21.1 A circuit

1. The 9V voltage source with the 1Ω resistor in series can easily be transformed into a 9A current source with the 1Ω resistor in parallel.
2. Transforming the 42 V source is not simple since it has no element in series. Thus we need to make use of the parallel duplication of the V-source, which is illustrated in Figure 1.20.1. Thus, noting that there are two elements (the 1Ω resistor and the 2Ω resistor) on its left side, two copies are made of the 42 V source in parallel (Figure 1.21.2(a)). Each of them is associated with one of the two resistors in order to transform them into two current sources, one of 42 A in parallel with the 1Ω resistor and the other of 21 A in parallel with the 2Ω resistor (Figure 1.21.2(b)).
3. An alternative for the 42 V source is to associate each of its two copies with the two elements (the 3Ω resistor and the 3 A source) on its right side (Figure 1.21.3(a)). Then one copy of the 42 V source in series with the 3Ω resistor is transformed into a 14 A source in parallel with the 3Ω resistor, while the other one in series with the 3 A source may well walk out complaining that no role has been given to it (Figure 1.21.3(b)).
4. Transforming the 3 A source into voltage source(s) is also not simple since it has no element in parallel. Thus we need to make use of the series duplication of the I-source, which is illustrated in Figure 1.20.2. Thus, noting that there are two elements (the 3Ω resistor and the 4Ω resistor) on its right side, two copies are made of the 3 A source in series (Figure 1.21.4(a)). Each of them is associated with one of the two resistors in order to transform them into two voltage sources, one of 9 V in series with the 3Ω resistor and the other of 12 V in series with the 4Ω resistor (Figure 1.21.4(b)).

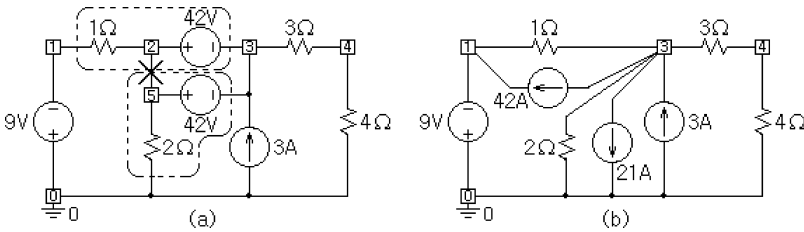


Figure 1.21.2 Voltage-to-current source transformation using parallel duplication of the V-source

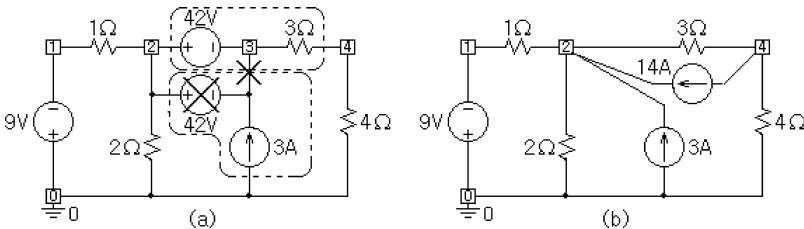


Figure 1.21.3 Voltage-to-current source transformation using parallel duplication of the V-source

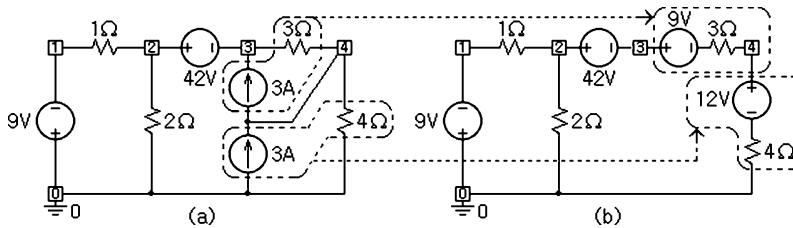


Figure 1.21.4 Current-to-voltage source transformation using series duplication of the I-source

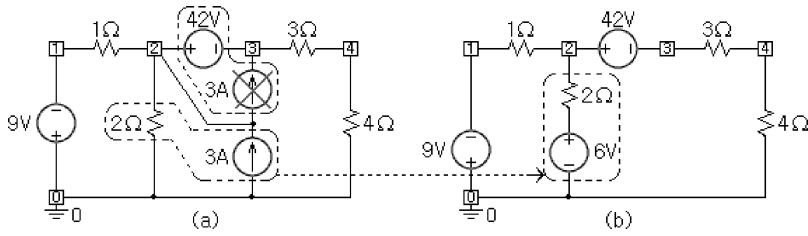


Figure 1.21.5 Current-to-voltage source transformation using series duplication of the I-source

5. An alternative for the 3 A source is to associate each of its two copies with the two elements (the 42 V voltage source and the 2 Ω resistor) on its left side (Figure 1.21.5(a)). Then one copy of the 3 A current source in parallel with the 42 V voltage source may well go away complaining that no role has been given to it, while the other one in parallel with the 2 Ω resistor is transformed into a 6 V source in series with the 2 Ω resistor (Fig. 1.21.5(b)).

Readers who would like to check whether these source transformations are valid and/or to know what they can be used for are invited to solve Problem 1.11.

1.6 Series and Parallel Connections

Although the terms ‘series’ and ‘parallel’ have already been used in the previous sections on the assumption that readers know what they mean, they should be defined formally in order to remove any room for confusion. Two or more circuit elements are said to be connected in *series* if their currents are the same (in view of their reference directions). In order for two elements to be in series, they must have a common (simple) node shared by them exclusively so that any current leaving one element enters the other. Two or more circuit elements are said to be connected in *parallel* if their voltages are the same (in view of their reference polarities). In order for two elements to be in parallel, they must have a common pair of nodes. When mentioning ‘series’ or ‘parallel’, usually it is only the physical connection that is of interest, without minding the reference direction of the current or the reference polarity of the voltage since they are arbitrary.

Especially for series and parallel connections of two resistors R_1 and R_2 , the following formulas are used to obtain their equivalent resistances, which will be derived in Section 2.1:

1. The equivalent resistance of two resistors in series is

$$R_{\text{eq},s} = R_1 + R_2 \quad (1.33a)$$

2. The equivalent resistance of two resistors in parallel is

$$R_{\text{eq},p} = R_1 || R_2 = \frac{1}{1/R_1 + 1/R_2} = \frac{R_1 R_2}{R_1 + R_2} \quad (1.33b)$$

Problems

1.1 Resistance and Resistivity

- Noting that the resistivity of copper is $\rho = 1.68 \times 10^{-8} \Omega \text{m}$, find the resistance of a copper wire that is 1 m long and 1 mm in diameter.
- Find the inductance of an inductor that consists of a coil of insulated wire wound 100 turns around a core having the permeability of $\mu = \mu_r \mu_0 = 1000 \times 4\pi \times 10^{-7} = 1.256 \times 10^{-3} \text{ H/m}$, a diameter of 5 mm, and an average length of 5 cm.

- (c) Find the capacitance of a capacitor consisting of two parallel square metal plates, each $1\text{ cm} \times 1\text{ cm}$ in size, between which a 1 mm -thick ceramic dielectric with a dielectric constant (permittivity) of $\varepsilon = \varepsilon_r \varepsilon_0 = 1000 \times 8.854 \times 10^{-12} = 8.854 \times 10^{-9}\text{ F/m}$ is inserted.

1.2 Permissible Voltage/Current under Power Rating

- (a) Suppose there is an incandescent electric lamp whose rated power and voltage are 100 W and 100 V , respectively. What is the magnitude of the current flowing through it under the rated voltage? Also find its resistance.
- (b) Find the voltage and current that are allowed to be applied continuously for a $1\text{ k}\Omega$ resistor with the power rating of 10 W .

1.3 Load Resistors in Series and Parallel

- (a) Find the powers of the two resistors $1\ \Omega$ and $2\ \Omega$ in series with a 12 V voltage source in the circuit of Figure P1.3(a).

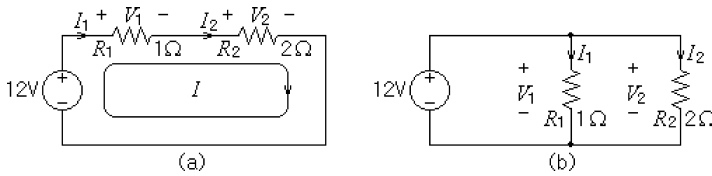


Figure P1.3

- (b) Find the powers of the two resistors $1\ \Omega$ and $2\ \Omega$ in parallel with a 12 V voltage source in the circuit of Figure P1.3(b).
- (c) Circle the correct word in each set of parentheses in the following statement: Combining the results obtained in (a) and (b), it is conjectured that less power is dissipated by (smaller, larger) resistance in a series connection and (larger, smaller) resistance in a parallel connection.

1.4 Sign of Power and Passive Sign Convention

Consider the circuit of Figure P1.4 in which the mesh current i circulating through the mesh is found to be $(12 - 3)/3 = 3\text{ A}$ in its (clockwise) reference direction.

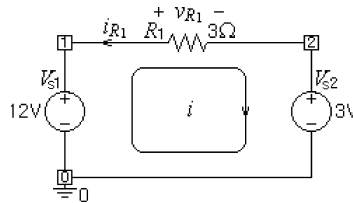


Figure P1.4

- (a) Find the current i_{R1} through the resistor $R1$ with its sign in view of its reference direction depicted in the circuit diagram. Find also the voltage v_{R1} across the resistor $R1$ with its sign in view of its reference polarity.
- (b) Find the power of the $3\ \Omega$ resistor, which should be nonnegative because a resistor can supply no energy even instantly. What is wrong?

- (c) Find the powers of the two voltage sources V_{s1} and V_{s2} and tell whether each of them delivers or absorbs the power.
- (d) Find the total power of the two voltage sources and the $3\ \Omega$ resistor. What does the result imply?

1.5 Ohm's Law, KVL, and KCL for an R - $2R$ Ladder Network

Consider the R - $2R$ ladder circuit of Figure P1.5.

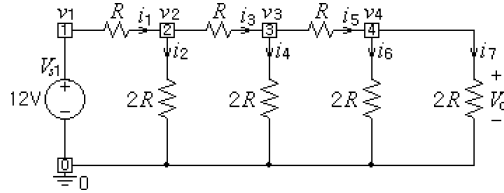


Figure P1.5 R - $2R$ ladder network

- (a) To find the output voltage V_o first, assume it to be known, and then express all the branch currents and node voltages including v_1 in terms of V_o , starting from the branch farthest away from the source, via the repeated use of Ohm's law, KCL, and KVL. Lastly, use $v_1 = 12\text{ V}$ to obtain V_o .
- (b) To find the input current i_1 first, apply the series and parallel combination formulas for resistors to find the overall equivalent resistance of the ladder network seen from the input terminals 1-0. Then find the input current i_1 , the node voltage v_2 , the branch currents i_2 and i_3 , the node voltage v_3 , the branch currents i_4 and i_5 , and finally the output voltage $v_4 = V_o$.

1.6 KVL/KCL, Tellegen's Theorem, Node Voltages, Mesh Currents, and Branch Voltages/Currents

Consider the circuit of Figure P1.6(a) in which the node 0 is grounded so that its node voltage is zero, that is $v_0 = 0$. Note that the grounded node is called the reference node.

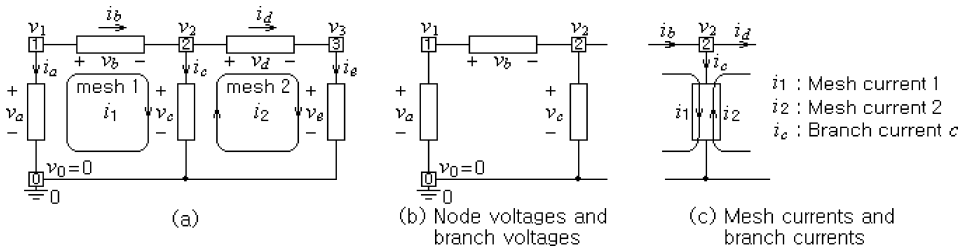


Figure P1.6

- (a) Apply KVL around mesh 1 to write a KVL equation in the branch voltages v_a , v_b , and v_c . Referring to Figure P1.6(b), note that the branch voltages v_a , v_b , and v_c can be expressed in terms of the node voltages v_1 and v_2 as follows:

$$v_a = v_1 - v_0 = v_1 - 0 = v_1, \quad v_b = v_1 - v_2, \quad \text{and} \quad v_c = v_2 - v_0 = v_2 - 0 = v_2 \quad (\text{P1.6.1})$$

Substitute these expressions into the KVL equation for mesh 1 to check whether it is satisfied.

Note. This implies that just expressing branch voltages in terms of node voltages is based on KVL and thus it corresponds to applying KVL implicitly.

- (b) Apply KCL to node 2 to write a KCL equation in the branch currents i_b , i_c , and i_d . Referring to Figure P1.6(c), note that the branch currents i_b , i_c , and i_d can be expressed in terms of the mesh currents i_1 and i_2 as follows:

$$i_b = i_1, \quad i_c = i_1 - i_2, \quad i_d = i_2 \quad (\text{P1.6.2})$$

Substitute these expressions into the KCL equation for node 2 to check whether it is satisfied. *Note.* This implies that just expressing branch currents in terms of mesh currents is based on KCL and thus it corresponds to applying KCL implicitly.

- (c) The total power of all the elements in the circuit can be written as

$$p = v_a i_a + v_b i_b + v_c i_c + v_d i_d + v_e i_e \quad (\text{P1.6.3})$$

Verify that substituting the expressions of the branch voltages in terms of the node voltages into this equation yields

$$p = v_1(i_a + i_b) + v_2(i_c + i_d - i_b) + v_3(i_e - i_d) \quad (\text{P1.6.4})$$

which turns out to be zero with the KCL equation at every node. This implies Tellegen's theorem, described by the following equation:

$$\sum_{k=1}^b p_k = \sum_{k=1}^b v_k i_k = 0, \quad b : \text{ the number of branches} \quad (\text{P1.6.5})$$

1.7 KVL and KCL for a Circuit

Consider the circuit of Figure P1.7 in which the node 0 is grounded so that its node voltage is zero, that is $v_0 = 0$.

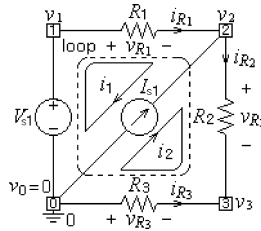


Figure P1.7

- (a) After removing every source by short-circuiting the voltage source(s) and open-circuiting the current source(s), find b , the number of branches, and n , the number of nodes. Referring to Section 1.4.4, find the number of independent KCL and KVL equations.
- (b) Unlike the current through the voltage source V_{s1} , the currents through the resistors R_1 , R_2 , and R_3 can be expressed in terms of the node voltages $v_1 = V_{s1}$, v_2 , and v_3 based on Ohm's law as

$$i_{R1} = \frac{v_1 - v_2}{R_1} = \frac{V_{s1} - v_2}{R_1}, \quad i_{R2} = \frac{v_2 - v_3}{R_2}, \quad \text{and} \quad i_{R3} = \frac{v_0 - v_3}{R_3} = -\frac{v_3}{R_3} \quad (\text{P1.7.1})$$

Find the appropriate nodes (including any supernode) to which KCL can be applied. Write the KCL equations in the branch currents i_{R_1} , i_{R_2} , and i_{R_3} at those nodes. Substitute the above VCRs (voltage–current relationships) of the branch elements into the independent KCL equation(s) to find the node equations in the node voltages v_2 and v_3 . Are there as many independent KCL equations as estimated in (a)?

- (c) Unlike the voltage across the current source I_{s1} , the voltages across the resistors R_1 , R_2 , and R_3 can be expressed in terms of the mesh currents i_1 and i_2 based on Ohm's law as

$$v_{R_1} = R_1 i_{R_1} = R_1 i_1, \quad v_{R_2} = R_2 i_{R_2} = R_2 i_2, \quad \text{and} \quad v_{R_3} = R_3 i_{R_3} = -R_3 i_2 \quad (\text{P1.7.2})$$

Find the appropriate mesh(es) or loop(s) around which KVL can be applied. Write the KVL equation(s) in the branch voltages v_{R_1} , v_{R_2} , and v_{R_3} around the mesh(es) and/or loop(s). Is there any loop that can be called a supermesh? Substitute the above VCRs (voltage–current relationships) of the branch elements into the KVL equation(s) to find the mesh equations in i_1 and i_2 . Are there as many independent KVL equations as estimated in (a)? Is the number of independent KVL equations the same as that of the unknown mesh currents, thus enabling the KVL equation(s) to be solved for i_1 and i_2 ? If not, write another equation $i_2 - i_1 = I_{s1}$, which is presented by the current source I_{s1} involved in a supermesh, i.e. shared by the two meshes and playing the role of matchmaker to relate the two mesh currents i_1 and i_2 .

1.8 KVL and KCL for a Circuit

Consider the circuit of Figure P1.8.

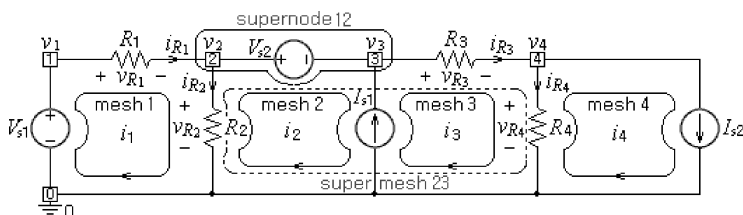


Figure P1.8

- After removing every source by short-circuiting the voltage source(s) and open-circuiting the current source(s), find the number of branches, b , and the number of nodes, n . Referring to Section 1.4.4, find the number of independent KCL and KVL equations.
- Apply KCL to the appropriate nodes (possibly including any supernode) to write as many independent KCL equations as possible in the three unknown node voltages v_2 , v_3 , and v_4 , where $v_1 = V_{s1}$ is already known. Is there the same number of KCL equations as estimated in (a)? Is the number of independent KCL equations the same as that of the unknown node voltages, thus enabling the KCL equation(s) to be solved for v_2 , v_3 , and v_4 ? If not, use another equation $v_2 - v_3 = V_{s2}$, or $v_2 = v_3 + V_{s2}$, which is presented by the voltage source V_{s2} involved in a supernode, i.e. shared by the two nodes and playing the role of matchmaker to relate the two node voltages v_1 and v_2 .
- Apply KVL to the appropriate meshes or loops (possibly including any supermesh) to write as many independent KVL equations in the three unknown mesh currents i_1 , i_2 , and i_3 , where

$i_4 = I_{s2}$ is already known. Is this the same number of KVL equations as estimated in (a)? Is the number of independent KCL equations the same as that of the unknown mesh currents, thus enabling the KVL equation(s) to be solved for i_1 , i_2 , and i_3 ? If not, use another equation $i_3 - i_2 = I_{s1}$, or $i_3 = i_2 + I_{s1}$, which is presented by the current source I_{s1} involved in a supermesh, i.e. shared by the two meshes and playing the role of matchmaker to relate i_2 and i_3 .

- (d) With $R_1 = 1\ \Omega$, $R_2 = 2\ \Omega$, $R_3 = 3\ \Omega$, $R_4 = 4\ \Omega$, $V_{s1} = 12\text{ V}$, $V_{s2} = 6\text{ V}$, $I_{s1} = 20\text{ A}$, and $I_{s2} = 23\text{ A}$, solve the set of KCL equations for v_2 , v_3 , and v_4 and use the solution to find i_{R_1} , i_{R_2} , i_{R_3} , and i_{R_4} . Also solve the set of KVL equations for i_1 , i_2 , and i_3 and use the solution to find i_{R_1} , i_{R_2} , i_{R_3} , and i_{R_4} . Do the two solutions agree with each other?

1.9 KCL, KVL, and the Source Combination

Consider the circuit of Figure P1.9.

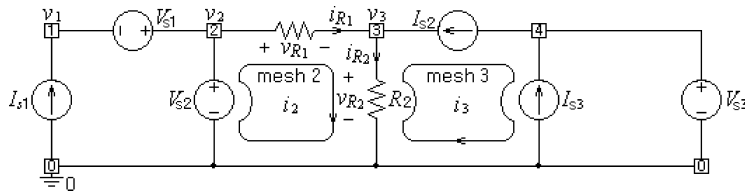


Figure P1.9

- Applying the rules of combining the voltage source and current source described in Figures 1.18.7 and 1.18.8, remove the sources that are dispensable for finding the current i_{R_2} through R_2 .
- Apply KCL to node 3 to write a KCL equation in i_{R_1} and i_{R_2} , substitute the expressions of the branch currents i_{R_1} and i_{R_2} in terms of the node voltage v_3 into the KCL equation, and solve it for v_3 . Use the result to find the current i_{R_2} through the resistor R_2 .
- Apply KVL to mesh 2 to write a KVL equation in v_{R_1} and v_{R_2} , substitute the expressions of the branch voltages v_{R_1} and v_{R_2} in terms of the mesh currents i_2 and i_3 into the KVL equation, substitute $i_3 = -I_{s2}$, and solve it for i_2 . Use the result to find the current i_{R_2} through the resistor R_2 .

1.10 Source Transformation and Equivalent Circuit for Parallel Voltage Sources with a Resistor in Series

Consider the circuit of Figure P1.10(a) in which two voltage sources each having a resistor in series are connected in parallel and applied to a load resistor R_L .

- Apply KCL to node 1 to write a KCL equation in i_{R_1} , i_{R_2} , and i_{R_L} , substitute the expressions of the branch currents i_{R_1} , i_{R_2} , and i_{R_L} in terms of the node voltage v_1 into the KCL equation, and solve it for v_1 . Find the current i_{R_L} through R_L .
- Referring to the source transformation introduced in Section 1.5.2, transform each of the two voltage sources (with a resistor in series) into a current source (with a resistor in parallel), as shown in Figure P1.10(b). Then combine the two parallel current sources into an equivalent one (referring to Section 1.5.1) and the two parallel resistors into an equivalent one (referring to Section 1.6) so that an equivalent of the source part of the original circuit seen from the terminals 1–0 is obtained, as shown in Figure P1.10(c). Express the values of the equivalent voltage source V_s and the equivalent resistor R in terms of V_{s1} , V_{s2} , R_1 , and R_2 . Using this equivalent circuit, find the current i_{R_L} through R_L .

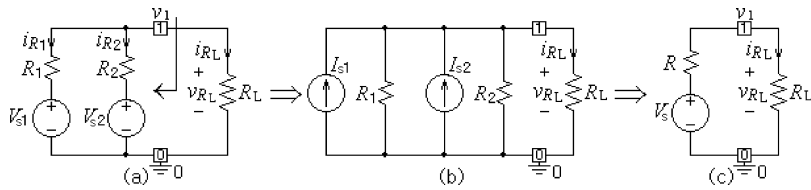


Figure P1.10 Simplification of the circuit using the source transformation

1.11 Simplification of a Circuit by Source Transformation

To see how source transformation can be used to simplify circuits, the source transformation technique will be applied successively for the circuits that were obtained in Figures 1.21.2 to 1.21.5.

- (a) Figure P1.11.1(a) shows the circuit in Figure 1.21.2(b). The 42A source with the 1Ω resistor in parallel can be transformed into a 42V source in series with the 1Ω resistor, combined with the 9V source to make a 51V source, as in Figure P1.11.1(b), and transformed into a 51A source with the 1Ω resistor in parallel, as in Figure P1.11.1(c). Lastly, the two resistors (of 1Ω and 2Ω) and the three current sources (of 51A, 21A, and 3A) in parallel can be transformed as a whole into a () V-source in series with a () Ω -resistor, as depicted in Figure P1.11.1(d).

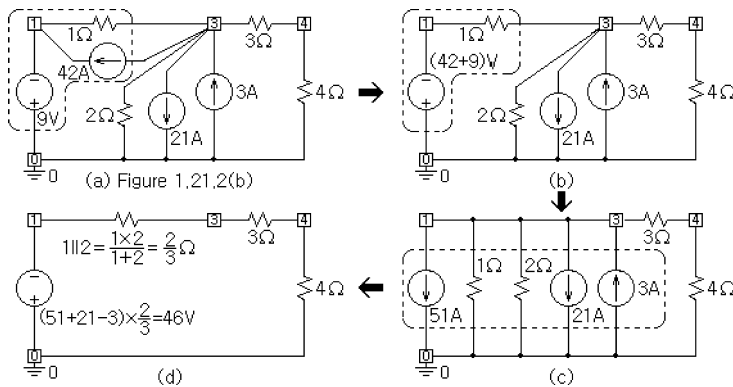


Figure P1.11.1

- (b) Figure P1.11.2(a) shows the circuit in Fig. 1.21.3(b). The 9V source with the 1Ω resistor in series can be transformed into a 9A source in parallel with the 1Ω resistor (Figure P1.11.2(b)) and combined with the 3A source and the 2Ω resistor in parallel to make a () A-source in parallel with a () Ω resistor, as in Fig. P1.11.2(c). Lastly, the 6A source with the $(2/3)\Omega$ resistor in parallel can be transformed into a 4V source in series with the $(2/3)\Omega$ resistor and then combined with the 42V source in series, as depicted in Figure P1.11.2(d).
- (c) Figure P1.11.3(a) shows the circuit in Figure 1.21.4(b). The 9V source with the 1Ω resistor in series can be transformed into a 9A source in parallel with the 1Ω resistor, the three voltage sources of 42V, 9V, and 12V are combined into a 63V source, as in Fig. P1.11.3(b), and then the two resistors (1Ω and 2Ω) in parallel are combined to make one resistor of $(2/3)\Omega$, as in Figure P1.11.3(c). Lastly, the 9A source in parallel with the $(2/3)\Omega$ resistor can be transformed into a () V-source in series with the () Ω resistor and then combined with the 63V source in series as depicted in Figure P1.11.3(d).

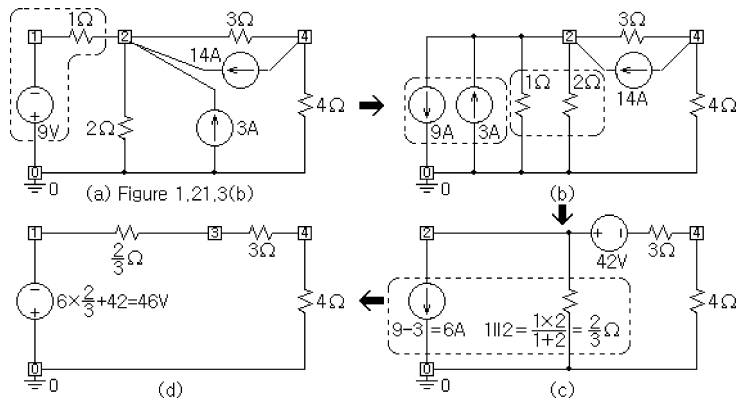


Figure P1.11.2

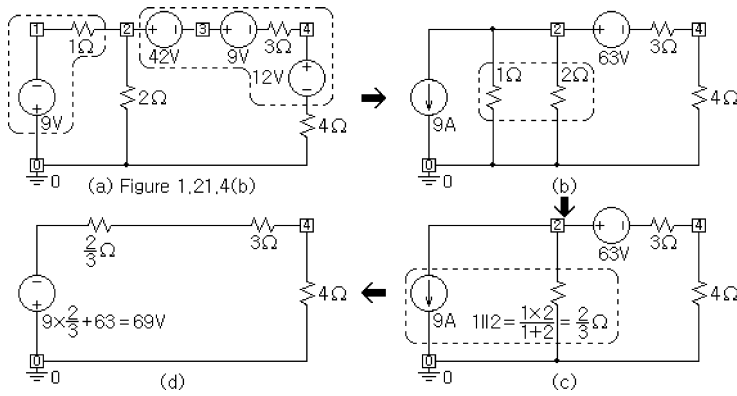


Figure P1.11.3

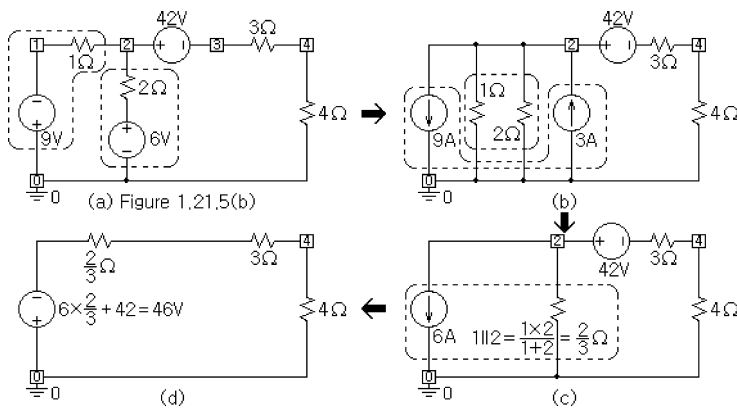


Figure P1.11.4

- (d) Figure P1.11.4(a) shows the circuit in Figure 1.21.5(b). The 9V source with the 1Ω resistor in series can be transformed into a 9A source in parallel with the 1Ω resistor, and the 6V source with the 2Ω resistor in series into a 3A source in parallel with the 2Ω resistor (Figure P1.11.4(b)). Then the two current sources of 9A and 3A in parallel and two resistors of 1Ω and 2Ω in parallel can be combined to make a ()A-source in parallel with a () Ω -resistor, as in Figure P1.11.4(c). Lastly, the 6A source with the $(2/3)\Omega$ resistor in parallel can be transformed into a 4V source in series with the $(2/3)\Omega$ resistor and then combined with the 42V source in series, as depicted in Figure P1.11.4(d).
- (e) Simplify the circuit in Figure 1.21.1 by starting from the transformation of the 9V source in series with the 1Ω resistor into a 9A source in parallel with the 1Ω resistor.
- (f) Let the objective be to find the current through the 4Ω resistor. Among the equivalent circuits obtained in (a), (b), (c), (d), and (e), find one that does not serve the purpose and explain the reason why it does not.

Note. This problem implies that it is better not to touch the target element (whose voltage or current we are interested in) when using the equivalence to simplify a circuit.

2

Resistor Circuits

The analysis of a circuit refers to the process of solving for the voltages and currents present in the circuit. This chapter introduces two major methods for systematic circuit analysis: mesh analysis and node analysis. Mesh analysis sets up the circuit equation(s) to solve for the mesh current variable(s) and node analysis sets up the circuit equation(s) to solve for the node voltage variable(s). How to compare these two contrasting methods in terms of the number of equations/unknowns will also be introduced. These methods can be applied not only for resistor circuits in this chapter but also for circuits containing inductors/capacitors in subsequent chapters. Resulting circuit equations normally constitute a set of simultaneous linear equations, which can easily be solved by using MATLAB or an equivalent on a computer.

In the framework of the two general circuit analysis methods, this chapter discusses series and parallel combinations of several resistors, Δ -Y(Π -T) and Y- Δ (T- Π) conversion formulas, the Thevenin and Norton equivalent circuits, OP amp circuits with negative/positive feedback, and the concept of linearity and loading effect, which are very useful for analyzing circuits.

Many examples and problems will be worked on throughout this book, which will make the usefulness of circuit analysis and design obvious. Engineers always should try to find a solution of practical accuracy with as small amount of time and effort as possible, and further ensure that the solution is valid as well as correct, or at least makes sense. Engineers are also expected to be able to explain how they reached the solution and what physical interpretation can be put on the solution.

2.1 Combination of Resistors

2.1.1 Series Combination of Resistors

Figure 2.1(a) shows a series connection of N resistors each of resistance R_n . Since all the resistors in series have the same current $i_1 = i_2 = \dots = i_N = i$, the voltage of each resistor can be written as

$$v_1 = R_1 i, \quad v_2 = R_2 i, \quad \dots, \quad v_N = R_N i$$

and the sum of all the resistor voltages makes the overall voltage as

$$v = \sum_{n=1}^N v_n = \sum_{n=1}^N R_n i = \left(\sum_{n=1}^N R_n \right) i = R_S i$$

This implies that the equivalent resistance of N resistors in series is the sum of all the individual resistances:

$$R_S = \sum_{n=1}^N R_n \quad (2.1)$$

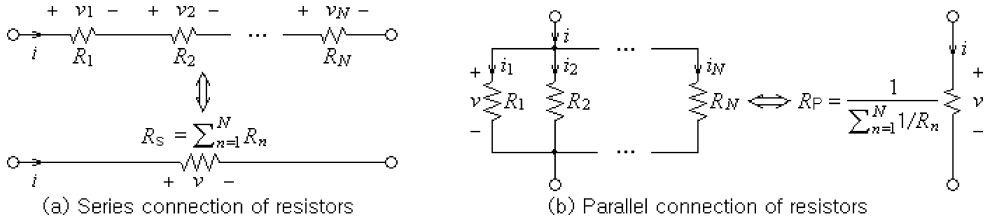


Figure 2.1 Series/parallel connection of resistors

Note. The series combination of resistors yields the equivalent resistance, which is larger than any one of the resistances.

2.1.2 Parallel Combination of Resistors

Figure 2.1(b) shows a parallel connection of N resistors each of resistance R_n , or, equivalently, of conductance $G_n = 1/R_n$. Since all the resistors in parallel have the same voltage $v_1 = \dots = v_N = v$, the current of each resistor can be written as

$$i_1 = G_1 v, \quad i_2 = G_2 v, \quad \dots, \quad i_N = G_N v$$

and the sum of all the resistor currents makes the overall current as

$$i = \sum_{n=1}^N i_n = \sum_{n=1}^N G_n v = \left(\sum_{n=1}^N G_n \right) v = G_P v$$

This implies that the equivalent conductance of N resistors in parallel is the sum of all the individual conductances:

$$G_P = \sum_{n=1}^N G_n, \quad \frac{1}{R_P} = \sum_{n=1}^N \frac{1}{R_n} \quad (2.2)$$

Note. The parallel combination of resistors yields the equivalent resistance, which is smaller than any one of the resistances.

The formula (2.2) for computing the equivalent resistance of a parallel combination of resistors can be cast into the following MATLAB routine `parallel_comb()`:

```
function Rp=parallel_comb(Rs)
Rp = 1/sum (1./Rs); % The parallel combination of resistors by Eq. (2.2)
```

In particular, the parallel combination of two resistors has the equivalent conductance and resistance as

$$G = G_1 + G_2 = \frac{1}{R_1} + \frac{1}{R_2} = \frac{R_1 + R_2}{R_1 R_2}, \quad R = R_1 || R_2 = \frac{1}{G} = \frac{R_1 R_2}{R_1 + R_2} \quad (2.3)$$

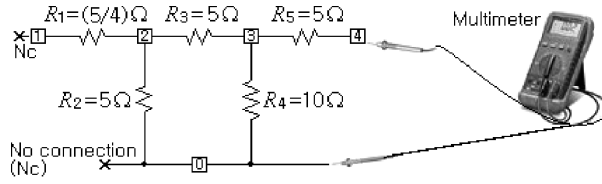


Figure 2.2 An example of resistors in a series–parallel connection

(Example 2.1) Series/Parallel Combination of Resistors

Find the resistance of the circuit of Figure 2.2 seen from the terminal 4–0 by using the series and parallel resistance combination formulas (2.1) and (2.2)/(2.3):

$$\begin{aligned} R_{23} &= R_2 + R_3 = 5 + 5 = 10\ \Omega \\ R_{234} &= R_{23} \parallel R_4 = \frac{R_{23}R_4}{R_{23} + R_4} = \frac{10 \times 10}{10 + 10} = 5\ \Omega \\ R_{4-0} &= R_{234} + R_5 = 5 + 5 = 10\ \Omega \end{aligned}$$

2.2 Voltage/Current Divider

2.2.1 Voltage Divider

Figure 2.3(a) shows a circuit consisting of two resistors R_1 and R_2 in series. Since KVL applied around the left loop yields

$$-v + v_1 + v_2 \stackrel{(1.6a)}{=} -v + R_1 i_1 + R_2 i_2 \stackrel{\text{KVL}}{=} 0 \quad \begin{matrix} i_1 = i_2 = i \\ \text{series} \end{matrix} \quad R_1 i + R_2 i = v; \quad i = \frac{v}{R_1 + R_2}$$

the voltages across R_1 and R_2 can be obtained as

$$v_1 \stackrel{(1.6a)}{=} R_1 i = \frac{R_1}{R_1 + R_2} v \quad \text{and} \quad v_2 \stackrel{(1.6a)}{=} R_2 i = \frac{R_2}{R_1 + R_2} v \quad (2.4)$$

This corresponds to the division of the overall input voltage v in proportion to the resistance. Therefore a network consisting of resistors in series is referred to as a *voltage divider* or voltage attenuator.

One thing to note about the voltage divider is that if a load R_L is connected in parallel with one of the two resistors, say R_2 , as depicted in Figure 2.3(b), then the voltage gain, i.e. the ratio of the (output) voltage across the parallel resistors ($R_2 \parallel R_L$) to the overall input voltage decreases as

$$\frac{v_2}{v} = \frac{R_2 \parallel R_L}{R_1 + (R_2 \parallel R_L)} < \frac{R_2}{R_1 + R_2} \quad (\because R_2 \parallel R_L < R_2) \quad (2.5)$$

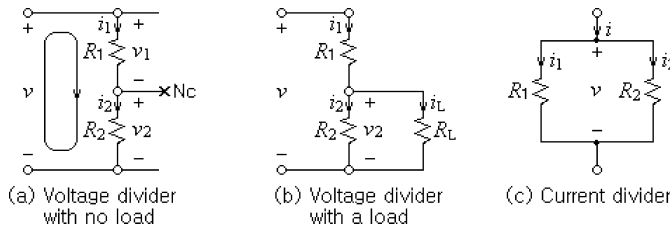


Figure 2.3 Voltage divider and current divider

which is smaller than that with no load. The load current i_L adds to the current flowing through R_1 so that the voltage drop across R_1 becomes larger, which is responsible for the reduced output voltage. The reduction of the output due to the load as in this case is referred to as the *loading effect*.

2.2.2 Current Divider

Figure 2.3(c) shows a circuit consisting of two resistors R_1 and R_2 in parallel. Since KCL applied to the top node yields

$$-i + i_1 + i_2 \stackrel{(1.6b)}{=} -i + G_1 v_1 + G_2 v_2 \stackrel{\text{KCL}}{=} 0 \xrightarrow[\text{parallel}]{v_1 = v_2 = v} G_1 v + G_2 v = i; \quad v = \frac{i}{G_1 + G_2}$$

the currents through R_1 and R_2 can be obtained as

$$i_1 \stackrel{(1.6b)}{=} G_1 v = \frac{G_1}{G_1 + G_2} i = \frac{1/R_1}{1/R_1 + 1/R_2} i = \frac{R_2}{R_1 + R_2} i \quad \text{and} \quad i_2 \stackrel{(1.6b)}{=} G_2 v = \frac{R_1}{R_1 + R_2} i \quad (2.6)$$

The current gain, i.e. the ratio of the current through one of the two parallel resistors, say R_2 , to the overall input current turns out to be

$$\frac{i_2}{i} = \frac{R_1}{R_1 + R_2}$$

This corresponds to the division of the overall input current i in inverse proportion to the resistance and thus a network consisting of resistors in parallel is referred to as a *current divider*.

2.3 Δ -Y(Π -T) Transformation

Figures 2.4(a) and (b) show the Δ - and Y-connections, respectively, which cannot really be dealt with as series or parallel combinations. When a Δ - or Y-connection is encountered in a circuit, it may be helpful in reducing the complexity of the circuit to convert it to the equivalent Y- or Δ -connection. Like the equivalence between a voltage source and a current source discussed in Section 1.5.2, a Δ -connection and a Y-connection are said to be *equivalent* if they exhibit the same terminal behavior.

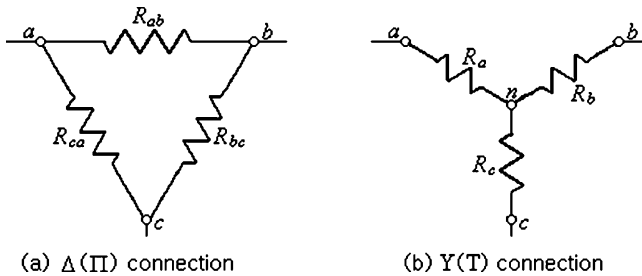


Figure 2.4 Δ (Π) connection and Y(T) connection

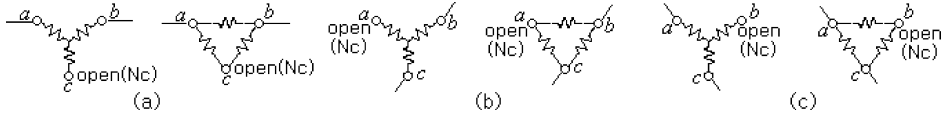


Figure 2.5.1 Conditions for equivalence of a Δ(Π)-connection and a Y(T)-connection

2.3.1 Δ-Y (Π-T) Conversion Formula

With one of the three terminals a , b , and c opened (connected with no external element), as depicted in Figure 2.5.1(a), (b), and (c), the resistances between the other two terminals of the Δ- and Y-connections can be equated to write

$$R_a + R_b = R_{ab} || (R_{bc} + R_{ca}) = \frac{R_{ab}(R_{bc} + R_{ca})}{R_{ab} + R_{bc} + R_{ca}}$$

$$R_b + R_c = R_{bc} || (R_{ca} + R_{ab}) = \frac{R_{bc}(R_{ca} + R_{ab})}{R_{ab} + R_{bc} + R_{ca}}$$

$$R_c + R_a = R_{ca} || (R_{ab} + R_{bc}) = \frac{R_{ca}(R_{ab} + R_{bc})}{R_{ab} + R_{bc} + R_{ca}}$$

Dividing the summation of these three equations by two yields

$$R_a + R_b + R_c = \frac{R_{ab} R_{bc} + R_{bc} R_{ca} + R_{ca} R_{ab}}{R_{ab} + R_{bc} + R_{ca}}$$

Each of the above three equations can be subtracted from this to get the Δ-Y conversion formulas summarized in Table 2.1.

2.3.2 Y-Δ (T-Π) Conversion Formula

With two of the three terminals a , b , and c shorted (connected with each other), as depicted in Figures 2.5.2(a), (b), and (c), the resistances between the common terminal and the other one of the Δ- and Y-connections can be equated to write

$$\frac{R_{ab} R_{bc}}{R_{ab} + R_{bc}} = \frac{R_c R_a}{R_c + R_a} + R_b = \frac{R_a R_b + R_b R_c + R_c R_a}{R_c + R_a}$$

$$\frac{R_{bc} R_{ca}}{R_{bc} + R_{ca}} = \frac{R_a R_b}{R_a + R_b} + R_c = \frac{R_a R_b + R_b R_c + R_c R_a}{R_a + R_b}$$

$$\frac{R_{ca} R_{ab}}{R_{ca} + R_{ab}} = \frac{R_b R_c}{R_b + R_c} + R_a = \frac{R_a R_b + R_b R_c + R_c R_a}{R_b + R_c}$$

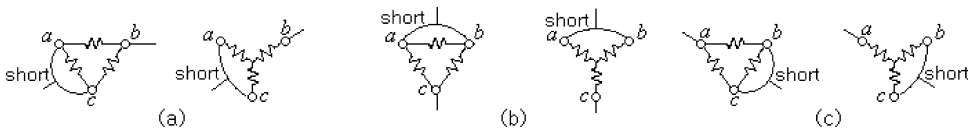


Figure 2.5.2 Conditions for equivalence of a Δ(Π)-connection and a Y(T)-connection

Table 2.1 Δ -Y(II-T) and Y- Δ (T-II) conversion formulas

Δ -Y conversion formulas		Y- Δ conversion formulas	
$R_a = \frac{R_{ca}R_{ab}}{R_{ab} + R_{bc} + R_{ca}}$	(2.7a)	$R_{ab} = \frac{R_a R_b + R_b R_c + R_c R_a}{R_c}$	(2.8a)
$R_b = \frac{R_{ab}R_{bc}}{R_{ab} + R_{bc} + R_{ca}}$	(2.7b)	$R_{bc} = \frac{R_a R_b + R_b R_c + R_c R_a}{R_a}$	(2.8b)
$R_c = \frac{R_{bc}R_{ca}}{R_{ab} + R_{bc} + R_{ca}}$	(2.7c)	$R_{ca} = \frac{R_a R_b + R_b R_c + R_c R_a}{R_b}$	(2.8c)

Taking the reciprocals of these three equations yields

$$\begin{aligned} \frac{1}{R_{ab}} + \frac{1}{R_{bc}} &= \frac{R_c + R_a}{R_a R_b + R_b R_c + R_c R_a} \\ \frac{1}{R_{bc}} + \frac{1}{R_{ca}} &= \frac{R_a + R_b}{R_a R_b + R_b R_c + R_c R_a} \\ \frac{1}{R_{ca}} + \frac{1}{R_{ab}} &= \frac{R_b + R_c}{R_a R_b + R_b R_c + R_c R_a} \end{aligned}$$

Dividing the summation of these three equations by two yields

$$\frac{1}{R_{ab}} + \frac{1}{R_{bc}} + \frac{1}{R_{ca}} = \frac{R_a + R_b + R_c}{R_a R_b + R_b R_c + R_c R_a}$$

Each of the above three equations can be subtracted from this to get the Y- Δ conversion formulas summarized in Table 2.1.

2.4 Node Analysis

Two systematic methods are available to solve circuits, i.e. to find all branch voltages and currents of the circuits: the node analysis method, discussed in this section, and the mesh analysis method, discussed in the next section.

In the node analysis, one node is selected to be a *reference* or *ground node*, regarding the voltage at that node as zero (grounded). Then we apply KCL to the nodes to write a set of node equations and solve it for the unknown node voltages of the nonreference nodes with respect to the reference node. The node analysis procedure of setting up the node equations for the circuit in Figure 2.6 consists of the following steps:

Step 0. Select one node as the reference node and label it 0, assuming its node voltage to be zero. (The reference node is usually chosen to be the node with the most connections, like the bottom node

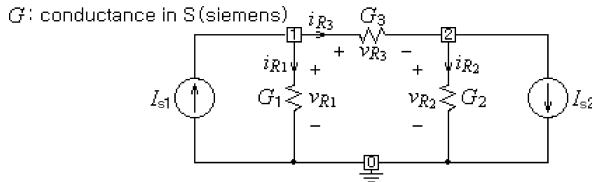


Figure 2.6 A circuit with independent voltage sources

of the circuit in Figure 2.6. This may help reduce the complexity of the node equation.) Then label all the nonreference nodes, say the numbers 1, 2, ..., that correspond to assigning the node voltage variable $v_1, v_2, \dots$ to each node, respectively.

Step 1. Based on the VCR (voltage-current relationship), express the branch current of every element except sources in terms of the node voltages (complying with the passive sign convention):

$$i_{R_1} = G_1 v_{R_1} = G_1 v_1, \quad i_{R_2} = G_2 v_{R_2} = G_2 v_2, \quad i_{R_{12}} = G_{12} v_{R_{12}} = G_{12} (v_1 - v_2)$$

Step 2. Apply KCL to the nonreference nodes to write the node equations with the branch current terms on the left-hand side and the source terms on the right-hand side:

$$\begin{aligned} i_{R_1} + i_{R_{12}} &= I_{s1} \\ -i_{R_{12}} + i_{R_2} &= -I_{s2} \end{aligned}$$

Step 3. Substitute the VCR equations (obtained in Step 1) into the KCL equations (obtained in Step 2) to write a set of node equations and solve it for the node voltages:

$$\left. \begin{aligned} G_1 v_1 + G_{12}(v_1 - v_2) &= I_{s1} \\ G_{12}(v_2 - v_1) + G_2 v_2 &= -I_{s2} \end{aligned} \right\}; \quad \mathbf{Y} \mathbf{v} = \mathbf{I}_s; \quad \mathbf{v} = \mathbf{Y}^{-1} \mathbf{I}_s \quad (2.9)$$

where

$$\mathbf{Y} = \begin{bmatrix} G_1 + G_{12} & -G_{12} \\ -G_{12} & G_2 + G_{12} \end{bmatrix} \text{ (node admittance matrix), } \mathbf{v} = \begin{bmatrix} v_1 \\ v_2 \end{bmatrix}, \text{ and } \mathbf{I}_s = \begin{bmatrix} I_{s1} \\ -I_{s2} \end{bmatrix}$$

Note. Just as impedance is the generalized concept of resistance, admittance is the generalized concept of conductance, which is the reciprocal of resistance.

Note. KVL has been implicitly used in expressing the branch voltages in terms of the node voltages in Step 1.

Looking at the coefficient matrix, $\mathbf{Y}$, of the node equation (2.9), which is called the *node admittance matrix*, it can be seen that the diagonal elements, $y_{11} = G_1 + G_{12}$ and $y_{22} = G_2 + G_{12}$, are the sums of conductances of the elements connected to node 1 and node 2, respectively. The off-diagonal elements, $y_{12} = -G_{12}$ and $y_{21} = -G_{12}$, are the negative of the conductance of the element connecting the two nodes 1 and 2. The RHS (right-hand side) vector consists of the values of the current sources entering each node. This observation suggests the following formula by which the node admittance matrix and the RHS vector for the node equations can be composed by inspection of the circuit having no voltage source:

$$\mathbf{Y} = \begin{bmatrix} y_{11} & y_{12} & \cdots & y_{1,N-1} \\ y_{21} & y_{22} & \cdots & y_{2,N-1} \\ \vdots & \vdots & \ddots & \vdots \\ y_{N-1,1} & y_{N-1,2} & \cdots & y_{N-1,N-1} \end{bmatrix} \text{ (node admittance matrix), } \mathbf{v} = \begin{bmatrix} v_1 \\ v_2 \\ \vdots \\ v_{N-1} \end{bmatrix}, \text{ and } \mathbf{I}_s = \begin{bmatrix} I_{s1} \\ I_{s2} \\ \vdots \\ I_{s,N-1} \end{bmatrix} \quad (2.10)$$

where

N = the number of nodes

y_{mm} (self-admittance of node m) = the sum of conductances (admittances) of the elements connected to node m

y_{mn} , $m \neq n$ (mutual admittance between nodes m and n) = the negative of the sum of conductances (admittances) of the elements connecting node m and node n

$I_{s,m}$ = the algebraic sum of currents (entering node m) produced by the current sources connected to node m

A question may arise: ‘How can we set up the node equations for circuits having voltage sources?’ There are two approaches, the supernode method and the source transformation method, which will be explained and tried in the subsequent sections.

2.4.1 Circuits Having No Dependent Sources

2.4.1.1 Node Analysis of Circuits Having No Voltage Sources

The above-mentioned method can be used to analyze circuits that are excited by independent current source(s) only. Let us take a look at the following example.

(Example 2.2) A Circuit Excited by Independent Current Sources Only

For the circuit of Figure 2.7(a), the top three nodes are labeled 1, 2, and 3 and the bottom node 0 as the reference node. Then the formula (2.10) is applied to write a set of node equations as

$$\begin{bmatrix} y_{11} & y_{12} & y_{13} \\ y_{21} & y_{22} & y_{23} \\ y_{31} & y_{32} & y_{33} \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 1+2 & -2 & -1 \\ -2 & 2+3 & -3 \\ -1 & -3 & 4+1+3 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 3-1-6 \\ 1+2-4 \\ 6+4+7 \end{bmatrix};$$

$$\begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 3 & -2 & -1 \\ -2 & 5 & -3 \\ -1 & -3 & 8 \end{bmatrix}^{-1} \begin{bmatrix} -4 \\ -1 \\ 17 \end{bmatrix} = \begin{bmatrix} 1 \\ 2 \\ 3 \end{bmatrix}$$

Figure 2.7(b) shows the solution for the node voltages and the resulting branch currents.

Note. See Appendix B for more details about matrix inversion.

2.4.1.2 Node Analysis of Circuits Having Voltage Sources

It is not possible to write a KCL equation for a node connected directly to a voltage source(s), because the current through a voltage source cannot be expressed in terms of node voltages. There are two ways of handling circuits having a voltage source(s), the supernode method and the source transformation method. The *supernode method* regards the group of nonreference nodes connected only via the voltage source(s) as a supernode and applies KCL individually to the nonreference nodes as well as the supernode(s). One thing to note is that if a voltage source is directly connected to the reference node, the node voltage at the other end node of the voltage source is already known so that KCL does not have to be applied to that node, to say nothing of the reference node. Consequently, the number of node

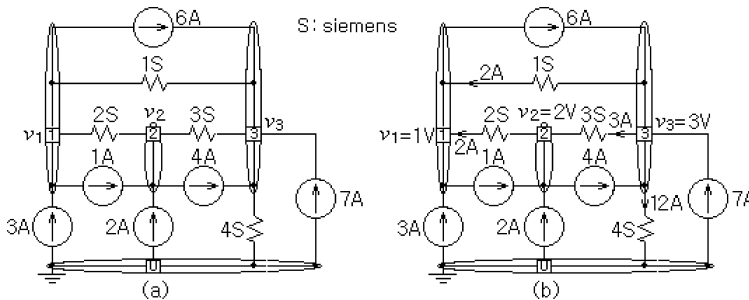


Figure 2.7 Circuit with independent current sources and its solution for Example 2.2

equations or unknown node voltage variables decreases by the number of voltage sources that are connected to the reference node. The *source transformation method* associates every voltage source with an element in series, transforms it into the equivalent current source by using the technique introduced in Section 1.5.2 as needed, and applies the formula (2.10) to set up the node equation(s). Another thing to note is that if a portion of the circuit involving the interested voltages/currents has been modified via source transformation, it should be restored back to the original connection after solving the node equation(s).

It is desired to have not only the capability of applying the supernode method and the source transformation method but also an eye for judging which one takes less trouble. Note that the supernode method requires individual application of KCL to each (nonreference) node, while it does not need a source transformation(s).

(Example 2.3) A Circuit Having Independent Voltage Sources with Series Elements

Find the current, i_{R_3} , through the 3S resistor in the circuit of Figure 2.8(a) by using the node analysis. Since each of the voltage sources is connected to the reference node at one end, KCL does not have to be applied to the other end nodes 1 and 3 of the voltage sources. Thus KCL is applied to node 2 only, yielding the node equation, which is solved as

$$i_{R_1} + i_{R_2} + i_{R_3} = 4; \quad 1(v_2 - v_1) + 2(v_2 - 0) + 3(v_2 - v_3) = 4 \xrightarrow{v_1=5, v_3=-1} 6v_2 = 6; \quad v_2 = 1 \text{ V} \quad (\text{E2.3.1})$$

Technically, it is possible to proceed hypothetically to write the node equations in matrix–vector form as if there were no voltage source:

$$\begin{bmatrix} \times & \times & \times \\ -1 & 1+2+3 & -3 \\ \times & \times & \times \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} \times \\ 4 \\ \times \end{bmatrix} \xrightarrow{v_1=5, v_3=-1} \begin{bmatrix} \times & \times & \times \\ -1 & 6 & -3 \\ \times & \times & \times \end{bmatrix} \begin{bmatrix} 5 \\ v_2 \\ -1 \end{bmatrix} = \begin{bmatrix} \times \\ 4 \\ \times \end{bmatrix} \quad (\text{E2.3.2})$$

This naturally yields the same result:

$$-5 + 6v_2 + 3 = 4; \quad 6v_2 = 6; \quad v_2 = 1 \text{ V} \quad (\text{E2.3.3})$$

Now, the source transformation method will be tried. Since both of the two voltage sources have series resistors, they can easily be transformed into the equivalent current sources as depicted in Figure 2.8(b). Since the resulting circuit has two nodes, we apply KCL only to one node, i.e. the top node 2 or just use the formula (2.10), to write the node equation and solve it to get the same result as

$$i_{R_1} + i_{R_2} + i_{R_3} = 5 + 4 - 3; \quad 1v_2 + 2v_2 + 3v_2 = 6; \quad 6v_2 = 6; \quad v_2 = 1 \text{ V} \quad (\text{E2.3.4})$$

Whichever method is used, we find the current, i_{R_3} , through the 3S resistor in the circuit of Figure 2.8(a), based on the node voltages:

$$i_{R_3} = 3(v_2 - v_3) = 3[1 - (-1)] = 6 \text{ A} \quad (\text{E2.3.5})$$

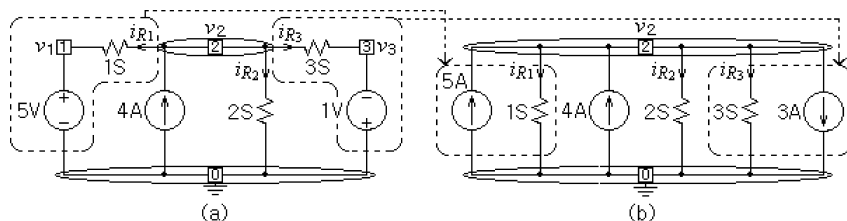


Figure 2.8 Circuit with current and voltage sources for Example 2.3

(Question) If you find the current, i_{R_3} , through the 3S resistor in the circuit of Figure 2.8(b), do you get the same result? If not, find out what is wrong with it.

(Example 2.4) A Circuit Having an Independent Voltage Source with No Series Element

Find the voltage, v_{R_3} , across the 3S resistor in the circuit of Figure 2.9(a) by using the node analysis. Since the circuit has a voltage source between two nonreference nodes 1 and 2, the supernode method and the source transformation method can be tried.

(a) Supernode Method

Referring to Figure 2.9(a), we regard the group of two nodes 1 and 2 connected via the 1 V voltage source as a supernode, say 12, and apply KCL to supernode 12 and node 3 to write the node equations as

$$\begin{aligned} G_1 v_1 + G_2(v_1 - v_3) + G_4 v_2 &= v_1 + 2(v_1 - v_3) + 4 v_2 = -14 \\ G_2(v_3 - v_1) + G_3 v_3 &= 2(v_3 - v_1) + 3 v_3 = 14 \end{aligned} \quad (\text{E2.4.1})$$

Since there are only two equations with three unknowns, one more equation is needed, which will be provided by the matchmaker, the 1V voltage source, between the two nodes 1 and 2:

$$v_1 - v_2 = -1; \quad v_2 = v_1 + 1 \quad (\text{E2.4.2})$$

We substitute this equation into Equation (E2.4.1) and solve it as

$$\begin{bmatrix} 7 & -2 \\ -2 & 5 \end{bmatrix} \begin{bmatrix} v_1 \\ v_3 \end{bmatrix} = \begin{bmatrix} -18 \\ 14 \end{bmatrix}; \quad \begin{bmatrix} v_1 \\ v_3 \end{bmatrix} = \begin{bmatrix} 7 & -2 \\ -2 & 5 \end{bmatrix}^{-1} \begin{bmatrix} -18 \\ 14 \end{bmatrix} = \frac{1}{7 \times 5 - (-2)(-2)} \begin{bmatrix} 5 & 2 \\ 2 & 7 \end{bmatrix} \begin{bmatrix} -18 \\ 14 \end{bmatrix} = \begin{bmatrix} -2 \\ 2 \end{bmatrix} \quad (\text{E2.4.3})$$

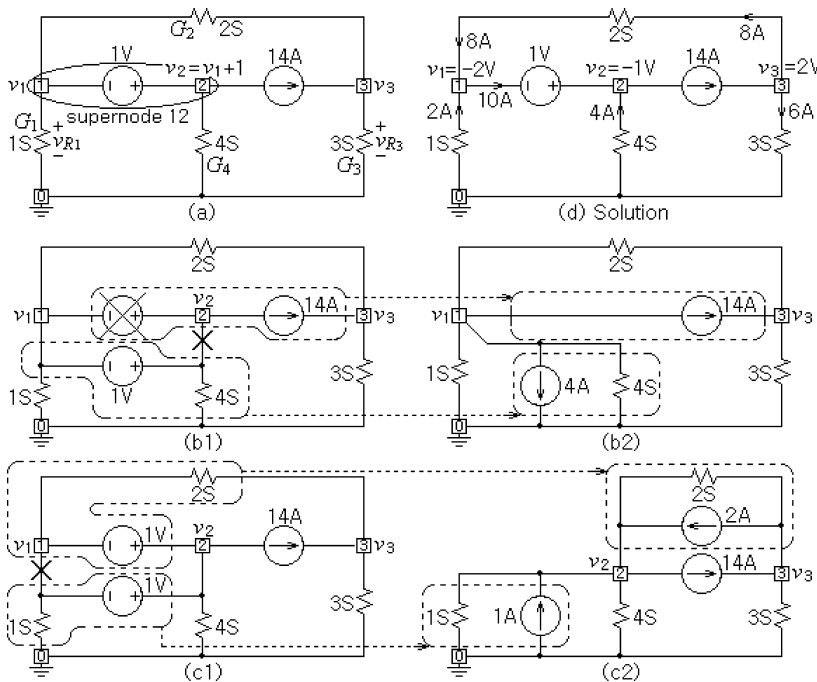


Figure 2.9 Circuit and its equivalents for Example 2.4

Figure 2.9(d) shows the voltage and current solutions as a whole, where $v_{R_3} = v_3 = 2\text{ V}$.

(b) Source Transformation Method

To transform the 1V voltage source into the equivalent current source, it should first be connected with an element in series. For this purpose, it is duplicated to make two copies in parallel, as shown in Figures 2.9(b1) or (c1). Then we associate each of them with the 14 A current source and the 4S resistor, respectively, to transform the latter into a $1\text{ V} \times 4\text{ S} = 4\text{ A}$ current source in parallel with the 4 S resistor as depicted in Figure 2.9(b2), while the former voltage source connected in series with the 14 A current source has disappeared having lost its role (see Section 1.5.1.5). Then we use the formula (2.10) to set up the node equation for the equivalent circuit in Figure 2.9(b2) and solve it to get the same result as Equation (E2.4.3):

$$\begin{bmatrix} 1+2+4 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} v_1 \\ v_3 \end{bmatrix} = \begin{bmatrix} -18 \\ 14 \end{bmatrix}; \quad \begin{bmatrix} v_1 \\ v_3 \end{bmatrix} = \begin{bmatrix} -2 \\ 2 \end{bmatrix} \quad (\text{E2.4.4})$$

Alternatively, we can associate each of the duplicated 1V voltage sources with the 1S resistor and the 2S resistor, respectively, to transform the latter into a $1\text{ V} \times 2\text{ S} = 2\text{ A}$ current source in parallel with the 2S resistor and the former into a $1\text{ V} \times 1\text{ S} = 1\text{ A}$ current source in parallel with the 1S resistor, as depicted in Figure 2.9(c2). Then we use the formula (2.10) to set up the node equation for the equivalent circuit in Figure 2.9(c2) and solve it as

$$\begin{bmatrix} 1+2+4 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} -11 \\ 12 \end{bmatrix}; \quad \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 7 & -2 \\ -2 & 5 \end{bmatrix}^{-1} \begin{bmatrix} -11 \\ 12 \end{bmatrix} = \frac{1}{31} \begin{bmatrix} 5 & 2 \\ 2 & 7 \end{bmatrix} \begin{bmatrix} -11 \\ 12 \end{bmatrix} = \begin{bmatrix} -1 \\ 2 \end{bmatrix} \quad (\text{E2.4.5})$$

2.4.2 Circuits Having Dependent Sources

The existence of dependent sources in a circuit makes no difference in setting up the node equation(s). It only adds another computational load in solving the node equation, which is to express the controlling variable(s) of the dependent source(s) in terms of the node voltages and to move the terms involving the node voltages into the LHS (left-hand side) of the node equations, which is supposed to contain all the terms involving the node voltages. Let us take a look at the following examples.

(Example 2.5) A Circuit Having No Voltage Source

Apply the node analysis to solve the circuit of Figure 2.10 for the node voltages v_1 and v_2 . Since the circuit contains a dependent source, the controlling variable i_{R_3} should be expressed in terms of the node voltages as

$$i_{R_3} = G_3 v_{R_3} = 2(v_1 - v_2) \quad (\text{E2.5.1})$$

Then the formula (2.10) is used to set up the node equation as

$$\begin{bmatrix} 1+2 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} 1 \\ -4i_{R_3} \end{bmatrix} \quad (\text{E2.5.2})$$

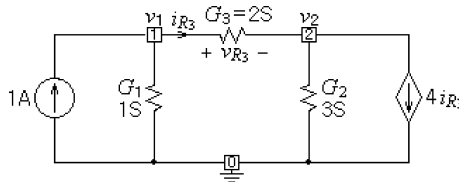


Figure 2.10 Circuit for Example 2.5

To make this equation solvable, the expression (E2.5.1) should be substituted and the node voltage terms moved into the LHS as

$$\begin{bmatrix} 1+2 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} \stackrel{(E2.5.2)}{=} \begin{bmatrix} 1 \\ -8(v_1 - v_2) \end{bmatrix} \stackrel{(E2.5.1)}{=} \begin{bmatrix} 1 \\ -8(v_1 - v_2) \end{bmatrix}; \quad \begin{bmatrix} 3 & -2 \\ 6 & -3 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} 1 \\ 0 \end{bmatrix} \quad (E2.5.3)$$

This equation can now be solved for the node voltages v_1 and v_2 as

$$\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \frac{1}{3} \begin{bmatrix} -3 & 2 \\ -6 & 3 \end{bmatrix} \begin{bmatrix} 1 \\ 0 \end{bmatrix} = \begin{bmatrix} -1 \\ -2 \end{bmatrix} \quad (E2.5.4)$$

(Example 2.6) A Circuit Having a Dependent Voltage Source with a Series Element

Apply the node analysis to solve the circuit of Figure 2.11(a) for the voltage, v_{R_1} , across the $1S$ resistor. Since the circuit contains two dependent sources, their controlling variables v_{R_1} and i_2 should be expressed in terms of the node voltages as

$$v_{R_1} = v_1 - 0 = v_1 \quad \text{and} \quad i_2 = 1 - i_1 = 1 - 1(v_1 - 0) = 1 - v_1 \quad (E2.6.1)$$

where the latter relation is obtained by applying KCL to node 1.

(a) Supernode Method

As depicted in Figure 2.11(a), the group of two nonreference nodes 2 and 3 connected via the $4v_{R_1}$ voltage source can be regarded as a supernode, say 23, and KCL applied to node 1 and the supernode 23 to write the node equations as

$$\begin{aligned} G_1 v_1 + G_2(v_1 - v_2) &= v_1 + 2(v_1 - v_2) = 1 \\ G_2(v_2 - v_1) + G_3 v_3 &= 2(v_2 - v_1) + 3 v_3 = 2i_2 \end{aligned} \quad (E2.6.2)$$

Since there are only two equations with three unknowns, one more equation is needed, which will be provided by the $4v_{R_1}$ voltage source between the two nodes 2 and 3:

$$v_2 - v_3 = 4v_{R_1} \stackrel{(E2.6.1)}{=} 4v_1, \quad v_2 = 4v_1 + v_3 \quad (E2.6.3)$$

This equation and Equation (E2.6.1) can be substituted into Equation (E2.6.2), which is solved as

$$\left. \begin{aligned} 3v_1 - 2(4v_1 + v_3) &= 1 \\ 2(4v_1 + v_3) - 2v_1 + 3v_3 &= 2(1 - v_1) \end{aligned} \right\}; \quad \begin{bmatrix} -5 & -2 \\ 8 & 5 \end{bmatrix} \begin{bmatrix} v_1 \\ v_3 \end{bmatrix} = \begin{bmatrix} 1 \\ 2 \end{bmatrix} \quad (E2.6.4)$$

$$\begin{bmatrix} v_1 \\ v_3 \end{bmatrix} = \frac{1}{-25 - (-16)} \begin{bmatrix} 5 & 2 \\ -8 & -5 \end{bmatrix} \begin{bmatrix} 1 \\ 2 \end{bmatrix} = \begin{bmatrix} -1 \\ 2 \end{bmatrix} \quad (E2.6.5)$$

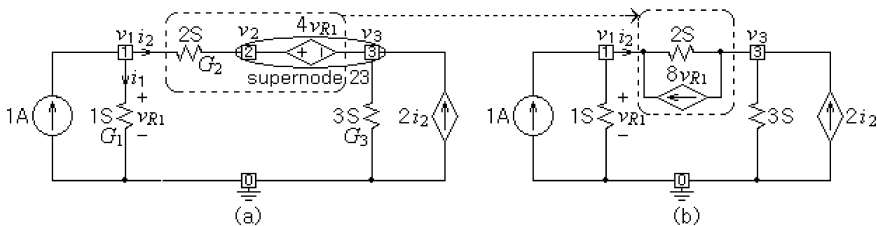


Figure 2.11 Circuit and its equivalent for Example 2.6

(b) Source Transformation Method

Since the $4v_{R_1}$ voltage source has a series resistor of conductance $2S$, it can easily be transformed into the equivalent current source of $4v_{R_1} [V] \times 2S = 8v_{R_1} [A]$ in parallel with the $2S$ resistor, as depicted in Figure 2.11(b). Then the formula (2.10) is used to set up the node equation for the equivalent circuit in Figure 2.11(b) as

$$\begin{bmatrix} 1+2 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} v_1 \\ v_3 \end{bmatrix} = \begin{bmatrix} 1+8v_{R_1} \\ -8v_1+2i_2 \end{bmatrix} \stackrel{(E2.6.1)}{=} \begin{bmatrix} 1+8v_1 \\ 2-10v_1 \end{bmatrix}; \quad \begin{bmatrix} -5 & -2 \\ 8 & 5 \end{bmatrix} \begin{bmatrix} v_1 \\ v_3 \end{bmatrix} = \begin{bmatrix} 1 \\ 2 \end{bmatrix} \quad (E2.6.6)$$

which is identical to Equation (E2.6.4). Note that the target voltage is found to be $v_{R_1} = v_1 = -1[V]$.

(Example 2.7) A Circuit Having a Dependent Voltage Source with No Series Element

Apply the node analysis to solve the circuit of Figure 2.12(a) for the voltage, v_{R_4} , across the $4S$ resistor. Since the circuit contains two dependent sources, the controlling variable i_0 should be expressed in terms of the node voltages as

$$i_0 \stackrel{\text{KCL to node 1}}{=} i_1 + i_2 = 1(v_1 - v_3) + 2(v_1 - v_2) \stackrel{v_1=2v_3}{\underset{\text{nodes 1} \sim 0}{=}} 1(2v_3 - v_3) + 2(2v_3 - v_2) = -2v_2 + 5v_3 \quad (E2.7.1)$$

which is obtained by applying KCL to node 1.

First, noting that the $2v_3$ voltage source is directly connected to the reference node, an attempt is made to set up the node equation without having to think in terms of ‘supernode’ as if there were no voltage source:

$$\begin{bmatrix} \times & \times & \times \\ -2 & 2+4 & -4 \\ -1 & -4 & 3+1+4 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} \times \\ 6-2i_0 \\ 14-6 \end{bmatrix} \stackrel{v_1=2v_3}{\underset{(E2.7.1)}{=}} \begin{bmatrix} \times & \times & \times \\ -2 & 6 & -4 \\ -1 & -4 & 8 \end{bmatrix} \begin{bmatrix} 2v_3 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} \times \\ 6-2(5v_3-2v_2) \\ 8 \end{bmatrix}$$

$$\begin{bmatrix} 2 & 2 \\ -4 & 6 \end{bmatrix} \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 6 \\ 8 \end{bmatrix}; \quad \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \frac{1}{20} \begin{bmatrix} 6 & -2 \\ 4 & 2 \end{bmatrix} \begin{bmatrix} 6 \\ 8 \end{bmatrix} = \begin{bmatrix} 1 \\ 2 \end{bmatrix} \quad (E2.7.2)$$

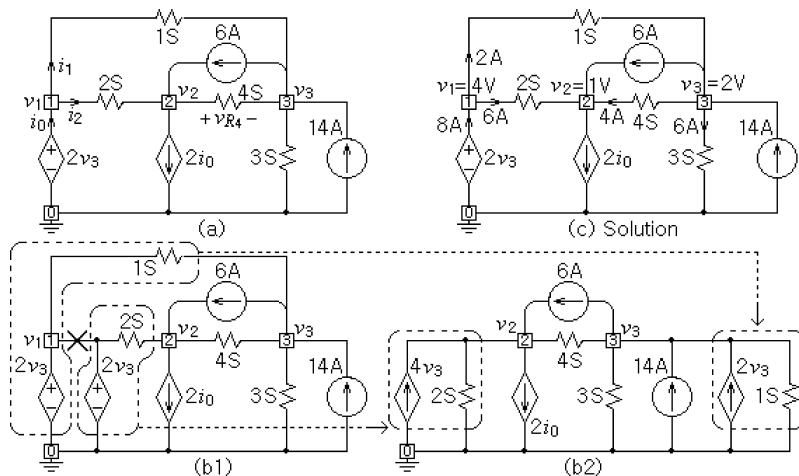


Figure 2.12 Circuit and its equivalents for Example 2.7

Now the source transformation method is tried. Noting that the $2v_3$ voltage source has no series element, it is duplicated to make two copies in parallel and each of them is associated with the $1S$ resistor and the $2S$ resistor, respectively, to transform the latter into a $2S \times 2v_3 [V] = 4v_3 [A]$ current source in parallel with the $2S$ resistor and the former into a $1S \times 2v_3 [V] = 2v_3 [A]$ current source in parallel with the $1S$ resistor, as depicted in Figures 2.12(b1) and (b2). Then the formula (2.10) can be applied to set up the node equation for the equivalent circuit in Figure 2.12(b2) as

$$\begin{bmatrix} 2+4 & -4 \\ -4 & 3+1+4 \end{bmatrix} \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 4v_3 + 6 - 2i_0 \\ 2v_3 + 14 - 6 \end{bmatrix} \stackrel{(E2.7.1)}{=} \begin{bmatrix} 6 + 4v_2 - 6v_3 \\ 8 + 2v_3 \end{bmatrix} \rightarrow \begin{bmatrix} 2 & 2 \\ -4 & 6 \end{bmatrix} \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 6 \\ 8 \end{bmatrix} \quad (E2.7.3)$$

This node equation is identical to the above one.

Figure 2.12(c) shows the voltage and current solutions as a whole, where the target voltage is found to be $v_{R_4} = v_2 - v_3 = 1 - 2 = -1 [V]$.

2.5 Mesh (Loop) Analysis

In the mesh (loop) analysis, we apply KVL to the meshes to write a set of mesh equations and solve it for the unknown mesh currents, where a mesh current is supposed to be circulating around each mesh. Mesh analysis is applicable to ‘planar circuits’, which can be drawn on a plane with no crossing branches. The mesh analysis procedure of setting up the mesh equations for the circuit in Figure 2.13 consists of the following steps:

Step 0. Label all the meshes, say the numbers $1, 2, \dots$, that correspond to assigning the mesh current variable $i_1, i_2, \dots$ commonly with the clockwise reference direction to each mesh, respectively.

Step 1. Based on the VCR (voltage–current relationship), express the branch voltage of every element except sources in terms of the mesh currents (complying with the passive sign convention):

$$v_1 = R_1 i_{R_1} = R_1 i_1, \quad v_2 = R_2 i_{R_2} = R_2 i_2, \quad v_{12} = R_{12} i_{R_{12}} = R_{12} (i_1 - i_2)$$

Step 2. Apply KVL around the meshes to write the mesh equations with the branch voltage terms on the left-hand side and the source terms on the right-hand side:

$$\begin{aligned} v_{R_1} + v_{R_{12}} &= V_{s1} \\ -v_{R_{12}} + v_{R_2} &= -V_{s2} \end{aligned}$$

Step 3. Substitute the VCR equations (obtained in Step 1) into the KVL equations (obtained in Step 2) to write a set of mesh equations and solve it for the mesh currents:

$$\left. \begin{aligned} R_1 i_1 + R_{12} (i_1 - i_2) &= V_{s1} \\ R_{12} (i_2 - i_1) + R_2 i_2 &= -V_{s2} \end{aligned} \right\}; \quad \mathbf{Z} \mathbf{i} = \mathbf{V}_s; \quad \mathbf{i} = \mathbf{Z}^{-1} \mathbf{V}_s \quad (2.11)$$

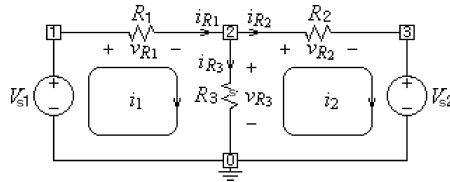


Figure 2.13 A two-mesh circuit with independent voltage sources

where

$$\mathbf{Z} = \begin{bmatrix} R_1 + R_{12} & -R_{12} \\ -R_{12} & R_2 + R_{12} \end{bmatrix} \text{ (mesh impedance matrix),} \quad \mathbf{i} = \begin{bmatrix} i_1 \\ i_2 \end{bmatrix}, \quad \text{and} \quad \mathbf{V}_s = \begin{bmatrix} V_{s1} \\ -V_{s2} \end{bmatrix}$$

Note. KCL has been implicitly used in expressing the branch currents in terms of the mesh currents in Step 1.

Note. The reference directions of all mesh currents can be chosen to be counterclockwise in the whole.

Let us take a look at the coefficient matrix, $\mathbf{Z}$, of the mesh equations (2.11), which is called the *mesh impedance matrix*. The diagonal elements, $z_{11} = R_1 + R_{12}$ and $z_{22} = R_2 + R_{12}$, are the sums of resistances of the elements contained in mesh 1 and mesh 2, respectively. The off-diagonal elements, $z_{12} = -R_{12}$ and $z_{21} = -R_{12}$, are in general the negative of the resistance of the element shared by the two meshes 1 and 2. The RHS (right-hand side) vector consists of the values of the voltage sources, giving rise to the mesh current in the reference direction. This observation suggests the following formula by which the mesh impedance matrix and the RHS vector for the mesh equations can be composed by inspection of the circuit diagram having no current source:

$$\mathbf{Z} = \begin{bmatrix} z_{11} & z_{12} & \cdots & z_{1,N-1} \\ z_{21} & z_{22} & \cdots & z_{2,N-1} \\ \vdots & \vdots & \ddots & \vdots \\ \vdots & \vdots & \ddots & \vdots \\ z_{N-1,1} & z_{N-1,2} & \cdots & z_{N-1,N-1} \end{bmatrix} \text{ (mesh impedance matrix),} \quad \mathbf{i} = \begin{bmatrix} i_1 \\ i_2 \\ \vdots \\ i_{N-1} \end{bmatrix}, \quad \text{and} \quad \mathbf{V}_s = \begin{bmatrix} V_{s1} \\ V_{s2} \\ \vdots \\ V_{s,N-1} \end{bmatrix} \quad (2.12)$$

where

N = the number of meshes

z_{mm} (self-impedance of mesh m) = the sum of resistances (impedances) of the elements contained in mesh m

$z_{mn}, m \neq n$ (mutual impedance between meshes m and n) = the negative of the sum of resistances (impedances) of the elements shared by mesh m and mesh n

$V_{s,m}$ = the algebraic sum of voltage rises (giving rise to mesh current i_m in the reference direction) produced by the voltage sources contained in mesh m

A question may arise: ‘How can we set up the mesh equations for circuits having current sources?’ There are two approaches, the supermesh method and the source transformation method, which will be explained and tried in the subsequent section.

2.5.1 Circuits Having No Dependent Sources

2.5.1.1 Mesh Analysis of Circuits Having No Current Sources

The above-mentioned method can be used to analyze circuits that are excited by independent voltage source(s) only. Let us take a look at the following example.

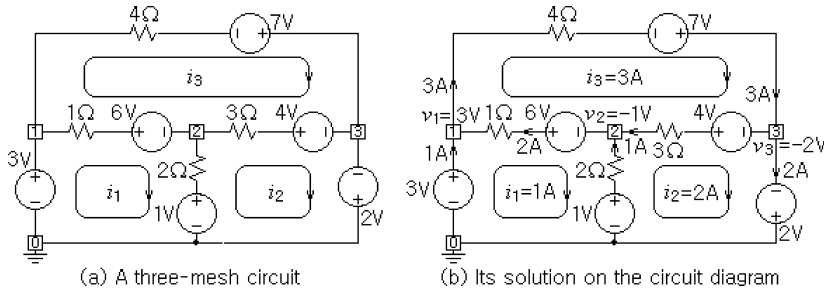


Figure 2.14 Circuit and its solution for Example 2.8

(Example 2.8) A Circuit Excited by Independent Voltage Sources Only

For the circuit of Figure 2.14(a), the three meshes are labeled 1, 2, and 3. Then the formula (2.12) is applied to write the set of node equations as

$$\begin{bmatrix} z_{11} & z_{12} & z_{13} \\ z_{21} & z_{22} & z_{23} \\ z_{31} & z_{32} & z_{33} \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} 1+2 & -2 & -1 \\ -2 & 2+3 & -3 \\ -1 & -3 & 4+1+3 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} 3-6-1 \\ 1-4+2 \\ 7+4+6 \end{bmatrix};$$

$$\begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} 3 & -2 & -1 \\ -2 & 5 & -3 \\ -1 & -3 & 8 \end{bmatrix}^{-1} \begin{bmatrix} -4 \\ -1 \\ 17 \end{bmatrix} = \begin{bmatrix} 1 \\ 2 \\ 3 \end{bmatrix}$$

Figure 2.14(b) shows the solution for the mesh currents and the resulting branch currents.

2.5.1.2 Mesh Analysis of Circuits Having Current Sources

It is not possible to write a KVL equation for a mesh containing a current source(s), because the voltage across a current source cannot be expressed in terms of mesh currents. There are two ways of handling circuits having a current source(s), the supermesh method and the source transformation method. The *supermesh method* regards the neighboring meshes sharing a current source(s) (pairwise) as a supermesh and applies KVL individually to the meshes as well as the supermesh(es). One thing to note is that if a current source is contained exclusively in a single mesh hanging on its outer branch, the mesh current is already determined as the value of the current source; therefore KVL does not have to be applied to that mesh. Consequently, the number of mesh equations and unknown mesh current variables decreases by the number of such current sources that are contained exclusively in one mesh. On the other hand, the *source transformation method* associates every current source with an element in parallel, transforms it into the equivalent voltage source by using the technique introduced in Section 1.5.2 as needed, and applies the formula (2.12) to set up the mesh equations. Another thing to note is that if a portion of the circuit involving the interested voltages/currents has been modified via the source transformation, it should be restored back to the original connection after the solution is obtained.

It is desired to have not only the capability of applying the supermesh method and the source transformation method but also an eye for judging which one takes less effort. Note that the supermesh method requires an individual application of KVL to each mesh (or loop), while it does not need source transformation(s).

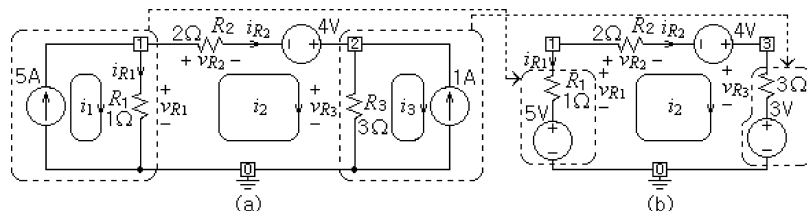


Figure 2.15 Circuit with current sources and its equivalent for Example 2.9

(Example 2.9) A Circuit Having Independent Current Sources with Parallel Elements

Find the current, i_{R_1} , through the $1\ \Omega$ resistor in the circuit of Figure 2.15(a) by using the mesh analysis. Since the two current sources are contained exclusively in mesh 1 and mesh 3 so that they determine the mesh currents i_1 and i_3 as 5 A and -1 A, respectively, KVL needs to be applied only to mesh 2 to write the mesh equation

$$\begin{aligned} -v_{R_1} + v_{R_2} + v_{R_3} &= -R_1 i_{R_1} + R_2 i_{R_2} + R_3 i_{R_3} = 4 \\ -1(i_1 - i_2) + 2i_2 + 3(i_2 - i_3) &= 4 \xrightarrow{i_1=5, i_3=-1} 6i_2 = 6; \quad i_2 = 1\text{ A} \end{aligned} \quad (\text{E2.9.1})$$

Technically, this can be done by proceeding hypothetically to write the mesh equations in matrix-vector form as if there were no voltage sources:

$$\begin{bmatrix} \times & \times & \times \\ -1 & 1+2+3 & -3 \\ \times & \times & \times \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} \times \\ 4 \\ \times \end{bmatrix} \xrightarrow{i_1=5, i_3=-1} \begin{bmatrix} \times & \times & \times \\ -1 & 6 & -3 \\ \times & \times & \times \end{bmatrix} \begin{bmatrix} 5 \\ i_2 \\ -1 \end{bmatrix} = \begin{bmatrix} \times \\ 4 \\ \times \end{bmatrix}; \quad 6i_2 = 6; \quad i_2 = 1 \quad (\text{E2.9.2})$$

Now let us try the source transformation method. Since both of the two current sources have parallel resistors, they can easily be transformed into the equivalent voltage sources, as depicted in Figure 2.15(b). Then we apply KVL to the (only one) mesh of the circuit or simply use the formula (2.12) to write the mesh equation and solve it to get the same result as

$$\begin{aligned} -v_{R_1} + v_{R_2} + v_{R_3} &= -R_1 i_{R_1} + R_2 i_{R_2} + R_3 i_{R_3} = -R_1(-i_2) + R_2 i_2 + R_3 i_2 = 5 + 4 - 3 \\ -1(-i_2) + 2i_2 + 3i_2 &= 6; \quad 6i_2 = 6; \quad i_2 = 1\text{ A} \end{aligned} \quad (\text{E2.9.3})$$

Whichever method is used, we find the current, i_{R_1} , through the $1\ \Omega$ resistor in the circuit of Figure 2.15(a), based on the mesh currents:

$$i_{R_1} = i_1 - i_2 = 5 - 1 = 4\text{ A} \quad (\text{E2.9.4})$$

(Question) If you find the current, i_{R_1} , through the $1\ \Omega$ resistor in the circuit of Figure 2.15(b), do you get the same result? If not, find out what is wrong with it.

(Example 2.10) A Circuit Having an Independent Current Source with No Parallel Element

Find the branch current i_{R_1} through the $1\ \Omega$ resistor in the circuit of Figure 2.16(a) by using the mesh analysis. Since the circuit has a current source between two meshes 1 and 2, both the supermesh method and the source transformation method can be tried.

(a) Supermesh Method

Referring to Figure 2.16(a), the group of two meshes sharing the 1 A current source can be regarded as a supermesh, say 12, and KVL can be applied to the supermesh 12 and mesh 3 to write the mesh

equations as

$$\begin{aligned} R_1 i_1 + R_2(i_1 - i_3) + R_4 i_2 &= i_1 + 2(i_1 - i_3) + 4 i_2 = -14 \\ R_2(i_3 - i_1) + R_3 i_3 &= 2(i_3 - i_1) + 3 i_3 = 14 \end{aligned} \quad (\text{E2.10.1})$$

Since there are only two equations with three unknowns, one more equation is needed, which will be provided by the matchmaker, the 1 A current source, between the two meshes 1 and 2:

$$i_1 - i_2 = -1; \quad i_2 = i_1 + 1 \quad (\text{E2.10.2})$$

We substitute this equation into Equation (E2.10.1) and solve it as

$$\begin{bmatrix} 7 & -2 \\ -2 & 5 \end{bmatrix} \begin{bmatrix} i_1 \\ i_3 \end{bmatrix} = \begin{bmatrix} -18 \\ 14 \end{bmatrix}; \quad \begin{bmatrix} i_1 \\ i_3 \end{bmatrix} = \begin{bmatrix} 7 & -2 \\ -2 & 5 \end{bmatrix}^{-1} \begin{bmatrix} -18 \\ 14 \end{bmatrix} = \frac{1}{7 \times 5 - (-2)(-2)} \begin{bmatrix} 5 & 2 \\ 2 & 7 \end{bmatrix} \begin{bmatrix} -18 \\ 14 \end{bmatrix} = \begin{bmatrix} -2 \\ 2 \end{bmatrix} \quad (\text{E2.10.3})$$

Figure 2.16(d) shows the voltage and current solutions as a whole. The target current i_{R_1} is obtained as

$$i_{R_1} = -i_1 = -(-2) = 2 \text{ A} \quad (\text{E2.10.4})$$

(b) Source Transformation Method

To transform the 1 A current source into the equivalent voltage source, it should first be made to have a parallel element. For this purpose, it is duplicated to make two copies in series, as shown in Figures 2.16(b1) or (c1). Then each of them is associated with the 14 V voltage source and the 4 Ω resistor, respectively, to transform the latter into a $1 \text{ A} \times 4 \Omega = 4 \text{ V}$ voltage source in series with the 4 Ω

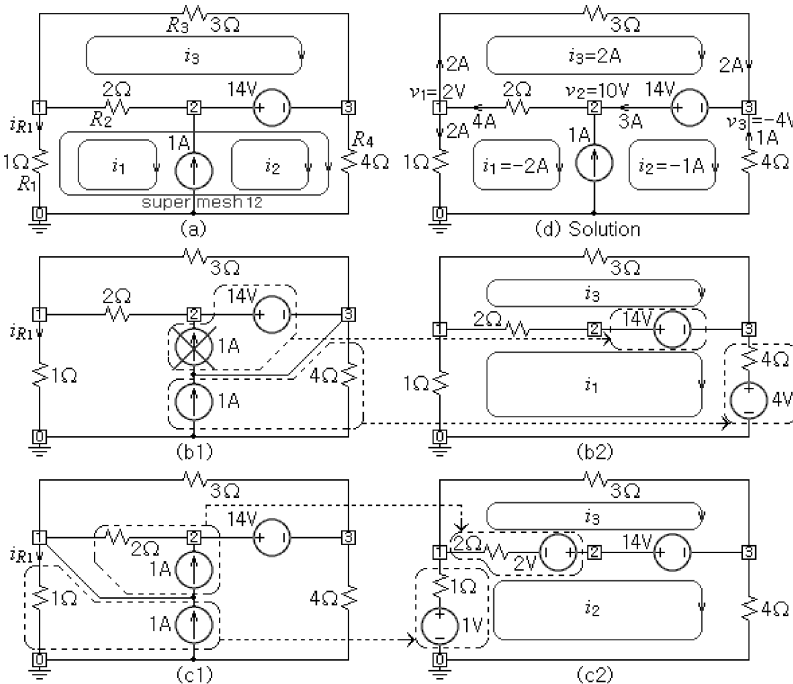


Figure 2.16 Circuit and its equivalents for Example 2.10

Figure 2.16 Circuit and its equivalents for Example 2.10

resistor, as depicted in Figure 2.16(b2), while the former current source connected in parallel with the 14 V voltage source has disappeared having lost its role (see Section 1.5.1.6). Then we use the formula (2.12) to set up the mesh equation for the equivalent circuit in Figure 2.16(b2) and solve it to get the same result as Equation (E2.10.3):

$$\begin{bmatrix} 1+2+4 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} i_1 \\ i_3 \end{bmatrix} = \begin{bmatrix} -18 \\ 14 \end{bmatrix}; \quad \begin{bmatrix} i_1 \\ i_3 \end{bmatrix} = \begin{bmatrix} -2 \\ 2 \end{bmatrix} \quad (\text{E2.10.5})$$

Alternatively, each of the duplicated 1 A current sources can be associated with the 2Ω resistor and the 1Ω resistor, respectively, to transform the latter into a $1\text{ A} \times 1\Omega = 1\text{ V}$ voltage source in series with the 1Ω resistor and the former into a $1\text{ A} \times 2\Omega = 2\text{ V}$ voltage source in series with the 2Ω resistor, as depicted in Figure 2.16(c2). Then we use the formula (2.12) to set up the mesh equation for the equivalent circuit in Figure 2.9(c2) and solve it as

$$\begin{bmatrix} 1+2+4 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} -11 \\ 12 \end{bmatrix}; \quad \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} 7 & -2 \\ -2 & 5 \end{bmatrix}^{-1} \begin{bmatrix} -11 \\ 12 \end{bmatrix} = \frac{1}{31} \begin{bmatrix} 5 & 2 \\ 2 & 7 \end{bmatrix} \begin{bmatrix} -11 \\ 12 \end{bmatrix} = \begin{bmatrix} -1 \\ 2 \end{bmatrix} \quad (\text{E2.10.6})$$

(Question) If you find the current, i_{R_1} , through the 1Ω resistor in the circuit of Figure 2.16(c2), do you get the same result? If not, find out what is wrong with it.

2.5.2 Circuits Having Dependent Sources

The existence of dependent sources in a circuit makes no difference in setting up the mesh equations. It only adds another computational load in solving the mesh equations, which is to express the controlling variable(s) of the dependent source(s) in terms of the mesh currents and to move the terms involving the mesh currents into the LHS (left-hand side) of the mesh equations, where the LHS is supposed to contain all the terms involving the mesh currents. Let us take a look at the following examples.

(Example 2.11) A Circuit Having No Current Source

Apply the mesh analysis to solve the circuit of Figure 2.17 for the mesh currents i_1 and i_2 . Since the circuit contains a dependent source, the controlling variable v_{R_3} should be expressed in terms of the mesh currents as

$$v_{R_3} = R_3 i_{R_3} = 2(i_1 - i_2) \quad (\text{E2.11.1})$$

Then the formula (2.12) is used to set up the mesh equation as

$$\begin{bmatrix} 1+2 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} 1 \\ -4v_{R_3} \end{bmatrix} \quad (\text{E2.11.2})$$

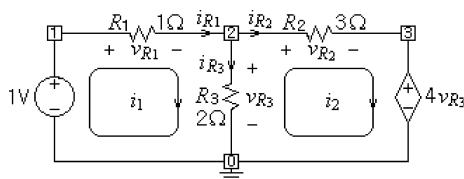


Figure 2.17 Two-mesh circuit with a dependent voltage source for Example 2.11

To make this equation solvable, the expression (E2.11.1) should be substituted and the mesh current terms moved into the LHS as

$$\begin{bmatrix} 1+2 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} \stackrel{(E2.11.2)}{\underset{(E2.11.1)}}{=} \begin{bmatrix} 1 \\ -8(i_1 - i_2) \end{bmatrix}; \quad \begin{bmatrix} 3 & -2 \\ 6 & -3 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} 1 \\ 0 \end{bmatrix} \quad (\text{E2.11.3})$$

Now this equation is solved for the mesh currents i_1 and i_2 as

$$\begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \frac{1}{3} \begin{bmatrix} -3 & 2 \\ -6 & 3 \end{bmatrix} \begin{bmatrix} 1 \\ 0 \end{bmatrix} = \begin{bmatrix} -1 \\ -2 \end{bmatrix} \quad (\text{E2.11.4})$$

(Example 2.12) A Circuit Having a Dependent Current Source with a Parallel Element

Apply the mesh analysis to solve the circuit of Figure 2.18(a) for the current, i_{R_2} , through the 2Ω resistor. Since the circuit contains two dependent sources, their controlling variables i_{R_1} and v_{R_2} should be expressed in terms of the mesh currents as

$$i_{R_1} = i_1 \quad \text{and} \quad v_{R_2} = v_2 = v_1 - R_1 i_{R_1} = 1 - 1i_1 = 1 - i_1 \quad (\text{E2.12.1})$$

where the latter expression is obtained by applying KVL to mesh 1.

(a) Supermesh Method

As depicted in Figure 2.18(a), the group of two meshes 2 and 3 sharing the $4i_{R_1}$ current source can be regarded as a supermesh, say 23, and KCL applied to mesh 1 and supermesh 23 to write the mesh equations as

$$\begin{aligned} R_1 i_1 + R_2(i_1 - i_2) &= i_1 + 2(i_1 - i_2) = 1 \\ R_2(i_2 - i_1) + R_3 i_3 &= 2(i_2 - i_1) + 3 i_3 = 2v_{R_2} \end{aligned} \quad (\text{E2.12.2})$$

Since there are only two equations with three unknowns, one more equation is needed, which will be provided by the $4i_{R_1}$ current source coupling the two meshes 2 and 3:

$$i_2 - i_3 = 4i_{R_1}; \quad i_2 = 4i_{R_1} + i_3 \stackrel{(E2.12.1)}{=} 4i_1 + i_3 \quad (\text{E2.12.3})$$

This equation and Equation (E2.12.1) can be substituted into Eq. (E2.12.2) and solved as

$$\left. \begin{aligned} 3i_1 - 2(4i_1 + i_3) &= 1 \\ 2(4i_1 + i_3) - 2i_1 + 3i_3 &= 2(1 - i_1) \end{aligned} \right\}; \quad \begin{bmatrix} -5 & -2 \\ 8 & 5 \end{bmatrix} \begin{bmatrix} i_1 \\ i_3 \end{bmatrix} = \begin{bmatrix} 1 \\ 2 \end{bmatrix} \quad (\text{E2.12.4})$$

$$\begin{bmatrix} i_1 \\ i_3 \end{bmatrix} = \frac{1}{-25 - (-16)} \begin{bmatrix} 5 & 2 \\ -8 & -5 \end{bmatrix} \begin{bmatrix} 1 \\ 2 \end{bmatrix} = \begin{bmatrix} -1 \\ 2 \end{bmatrix} \quad (\text{E2.12.5})$$

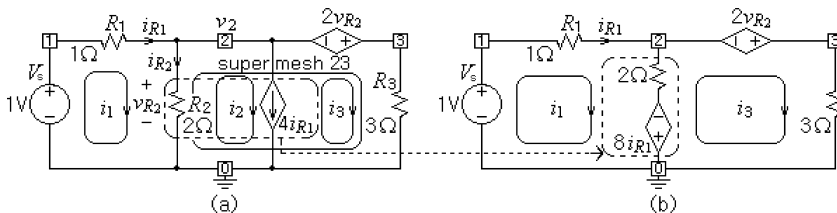


Figure 2.18 Circuit with a dependent current source and its equivalent for Example 2.12

Based on this result, the target current i_{R_2} can be obtained as

$$i_{R_2} = i_1 - i_2 \stackrel{(E2.12.3)}{=} i_1 - (4i_1 + i_3) = -3i_1 - i_3 \stackrel{(E2.12.5)}{=} -3(-1) - 2 = 1 \text{ [A]} \quad (E2.12.6)$$

(b) Source Transformation Method

Since the $4i_{R_1}$ current source has a parallel resistor of resistance 2Ω , it can easily be transformed into the equivalent voltage source of $2\Omega \times 4i_{R_1} \text{ [A]} = 8i_{R_1} \text{ [V]}$ in series with the 2Ω resistor, as depicted in Figure 2.18(b). Then the formula (2.12) is used to set up the mesh equation for the equivalent circuit in Figure 2.18(b) as

$$\begin{bmatrix} 1+2 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} i_1 \\ i_3 \end{bmatrix} = \begin{bmatrix} 1+8i_{R_1} \\ -8i_{R_1}+2v_{R_2} \end{bmatrix} \stackrel{(E2.12.1)}{=} \begin{bmatrix} 1+8i_1 \\ 2-10i_1 \end{bmatrix}; \quad \begin{bmatrix} -5 & -2 \\ 8 & 5 \end{bmatrix} \begin{bmatrix} i_1 \\ i_3 \end{bmatrix} = \begin{bmatrix} 1 \\ 2 \end{bmatrix} \quad (E2.12.7)$$

which is identical to Equation (E2.12.4).

(Question) If you find the current, i_{R_2} , through the 2Ω resistor in the circuit of Figure 2.18(b), do you get the same result? If not, find out what is wrong with it.

(Example 2.13) A Circuit Having a Dependent Current Source with No Parallel Element

Apply the mesh analysis to solve the circuit of Figure 2.19(a) for the current, i_{R_4} , through the 4Ω resistor. Since the circuit contains two dependent sources, the controlling variable v_1 should be expressed in terms of the mesh currents as

$$v_1 = v_3 - 14 + R_3 i_3 = 2v_1 - 14 + 3i_3; \quad v_1 = 14 - 3i_3 \quad (E2.13.1)$$

which is obtained by applying KVL along the path R_3 -14 V(source)- $2v_1$ (source) from the reference node to node 1.

First, noting that the $2i_3$ current source is contained exclusively in the mesh 1 so that it determines that mesh current to be $i_1 = 2i_3$, an attempt is made hypothetically to set up the mesh equations without having to think of ‘supermesh’ as if there were no current source:

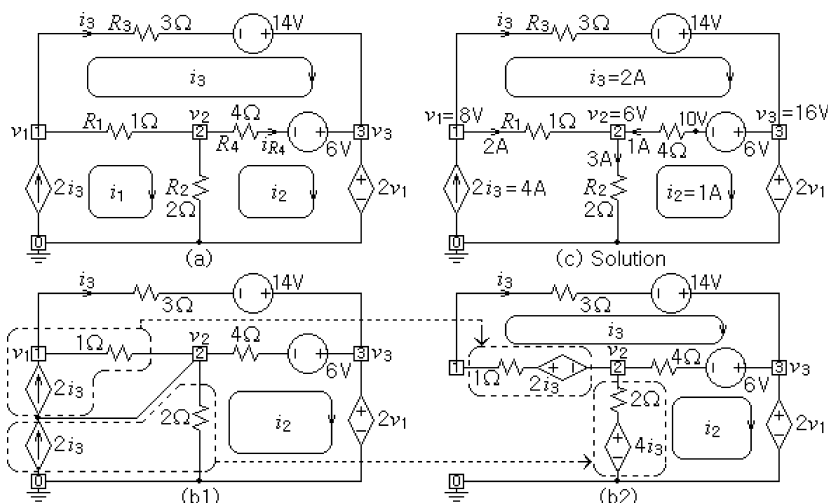


Figure 2.19 Circuit and its equivalents for Example 2.13

$$\begin{bmatrix} \times & \times & \times \\ -2 & 2+4 & -4 \\ -1 & -4 & 3+1+4 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} \times \\ 6-2v_1 \\ 14-6 \end{bmatrix} \xrightarrow[\text{(E2.13.1)}]{v_1=14-3v_3} \begin{bmatrix} \times & \times & \times \\ -2 & 6 & -4 \\ -1 & -4 & 8 \end{bmatrix} \begin{bmatrix} 2i_3 \\ i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} \times \\ 6-2(14-3i_3) \\ 8 \end{bmatrix}$$

$$\begin{bmatrix} 6 & -14 \\ -4 & 6 \end{bmatrix} \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} -22 \\ 8 \end{bmatrix}; \quad \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \frac{1}{6 \times 6 - (-14)(-4)} \begin{bmatrix} 6 & 14 \\ 4 & 6 \end{bmatrix} \begin{bmatrix} -22 \\ 8 \end{bmatrix} = \begin{bmatrix} 1 \\ 2 \end{bmatrix} \quad (\text{E2.13.2})$$

Now the source transformation method is tried. Noting that the $2i_3$ current source has no parallel element, it is duplicated to make two copies in series and each of them is associated with the 1Ω resistor and the 2Ω resistor, respectively, to transform the latter into a $2\Omega \times 2i_3 [\text{A}] = 4i_3 [\text{V}]$ voltage source in series with the 2Ω resistor and the former into a $1\Omega \times 2i_3 [\text{A}] = 2i_3 [\text{V}]$ voltage source in series with the 1Ω resistor, as depicted in Figures 2.19(b1) and (b2). Then the formula (2.12) can be applied to set up the mesh equation for the equivalent circuit in Figure 2.19(b2) as

$$\begin{bmatrix} 2+4 & -4 \\ -4 & 3+1+4 \end{bmatrix} \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} 4i_3+6-2v_1 \\ 2i_3+14-6 \end{bmatrix} \xrightarrow{\text{(E2.13.1)}} \begin{bmatrix} 6 & -14 \\ -4 & 6 \end{bmatrix} \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} -22 \\ 8 \end{bmatrix} \quad (\text{E2.13.3})$$

This mesh equation is identical to the one above.

Figure 2.19(c) shows the voltage and current solutions as a whole, where the target current is found to be $i_{R_4} = i_2 - i_3 = 1 - 2 = -1 [\text{A}]$.

2.6 Comparison of Node Analysis and Mesh Analysis

The previous two sections have discussed two important circuit analysis methods: the node analysis and the mesh (loop) analysis. In this section consideration is given to how to select one that is preferable to the other for a given circuit analysis problem. The two methods should be compared in terms of the following factors:

1. Which method has fewer equations?
2. Which is simpler/easier, the voltage-to-current source transformation method (transforming all the voltage sources into the equivalent current sources), the supernode method (dealing with any group of nodes connected via voltage sources as a supernode), the current-to-voltage source transformation method (transforming all the current sources into the equivalent voltage sources), or the supermesh method (dealing with any group of meshes sharing current sources pairwise as a supermesh)?
3. Which are the target quantities to find, voltage(s) or current(s)?
4. Mesh analysis is applicable only to planar circuits that can be drawn on a plane with no crossing branch.
5. Node analysis is the only choice for the approximate solution of OP Amp circuits.

Among these, the first consideration, (point 1) normally becomes the deciding factor. To estimate the number of node/mesh equations, we first use the series/parallel combinations to simplify the circuit, remove all the sources in the circuit by short-circuiting every voltage source and open-circuiting every current source, and then count the numbers of branches and nodes in the resulting circuit. With b branches and n nodes, the numbers of node equations and mesh equations turn out to be $n-1$ and $b-(n-1)$, respectively (refer to Section 1.4.4). As for the second consideration (point 2), some experience is required, to be accumulated through the examples in the previous sections. Regarding the third factor (point 3), the node/mesh analysis is preferred when voltages/currents are sought. If there is not much to choose between one or the other, the choice is a matter of personal style.

It is hoped that readers of this book become proficient in both analysis methods so that they can use one to solve a given problem and the other to check the result. Let us take a look at the following examples.

(Example 2.14) A Circuit with Four Branches and Three Nodes

Consider the circuit of Figure 2.20(a). With the voltage/current source short-circuited/open-circuited, the numbers of branches and nodes are $b = 4$ and $n = 3$, respectively, which implies that the numbers of node equations and mesh equations are two in common:

$$n - 1 = 3 - 1 = 2 \quad \text{and} \quad b - (n - 1) = 4 - (3 - 1) = 2 \quad (\text{E2.14.1})$$

Let us try both the node analysis and the mesh analysis.

(a) Node Analysis

The controlling variable of the $2i_3$ current source is first expressed in terms of the node voltages as

$$i_3 = \frac{v_1 - v_3}{3} \quad v_3 = -6 \quad \frac{v_1 + 6}{3} \quad (\text{E2.14.2})$$

Noting that the 6 V voltage source is connected to the reference node at one end, the node equation can be set up without thinking of 'supernode' in terms of having no voltage source:

$$\begin{bmatrix} 1/3 + 1 & -1 & -1/3 \\ -1 & 1 + 1/2 + 1/4 & -1/4 \\ \times & \times & \times \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 = -6 \end{bmatrix} = \begin{bmatrix} 2i_3 = 2(v_1 + 6)/3 \\ 0 \\ \times \end{bmatrix}$$

$$\begin{bmatrix} 2 & -3 \\ -4 & 7 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} 12 - 6 = 6 \\ -6 \end{bmatrix}; \quad \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \frac{1}{2} \begin{bmatrix} 7 & 3 \\ 4 & 2 \end{bmatrix} \begin{bmatrix} 6 \\ -6 \end{bmatrix} = \begin{bmatrix} 12 \\ 6 \end{bmatrix} \quad (\text{E2.14.3})$$

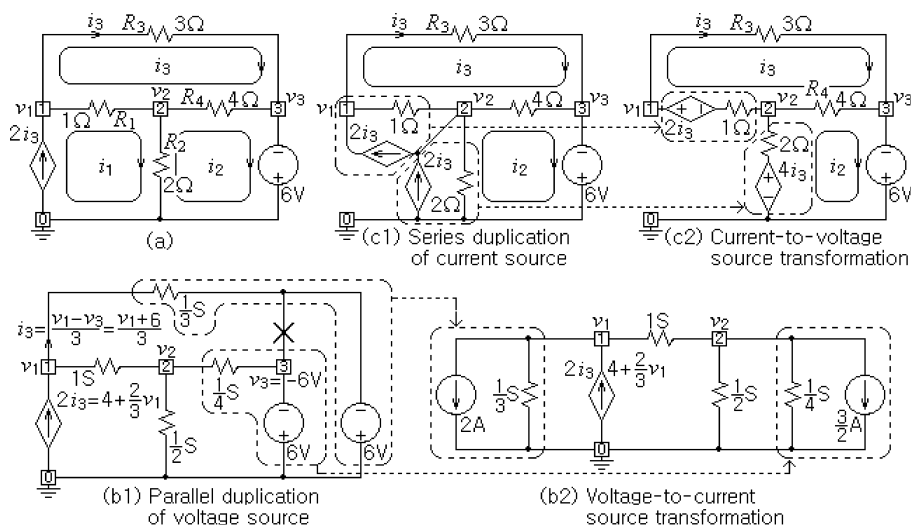


Figure 2.20 Circuit and its equivalents for Example 2.14

Now the source transformation method is tried. Noting that the 6 V voltage source has no series element, we duplicate it to make two copies in parallel and associate each of them with the $(1/4)$ S resistor and the $(1/3)$ S resistor, respectively, to transform the latter into a $6 \times 1/3 = 2$ A current source in parallel with the $(1/3)$ S resistor and the former into a $6 \times 1/4 = (3/2)$ A current source in parallel with the $(1/4)$ S resistor, as depicted in Figures 2.20(b1) and (b2). Then we apply formula (2.10) to set up the node equation for the equivalent circuit in Figure 2.20(b2) and solve it as

$$\begin{bmatrix} 1/3 + 1 & -1 \\ -1 & 1 + 1/2 + 1/4 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} -2 + 2(v_1 + 6)/3 \\ -1.5 \end{bmatrix}; \quad \begin{bmatrix} 2 & -3 \\ -4 & 7 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} 6 \\ -6 \end{bmatrix}; \quad \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} 12 \\ 6 \end{bmatrix} \quad (\text{E2.14.4})$$

(b) Mesh Analysis

The controlling variable of the $2i_3$ current source is already expressed in terms of the mesh currents. Noting that the current source is contained exclusively in mesh 1 so that it determines the mesh current $i_1 = 2i_3$, the mesh equation can be set up without thinking of ‘supermesh’ in terms of having no current source:

$$\begin{bmatrix} \times & \times & \times \\ -2 & 2 + 4 & -4 \\ -1 & -4 & 3 + 1 + 4 \end{bmatrix} \begin{bmatrix} i_1 = 2i_3 \\ i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} \times \\ 6 \\ 0 \end{bmatrix}; \quad \begin{bmatrix} 6 & -8 \\ -4 & 6 \end{bmatrix} \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} 6 \\ 0 \end{bmatrix}; \quad \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} 9 \\ 6 \end{bmatrix} \quad (\text{E2.14.5})$$

Now the source transformation method is tried. Noting that the $2i_3$ current source has no parallel element, it is duplicated to make two copies in series and each of them is associated with the 1Ω resistor and the 2Ω resistor, respectively, to transform the latter into a $2 \times 2i_3 = 4i_3$ [V] voltage source in series with the 2Ω resistor and the former into a $1 \times 2i_3 = 2i_3$ [V] voltage source in series with the 1Ω resistor, as depicted in Figures 2.20(c1) and (c2). Then we apply formula (2.12) to set up the mesh equation for the equivalent circuit in Figure 2.20(c2) and solve it as

$$\begin{bmatrix} 2 + 4 & -4 \\ -4 & 1 + 3 + 4 \end{bmatrix} \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} 4i_3 + 6 \\ 2i_3 \end{bmatrix}; \quad \begin{bmatrix} 6 & -8 \\ -4 & 6 \end{bmatrix} \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} 6 \\ 0 \end{bmatrix}; \quad \begin{bmatrix} i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} 9 \\ 6 \end{bmatrix} \quad (\text{E2.14.6})$$

If needed, the node voltage v_1 can be obtained by adding the voltage drop across the 3Ω resistor to the node voltage v_3 at node 3 as

$$v_1 = v_3 + R_3 i_{R_3} = -6 + 3i_3 \stackrel{(\text{E2.14.6})}{=} -6 + 3 \times 6 = 12 \text{ [V]} \quad (\text{E2.14.7})$$

and the node voltage v_2 can be obtained by adding the voltage drop across the 4Ω resistor to v_3 as

$$v_2 = v_3 + R_4 i_{R_4} = -6 + 4(i_2 - i_3) \stackrel{(\text{E2.14.6})}{=} -6 + 4 \times (9 - 6) = 6 \text{ [V]} \quad (\text{E2.14.8})$$

This result naturally agrees with the solution, (E2.14.3) or (E2.14.4), of the node equation.

(Example 2.15) A Circuit with Three Branches and Two Nodes

Consider the circuit of Figure 2.21(a). With the two 3 [V] and $2i_{R_2}$ [V] voltage sources short-circuited and the 3A current source open-circuited, the numbers of branches and nodes are $b = 3$ and $n = 2$, respectively, which implies that the numbers of node equations and mesh equations are

$$n - 1 = 2 - 1 = 1 \quad \text{and} \quad b - (n - 1) = 3 - (2 - 1) = 2 \quad (\text{E2.15.1})$$

Although the node analysis is expected to be better in terms of the number of equations, let us try both the node analysis and the mesh analysis for practice.

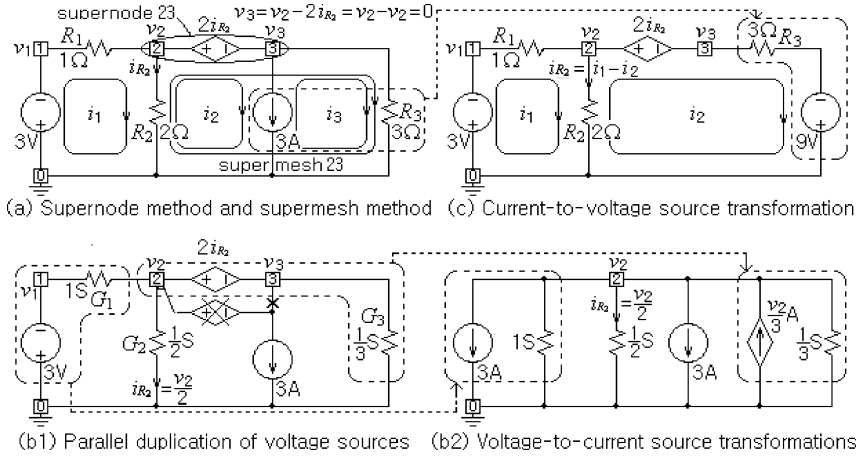


Figure 2.21 Circuit and its equivalents for Example 2.15

(a) Node Analysis

The controlling variable of the $2i_{R_2}$ [V] source is first expressed in terms of the node voltages as

$$i_{R_2} = v_2/2 \quad (\text{E2.15.2})$$

The 3V source connected to the reference node at one end determines the node voltage $v_1 = -3$ [V] at the other end node 1, while the $2i_{R_2}$ [V] source connecting the nonreference nodes 2 and 3 qualifies them to be in a group to be treated as a supernode. Thus the supernode method is used; i.e. KCL is applied to that supernode, which is the one and only nonreference node, and the node equation is written as

$$G_1 (v_2 - v_1) + G_2 v_2 + G_3 v_3 = 1(v_2 - v_1) + \frac{1}{2} v_2 + \frac{1}{3} v_3 = -3, \quad v_1 = -3 \rightarrow 9v_2 + 2v_3 = -36 \quad (\text{E2.15.3})$$

Together with one more equation provided by the $2i_{R_2}$ [V] source (coupling nodes 2 and 3), as

$$v_2 - v_3 = 2i_{R_2} \stackrel{(\text{E2.15.2})}{=} 2(v_2/2) = v_2; \quad v_3 = v_2 - v_2 = 0 \quad (\text{E2.15.4})$$

Equation (E2.15.3) yields

$$9v_2 \stackrel{(\text{E2.15.3}), (\text{E2.15.4})}{=} -36; \quad v_2 = -4 \text{ [V]} \quad (\text{E2.15.5})$$

Now the source transformation method is tried. The 3V source in series with the 1S resistor can easily be transformed into a $1 \times 3 = 3$ A source in parallel with the 1S resistor, as depicted in Figure 2.21(b2). Noting that the $2i_{R_2}$ [V] source has no series element, it is duplicated to make two copies in parallel and each of them is associated with the 3A source and the (1/3) S resistor, respectively, to transform the latter into a $(1/3) \times 2i_{R_2} \stackrel{(\text{E2.15.2})}{=} (v_2/3)$ A source in parallel with the (1/3) S resistor, while the former is removed because of being connected in series with a current source (see Section 1.5.1.5), as depicted in Figures 2.21(b1) and (b2). Then formula (2.10) is applied to set up the node equation for the equivalent circuit in Figure 2.21(b2), which is solved as

$$\left(1 + \frac{1}{2} + \frac{1}{3}\right)v_2 = \frac{v_2}{3} - 3 - 3; \quad \frac{3}{2}v_2 = -6; \quad v_2 = -4 \text{ [V]} \quad (\text{E2.15.6})$$

(b) Mesh Analysis

The controlling variable of the $2i_{R_2}$ [V] source is first expressed in terms of the mesh currents as

$$i_{R_2} = i_1 - i_2 \quad (\text{E2.15.7})$$

Noting that the two meshes 2 and 3 share the 3A source, the two meshes are combined into a supermesh, say 23, as depicted in Figure 2.21(a). Then KVL is applied to mesh 1 and the supermesh 23 to write the mesh equations as

$$\begin{aligned} R_1 i_1 + R_2(i_1 - i_2) &= i_1 + 2(i_1 - i_2) = -3 \\ R_2(i_2 - i_1) + R_3 i_3 &= 2(i_2 - i_1) + 3 i_3 = -2i_{R_2} \stackrel{(\text{E2.15.7})}{=} -2(i_1 - i_2) \end{aligned} \quad (\text{E2.15.8})$$

Together with one more equation provided by the 3 A source (coupling meshes 2 and 3) as

$$i_2 - i_3 = 3; \quad i_3 = i_2 - 3 \quad (\text{E2.15.9})$$

Equation (E2.15.8) can be rewritten in matrix vector form and solved as

$$\begin{bmatrix} 1+2 & -2 \\ -2+2 & 2+3-2 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} \stackrel{(\text{E2.15.8}, \text{E2.15.9})}{=} \begin{bmatrix} -3 \\ 9 \end{bmatrix}; \quad \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \frac{1}{3 \times 3} \begin{bmatrix} 3 & 2 \\ 0 & 3 \end{bmatrix} \begin{bmatrix} -3 \\ 9 \end{bmatrix} = \begin{bmatrix} 1 \\ 3 \end{bmatrix} \quad (\text{E2.15.10})$$

Now the source transformation method is tried. Noting that the 3A source has the 3Ω resistor in parallel, it is transformed into a $3\text{A} \times 3\Omega = 9\text{V}$ source in series with the 3Ω resistor, as depicted in Figure 2.21(c). Then formula (2.12) is applied to set up the mesh equation for the equivalent circuit in Figure 2.21(c) as

$$\begin{bmatrix} 1+2 & -2 \\ -2 & 2+3 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} -3 \\ -2i_{R_2} + 9 \end{bmatrix} \stackrel{(\text{E2.15.7})}{=} \begin{bmatrix} -3 \\ -2(i_1 - i_2) + 9 \end{bmatrix}; \quad \begin{bmatrix} 3 & -2 \\ -2+2 & 2+3-2 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} -3 \\ 9 \end{bmatrix} \quad (\text{E2.15.11})$$

which will yield the same result as obtained with (E2.15.10).

If needed, the node voltage v_2 can be obtained by subtracting the voltage drop across the 1Ω resistor from the node voltage $v_1 = -3\text{V}$ at node 1 as

$$v_2 = v_1 - R_1 i_{R_1} = -3 - 1i_1 \stackrel{(\text{E2.15.10})}{=} -3 - 1 \times 1 = -4\text{V} \quad (\text{E2.15.12})$$

This result naturally agrees with the solution, (E2.15.5) or (E2.15.6), of the node equation.

(Example 2.16) A Circuit with Three Branches and Three Nodes

Consider the circuit of Figure 2.22(a). With the voltage source short-circuited and the current sources open-circuited, the numbers of branches and nodes are $b = 3$ and $n = 3$, respectively, which implies that the numbers of node equations and mesh equations are

$$n - 1 = 3 - 1 = 2 \quad \text{and} \quad b - (n - 1) = 3 - (3 - 1) = 1 \quad (\text{E2.16.1})$$

Although the mesh analysis is expected to be better in terms of the number of equations, let us try both the node analysis and the mesh analysis for practice.

(a) Node Analysis

The controlling variable of the $2v_{12}$ [A] source is first expressed in terms of the node voltages as

$$v_{12} = v_1 - v_2 \quad (\text{E2.16.2})$$

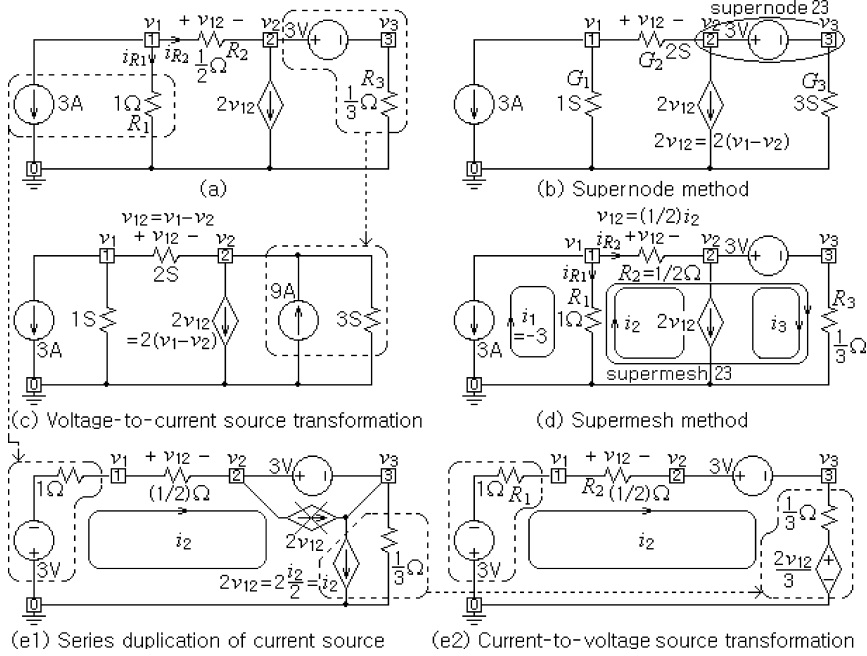


Figure 2.22 Circuit and its equivalents for Example 2.16

Noting that the $3V$ source connecting the nonreference nodes 2 and 3 qualifies them in a group to be treated as a supernode, say 23 (as depicted in Figure 2.22(b)), the supernode method is used; i.e. KCL is applied to node 1 and the supernode 23 to write the node equations as

$$\begin{aligned} \frac{v_1}{R_1} + \frac{v_1 - v_2}{R_2} &= -3; & \frac{v_1}{1} + \frac{v_1 - v_2}{1/2} &= 3v_1 - 2v_2 = -3 \\ \frac{v_2 - v_1}{R_2} + \frac{v_3}{R_3} &= -2v_{12} \stackrel{(E2.16.2)}{=} -2(v_1 - v_2); & \frac{v_2 - v_1}{1/2} + \frac{v_3}{1/3} + 2(v_1 - v_2) &= 3v_3 = 0 \end{aligned} \quad (E2.16.3)$$

Together with one more equation provided by the $3V$ source (coupling nodes 2 and 3) as

$$v_2 - v_3 = 3; \quad v_2 = v_3 + 3 \quad (E2.16.4)$$

Equations (E2.16.3) yield

$$v_3 = 0, \quad v_2 \stackrel{(E2.16.4)}{=} v_3 + 3 \stackrel{(E2.16.3)}{=} 3; \quad \text{and} \quad v_1 \stackrel{(E2.16.3)}{=} \frac{1}{3}(2v_2 - 3) = 1 \quad (E2.16.5)$$

Now the source transformation method is tried. With the $3V$ source in series with the $(1/3)\Omega$ resistor transformed into a $3V/(1/3)\Omega = 9A$ source in parallel with the $(1/3)\Omega$ resistor, as depicted in Figure 2.22(c), formula (2.10) is applied to set up the node equation as

$$\begin{bmatrix} 1 + 2 & -2 \\ -2 & 2 + 3 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} -3 \\ 9 - 2v_{12} \end{bmatrix} \stackrel{(E2.16.2)}{=} \begin{bmatrix} -3 \\ 9 - 2(v_1 - v_2) \end{bmatrix}; \quad \begin{bmatrix} 3 & -2 \\ 0 & 3 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} -3 \\ 9 \end{bmatrix} \quad (E2.16.6)$$

which yields

$$\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \frac{1}{9} \begin{bmatrix} 3 & 2 \\ 0 & 3 \end{bmatrix} \begin{bmatrix} -3 \\ 9 \end{bmatrix} = \begin{bmatrix} 1 \\ 3 \end{bmatrix} \quad (\text{E2.16.7})$$

(b) Mesh Analysis

After labeling the three mesh currents i_1 , i_2 , and i_3 , as depicted in Figure 2.22(d), the controlling variable of the $2v_{12}$ [A] source is expressed in terms of the mesh currents as

$$v_{12} = R_2 i_{R_2} = (1/2) i_2 \quad (\text{E2.16.8})$$

The 3 A source contained exclusively in mesh 1 determines the mesh current $i_1 = -3$ A. Noting that meshes 2 and 3 share the $2v_{12}$ [A] source, the two meshes are combined into a supermesh, say 23, as depicted in Figure 2.22(d). KVL can be applied to the supermesh to write the mesh equation as

$$R_1(i_2 - i_1) + R_2 i_2 + R_3 i_3 = 1(i_2 - (-3)) + \frac{1}{2} i_2 + \frac{1}{3} i_3 = -3; \quad 9 i_2 + 2 i_3 = -36 \quad (\text{E2.16.9})$$

Together with one more equation provided by the $2v_{12}$ [A] source (coupling meshes 2 and 3), as

$$i_2 - i_3 = 2v_{12} \stackrel{(\text{E2.16.8})}{=} i_2; \quad i_3 = i_2 - i_2 = 0 \quad (\text{E2.16.10})$$

Equation (E2.16.9) yields

$$i_2 = \frac{1}{9}(-2 i_3 - 36) \stackrel{(\text{E2.16.10})}{=} -4 \quad (\text{E2.16.11})$$

Now the source transformation method is tried. Noting that the $2v_{12}$ [A] source has no parallel element, it is duplicated to make two copies in series, each of which is associated with the 3 V source and the $(1/3) \Omega$ resistor, respectively, to transform the latter into a $(1/3) \Omega \times 2v_{12}$ [A] = $(2/3)v_{12}$ [V] source in series with the $(1/3) \Omega$ resistor, the former being removed because of being connected in parallel with a voltage source, as depicted in Figures 2.22(e1) and (e2). Then formula (2.12) is applied to set up the mesh equation for the equivalent circuit in Figure 2.22(e2), which is solved as

$$(1 + \frac{1}{2} + \frac{1}{3}) i_2 = -3 - 3 + \frac{2}{3} v_{12} \stackrel{(\text{E2.16.8})}{=} -6 + \frac{1}{3} i_2, \quad \frac{3}{2} i_2 = -6; \quad i_2 = -4 \quad (\text{E2.16.12})$$

If needed, the node voltage v_1 can be obtained by subtracting the voltage drop across the 1Ω resistor and the 3 V source from $v_0 = 0$, which is the voltage at node 0 (in Figure 2.22(e2)), as

$$v_1 = 0 - 3 - R_1 i_{R_1} = -3 - 1 i_2 \stackrel{(\text{E2.16.12})}{=} -3 - 1(-4) = 1 \text{ [V]} \quad (\text{E2.16.13})$$

The same result can be obtained by finding the voltage across the 1Ω resistor in Figure 2.22(d), as

$$v_1 = R_1 i_{R_1} = 1(i_1 - i_2) \stackrel{(\text{E2.16.11})}{=} -3 - (-4) = 1 \text{ [V]} \quad (\text{E2.16.14})$$

The node voltage v_2 can be obtained by subtracting the voltage drop across R_2 from v_1 as

$$v_2 = v_1 - R_2 i_{R_2} = 1 - \frac{1}{2} i_2 \stackrel{(\text{E2.16.11})}{=} 1 - \frac{1}{2}(-4) = 3 \text{ [V]} \quad (\text{E2.16.15})$$

These results naturally agree with the solution, (E2.16.5) or (E2.16.7), of the node equation.

2.7 Thevenin/Norton Equivalent Circuits

Thevenin's theorem was first discovered by German scientist Hermann von Helmholtz in 1853 and was rediscovered in 1883 by French telegraph engineer Léon Charles Thévenin (1857–1926). It says that any network consisting of linear elements and independent/dependent sources may be replaced at a pair of its terminals (nodes) by the Thevenin equivalent circuit, which consists of a single element of impedance Z_{Th} in series with a single independent voltage source V_{Th} (see Figure 2.23(b)), where the values of Z_{Th} and V_{Th} are determined as follows:

T1. Thevenin equivalent voltage source V_{Th} :

The *open-circuit voltage* across the terminals, i.e. the voltage across the open terminals a – b (with $Z_L = \infty$)

T2. Thevenin equivalent impedance Z_{Th} :

The equivalent impedance of the circuit (with all the independent sources removed) seen from the terminals, where the impedance is the 'generalized' resistance.

Norton's theorem, published in 1926 by Bell Labs engineer Edward Lawry Norton (1898–1983), says that any linear network may be replaced at a pair of its terminals by the Norton equivalent circuit, which consists of a single element of impedance Z_{Nt} in parallel with a single independent current source I_{Nt} (see Figure 2.23(c)), where the values of Z_{Nt} and I_{Nt} are determined as follows:

N1. Norton equivalent current source I_{Nt} :

The *short-circuit current* through the terminals, i.e. the current through the shorted terminals a – b (with $Z_L = 0$)

N2. Norton equivalent impedance Z_{Nt} :

The equivalent impedance of the circuit (with all the independent sources removed) seen from the terminals

Since Thevenin and Norton equivalents are equivalent in representing a linear circuit seen from a pair of two terminals, one can be obtained from the other by using the source transformation introduced in Section 1.5.2. This suggests another formula for finding the equivalent impedance as

$$Z_{Th} = Z_{Nt} = \frac{V_{Th}}{I_{Nt}} = \frac{V_{oc}(\text{the open-circuit voltage})}{I_{sc}(\text{the short-circuit current})} \quad (2.13)$$

Note that the equivalent impedance should be found after removing every independent source, i.e. by short-circuiting/open-circuiting every voltage/current source. For networks having no dependent source, the series/parallel combination and Δ -Y/Y- Δ conversion formulas often suffice for the purpose of

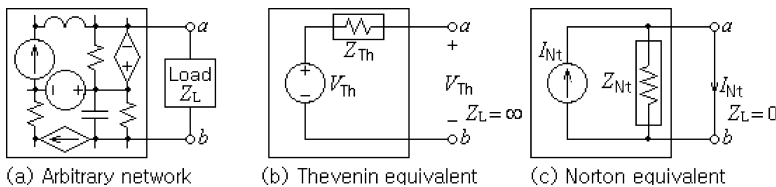


Figure 2.23 Thevenin and Norton equivalent circuits

finding the equivalent impedance. For networks having dependent sources, we can apply a test voltage source V_T across the terminals, measure the current I_T through the terminals (see Figure 2.24(a)), and write the relationship between V_T and I_T as

$$I_T = \frac{V_T - V_{Th}}{Z_{Th}} \rightarrow V_T = Z_{Th} I_T + V_{Th} \quad (2.14)$$

In this relationship, the value of the equivalent impedance Z_{Th} is found from the proportionality coefficient of the term in I_T and the equivalent source V_{Th} from the constant term independent of I_T . This suggests a *one-shot method* of finding the values of the equivalent impedance Z_{Th} and the equivalent source V_{Th} or I_{Nt} at a time. This one-shot method of finding the equivalent may be applied for any linear networks with or without independent/dependent sources. Another way to find the equivalent circuit is to apply a test current source I_T through the terminals (see Figure 2.24(b)), measure the voltage V_T across the terminals, and write down the relationship between V_T and I_T , as given in Equation (2.14).

The equivalents may be used to simplify portions of a network by making the analysis easy and efficient since they allow us to focus on the terminal behavior of each portion without paying attention to its internal characteristic. They may save considerable time and effort in the case of adjustable load, where there is a need to find the terminal voltage or current of a given linear network with several different values of load impedance. They are almost indispensable for the load line analysis of a linear network with a nonlinear load (see Problem 2.29).

[Remark 2.1] Thevenin and Norton Equivalents

1. Taking account of the fact that practical circuits have some degree of nonlinearity, the Thevenin equivalent is better for modeling circuits having an equivalent impedance smaller than the external load ($Z_{Th} < Z_L$), while the Norton equivalent is better for modeling circuits having an equivalent impedance larger than the external load ($Z_{Nt} > Z_L$).
2. The choice between the two methods, the test current source method (depicted in Figure 2.24(a)) and the test voltage source method (depicted in Figure 2.24(b)), depends mainly on which one of the node analysis and the mesh (loop) analysis is used to find the relationship between V_T and I_T .

(Example 2.17) An Experimental Method of Finding an Equivalent Circuit

Suppose there is a voltage source of adjustable value, a voltmeter, and an ammeter. How can the Thevenin equivalent be found of a given network seen from a pair of its terminals a - b ?

(Answer) With the voltage source applied across the terminals, a - b and the voltage across and the current through the terminals should be measured two times, each for two different values of the voltage source, so that two voltage–current pairs (V_{T1}, I_{T1}) and (V_{T2}, I_{T2}) , can be found that are supposed to satisfy Equation (2.14) as

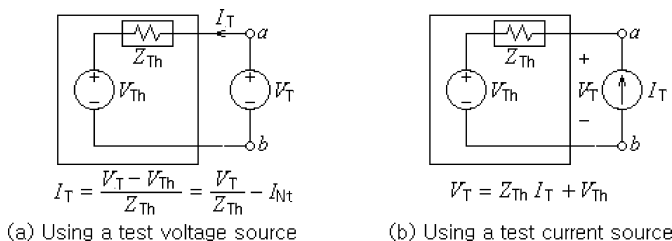


Figure 2.24 One-shot method for obtaining a Thevenin equivalent circuit

$$V_{T1} = Z_{Th} I_{T1} + V_{Th} \quad (E2.17.1)$$

$$V_{T2} = Z_{Th} I_{T2} + V_{Th} \quad (E2.17.2)$$

Then the equivalent impedance can be found from the difference between these two relations as

$$V_{T1} - V_{T2} = Z_{Th} (I_{T1} - I_{T2}); \quad Z_{Th} = \frac{V_{T1} - V_{T2}}{I_{T1} - I_{T2}} \quad (E2.17.3)$$

which is substituted into one of the above two relations, say Equation (E2.17.1), in order to find the value of the equivalent voltage source as

$$V_{Th} = V_{T1} - Z_{Th} I_{T1} \quad (E2.17.4)$$

(Example 2.18) Thevenin Equivalent of a Bridge Network

(a) Thevenin Equivalent Circuit of the Bridge Network in Figure 2.25(a1) Seen from Terminals 2 and 3

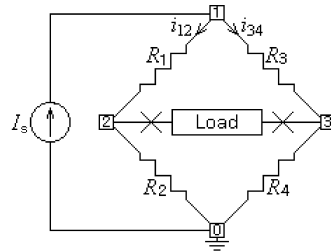
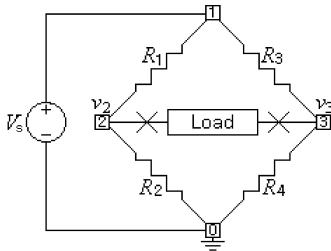
Since the voltages at nodes 2 and 3 are determined by the voltage divider rule (2.4), the open-circuit voltage across terminals 2 and 3 can be obtained as

$$V_{Th} = v_2 - v_3 = \frac{R_2}{R_1 + R_2} V_s - \frac{R_4}{R_3 + R_4} V_s = \frac{R_2 R_3 - R_1 R_4}{(R_1 + R_2)(R_3 + R_4)} V_s \quad (E2.18.1)$$

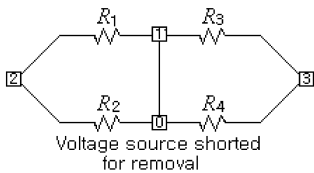
which is the value of the Thevenin equivalent voltage source.

To find the equivalent impedance, the voltage source is removed (deactivated) by short-circuiting it as depicted in Figure 2.25(b1). Then the parallel/series combination formulas are used to get the resistance between the two terminals 2 and 3 as

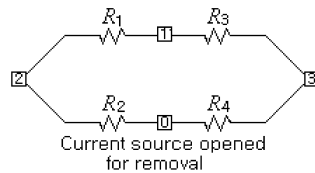
$$R_{Th} = R_1 \parallel R_2 + R_3 \parallel R_4 = \frac{R_1 R_2}{R_1 + R_2} + \frac{R_3 R_4}{R_3 + R_4} \quad (E2.18.2)$$



(a1) Bridge circuit excited by a voltage source (a2) Bridge circuit excited by a current source



(b1) Equivalent resistance



(b2) Equivalent resistance

Figure 2.25 Bridge circuits for Example 2.18

(b) Thevenin Equivalent Circuit of the Bridge Network in Figure 2.25(a2) Seen from Terminals 2 and 3

Since the currents through R_1 – R_2 and R_3 – R_4 are determined by the current divider rule (2.6), the open-circuit voltage across terminals 2 and 3 can be obtained as

$$\begin{aligned} V_{Th} &= v_2 - v_3 = R_2 i_{12} - R_4 i_{34} \\ &= \frac{R_2(R_3 + R_4)I_s}{(R_1 + R_2) + (R_3 + R_4)} - \frac{R_4(R_1 + R_2)I_s}{(R_1 + R_2) + (R_3 + R_4)} \\ &= \frac{R_2 R_3 - R_1 R_4}{(R_1 + R_2) + (R_3 + R_4)} I_s \end{aligned} \quad (\text{E2.18.3})$$

which is the value of the Thevenin equivalent voltage source.

To find the equivalent impedance, the current source is removed (deactivated) by open-circuiting it as depicted in Figure 2.25(b2). Then the series/parallel combination formulas are used to find the resistance between the two terminals 2 and 3 as

$$R_{Th} = (R_1 + R_3) \parallel (R_2 + R_4) = \frac{(R_1 + R_3)(R_2 + R_4)}{R_1 + R_2 + R_3 + R_4} \quad (\text{E2.18.4})$$

(Example 2.19) Thevenin Equivalent of a Network Having a Dependent Source

Let us find the Thevenin equivalent of the network of Figure 2.26(a) seen from the terminals a – b two times, once by using the test voltage source method and once by using the test current source method.

(a) Test Voltage Source Method

With the test voltage source V_T applied across the terminals a – b as depicted in Figure 2.26(a), we may well use the mesh analysis. The βi_B [A] source in parallel with R_E can be transformed into a $\beta R_E i_B$ [V] source in series with R_E , as depicted in Figure 2.26(b). Then, noting that the controlling variable, i_B , is the same as the mesh current, the mesh current is labeled i_B and the mesh equation is set up as

$$(R_B + R_E)i_B = V_T - \beta R_E i_B \quad (\text{E2.19.1})$$

$$V_T = (R_B + R_E + \beta R_E)i_B = [R_B + (\beta + 1)R_E]I_T \quad (\text{E2.19.2})$$

Matching this relation with Equation (2.14) yields

$$Z_{Th} = R_B + (\beta + 1)R_E \quad (\text{E2.19.3})$$

(b) Test Current Source Method

With the test current source I_T applied through the terminals a – b as depicted in Figure 2.26(c), we may well use the node analysis. Noting that the controlling variable, i_B , is the same as I_T , the node equation is set up and solved as

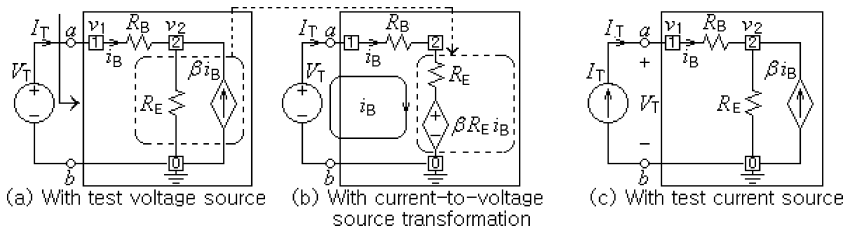


Figure 2.26 To find the equivalent circuit for Example 2.19

$$\begin{bmatrix} 1/R_B & -1/R_B \\ -1/R_B & 1/R_B + 1/R_E \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} I_T \\ \beta i_B \end{bmatrix} = \begin{bmatrix} I_T \\ \beta I_T \end{bmatrix} \quad (\text{E2.19.4})$$

$$\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \frac{I_T}{1/(R_B R_E)} \begin{bmatrix} 1/R_B + 1/R_E & 1/R_B \\ 1/R_B & 1/R_B \end{bmatrix} \begin{bmatrix} 1 \\ \beta \end{bmatrix} = \begin{bmatrix} R_B + (\beta + 1)R_E \\ (\beta + 1)R_E \end{bmatrix} I_T \quad (\text{E2.19.5})$$

Solving this equation for $v_1 = V_T$ yields the same result as obtained in (a):

$$V_T = v_1 = [R_B + (\beta + 1)R_E]I_T \quad (\text{E2.19.6})$$

Note the following about the node analysis of the circuit shown in Figure 2.26(c). As mentioned in Remark 1.3, R_B in series with the (test) current source can be removed (short-circuited) without making any difference in the analysis of the rest of the circuit. Thus only one node equation in v_2 is required, which can be written (by applying KCL to node 2) as

$$\frac{v_2}{R_E} = I_T + \beta i_B = I_T + \beta I_T; \quad v_2 = (\beta + 1)R_E I_T \quad (\text{E2.19.7})$$

The voltage, v_1 , at terminal a (node 1) is obtained by adding the voltage drop across R_B to the voltage, v_2 , at node 2 as

$$v_1 = v_2 + R_B i_B \stackrel{(\text{E2.19.7})}{=} (\beta + 1)R_E I_T + R_B I_T = [R_B + (\beta + 1)R_E]I_T \quad (\text{E2.19.8})$$

which naturally agrees with Equation (E2.19.2) or (E2.19.6). It would be simpler to just add the series resistance R_B to the equivalent resistance, $(\beta + 1)R_E$, seen from the terminals 2 and 0, which is obtained from Equation (E2.19.7).

(Example 2.20) Thevenin Equivalent of a Circuit Having a Dependent Source

As in Example 2.19, find the Thevenin equivalent seen from terminals 1 and 2 of the circuit containing a dependent source in Figure 2.27(a). Note that the right part of the circuit consisting of the 90 V voltage source and two resistors of 15 Ω and 10 Ω (with the switch closed) can be replaced equivalently with the rightmost part of the circuit consisting of a 36 V source and a 6 Ω resistor with the transfer switch connected to position b as depicted in Figure 2.27(b3).

(a) Apply a test voltage source V_T to terminals 1 and 2, as depicted in Figure 2.27(b1), and find the relationship between V_T and the current I_T (through V_T) using the mesh analysis. In this scheme, two copies of the $v_{23}/2$ [A] current source in series are made and each of them is associated with 4 Ω and V_T , respectively (Figure 2.27(b2)). For the moment, there is no need to pay any attention to the one associated with the voltage source V_T because any element in parallel with a voltage source can be neglected without making any difference in the analysis results of the rest of the circuit, as stated in Remark 1.3. The other associated with 4 Ω is transformed into a $v_{23}/2 \times 4 = 2v_{23}$ [V] voltage source in series with 4 Ω as depicted in Figure 2.27(b3). Consequently, for the two cases where the switch is opened and closed, the mesh equations are set up and solved as

$$(4 + 1 + 15)i_2 = 2v_{23} + V_T + 90 \stackrel{v_{23}=R_1 i_2=i_2}{\rightarrow} V_T = 18i_2 - 90 \stackrel{i_2=2I_T}{=} 36I_T - 90 \quad (\text{E2.20.1a})$$

$$\text{with } v_{23} = R_1 i_2 = i_2, \quad i_2 = I_T + \frac{1}{2}v_{23} = I_T + \frac{1}{2}i_2; \quad i_2 = 2I_T; \quad (\text{E2.20.2})$$

$$(4 + 1 + 6)i_2 = 2v_{23} + V_T + 36 \stackrel{v_{23}=R_1 i_2=i_2}{\rightarrow} V_T = 9i_2 - 36 \stackrel{i_2=2I_T}{=} 18I_T - 36 \quad (\text{E2.20.1b})$$

Note that account was taken of the neglected $v_{23}/2$ [A] source in parallel with V_T (in Figure 2.27(b1)) to get the relation (E2.20.2) between the mesh current i_2 and the (test) current I_T through V_T . These results imply that the Thevenin equivalent of the circuit with the switch opened consists

of a $36\ \Omega$ resistor in series with a -90 V voltage source and that of the circuit with the switch closed consists of a $18\ \Omega$ resistor in series with a -36 V voltage source:

$$R_{Th1} = 36\ \Omega, \quad V_{Th1} = -90\text{ V}; \quad R_{Th2} = 18\ \Omega, \quad V_{Th2} = -36\text{ V} \quad (\text{E2.20.3})$$

- (b) Apply a test current source I_T through terminals 1 and 2 as depicted in Figure 2.27(c) and find the relationship between I_T and the voltage V_T (across I_T) using the node analysis. In this scheme, the two voltage sources of 90 V and 36 V (in Figure 2.27(b3)) are transformed into their equivalent current sources of $90/15 = 6\text{ A}$ in parallel with a $(1/15)\text{ S}$ resistor and $36/6 = 6\text{ A}$ in parallel with a $(1/6)\text{ S}$ resistor, as depicted in Figure 2.27(c). Consequently, for the two cases of the switch being opened and closed, the node equations are set up and solved as:

$$\begin{bmatrix} 1/4 & 0 & 0 \\ 0 & 1 & -1 \\ 0 & -1 & 1 + 1/15 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} -I_T \\ I_T + v_{23}/2 \\ -6 \end{bmatrix} \quad \text{with } v_{23} = v_2 - v_3 \quad (\text{E2.20.4a})$$

$$\begin{bmatrix} 1/4 & 0 & 0 \\ 0 & 1/2 & -1/2 \\ 0 & -1 & 1 + 1/15 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} -I_T \\ I_T \\ -6 \end{bmatrix}; \quad \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 4 & 0 & 0 \\ 0 & 32 & 15 \\ 0 & 30 & 15 \end{bmatrix} \begin{bmatrix} -I_T \\ I_T \\ -6 \end{bmatrix}$$

$$V_T = v_2 - v_1 = (32I_T - 15 \times 6) - 4 \times (-I_T) = 36I_T - 90 \quad (\text{E2.20.5a})$$

$$\begin{bmatrix} 1/4 & 0 & 0 \\ 0 & 1/2 & -1/2 \\ 0 & -1 & 1 + 1/6 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} -I_T \\ I_T \\ -6 \end{bmatrix}; \quad \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 4 & 0 & 0 \\ 0 & 14 & 6 \\ 0 & 12 & 6 \end{bmatrix} \begin{bmatrix} -I_T \\ I_T \\ -6 \end{bmatrix} \quad (\text{E2.20.4b})$$

$$V_T = v_2 - v_1 = (14I_T - 6 \times 6) - 4 \times (-I_T) = 18I_T - 36 \quad (\text{E2.20.5b})$$

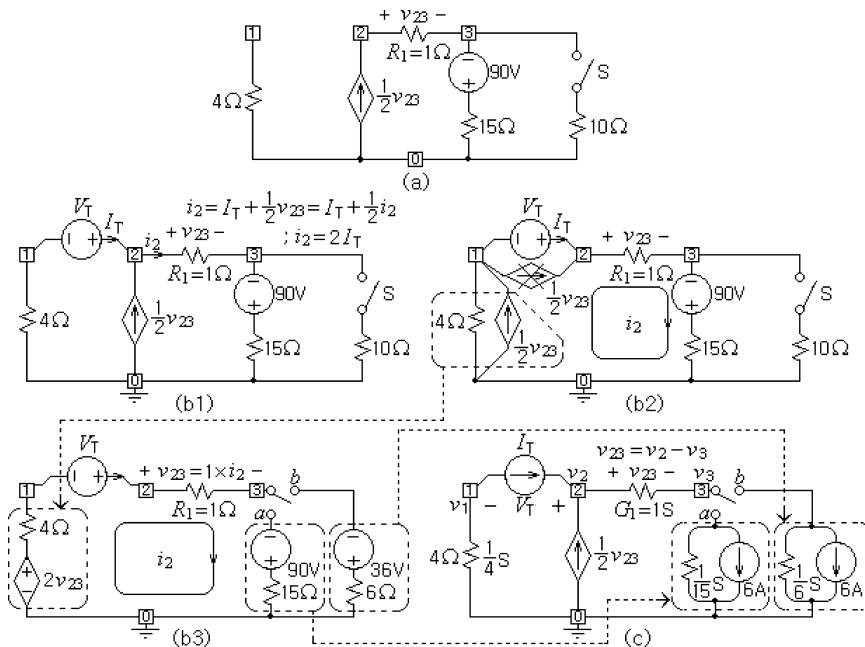


Figure 2.27 Thevenin equivalent of a circuit containing a dependent source for Example 2.20

This can be solved by using MATLAB (Appendix I) as follows:

```
>>syms IT
>>Y=[1/4 0 0; 0 1/2 -1/2; 0 -1 1+1/15]; I=[-IT; IT; -6];
>>V=Y\I; VT=V(2)-V(1) % (E2.20.5a)
VT = 36*IT - 90
>>Y=[1/4 0 0; 0 1/2 -1/2; 0 -1 1+1/6]; I=[-IT; IT; -6];
>>V=Y\I; VT=V(2)-V(1) % (E2.20.5b)
VT = 18*IT - 36
```

As expected, this is the same result as in (a).

Note. As stated in Remark 1.3, the 4Ω resistor in series with the current source I_T (in Figure 2.27(c)) can be neglected without making any difference to the analysis results of the rest of the circuit. If the 4Ω resistor is removed by short-circuiting it, then we get

$$V_T = v_2 = 32I_T - 15 \times 6 = 32I_T - 90 \text{ (with the SW (switch) connected to position } a \text{)} \quad (\text{E2.20.5a})$$

$$V_T = v_2 = 14I_T - 6 \times 6 = 14I_T - 36 \text{ (with the SW connected to position } b \text{)} \quad (\text{E2.20.5b})$$

implying that the partial equivalent resistance is 32Ω or 14Ω . Then the neglected resistor 4Ω is added to get 36Ω or 18Ω , which agrees with the above result.

(Example 2.21) Thevenin Equivalent of a Circuit Having Dependent Sources

Find the Thevenin equivalent seen from terminals 1 to 3 of the circuit in Figure 2.28(a).

- (a) Let us apply a test voltage source V_T to terminals 1 to 3 as depicted in Figure 2.28(b1) and find the relationship between V_T and the current I_T (through V_T) using the mesh analysis. At first, to avoid the (current-to-voltage) source transformation, KVL is applied individually around mesh 1 and supermesh 23 (containing the i_{12} [A] source) to write the mesh equations (see Figure 2.28(b2)), which yields

$$\left. \begin{aligned} 1(i_1 - i_3) &= V_s - v_3 = V_s - 1i_2; & i_1 + i_2 - i_3 &= V_s \\ 1(i_3 - i_1) + 1i_2 &= V_T + v_3 = V_T + 1i_2; & -i_1 + i_3 &= V_T \end{aligned} \right\} \rightarrow I_T = i_3 = 2V_T + V_s \quad (\text{E2.21.1})$$

$$i_2 - i_3 = i_{12} = i_1 - i_3; \quad i_1 = i_2$$

$$V_T = \frac{1}{2}I_T - \frac{V_s}{2} \quad (\text{E2.21.2})$$

This means that the Thevenin equivalent impedance and voltage source are $(1/2)\Omega$ and $-V_s/2$, respectively.

Now, in order to set up the mesh equations from visual inspection, it is advisable to make the effort of transforming the i_{12} [A] source into the voltage source(s). To do so, it is duplicated to make two copies in series so that each one can be associated with the (right) resistor $R_2 = 1\Omega$ and the (left) v_3 [V] source, respectively, as depicted in Figure 2.28(b3). (They might be associated with the voltage source V_T and the (left) resistor R_1 , but it would trespass on I_T , the current through V_T that needs to be found, which is not desirable.) The one associated with the v_3 [V] source can be neglected without affecting the analysis result of the rest part (Remark 1.3) and the other associated with $R_2 = 1\Omega$ can be transformed into a $1\Omega \times i_{12}$ [A] = i_{12} [V] source in series with 1Ω , as depicted in Figure 2.28(b4). Thus the mesh equation can be set up and solved as follows:

$$\begin{bmatrix} 1 & -1 \\ -1 & 1+1 \end{bmatrix} \begin{bmatrix} i_1 \\ i_{23} \end{bmatrix} = \begin{bmatrix} V_s - v_3 \\ v_3 + V_T - i_{12} \end{bmatrix} = \begin{bmatrix} V_s - (1i_{23} + i_{12}) \\ (1i_{23} + i_{12}) + V_T - i_{12} \end{bmatrix} = \begin{bmatrix} V_s - (i_{23} + i_1 - i_{23}) \\ i_{23} + V_T \end{bmatrix} \quad (\text{E2.21.3})$$

$$\begin{bmatrix} 1+1 & -1 \\ -1 & 1+1-1 \end{bmatrix} \begin{bmatrix} i_1 \\ i_{23} \end{bmatrix} = \begin{bmatrix} V_s \\ V_T \end{bmatrix}; \quad \begin{bmatrix} i_1 \\ i_{23} \end{bmatrix} = \frac{1}{2 \times 1 - (-1)(-1)} \begin{bmatrix} 1 & 1 \\ 1 & 2 \end{bmatrix} \begin{bmatrix} V_s \\ V_T \end{bmatrix} = \begin{bmatrix} V_s + V_T \\ V_s + 2V_T \end{bmatrix} \quad (\text{E2.21.4})$$

$$I_T = i_{23} = V_s + 2V_T; \quad V_T = \frac{1}{2}I_T - \frac{V_s}{2}$$

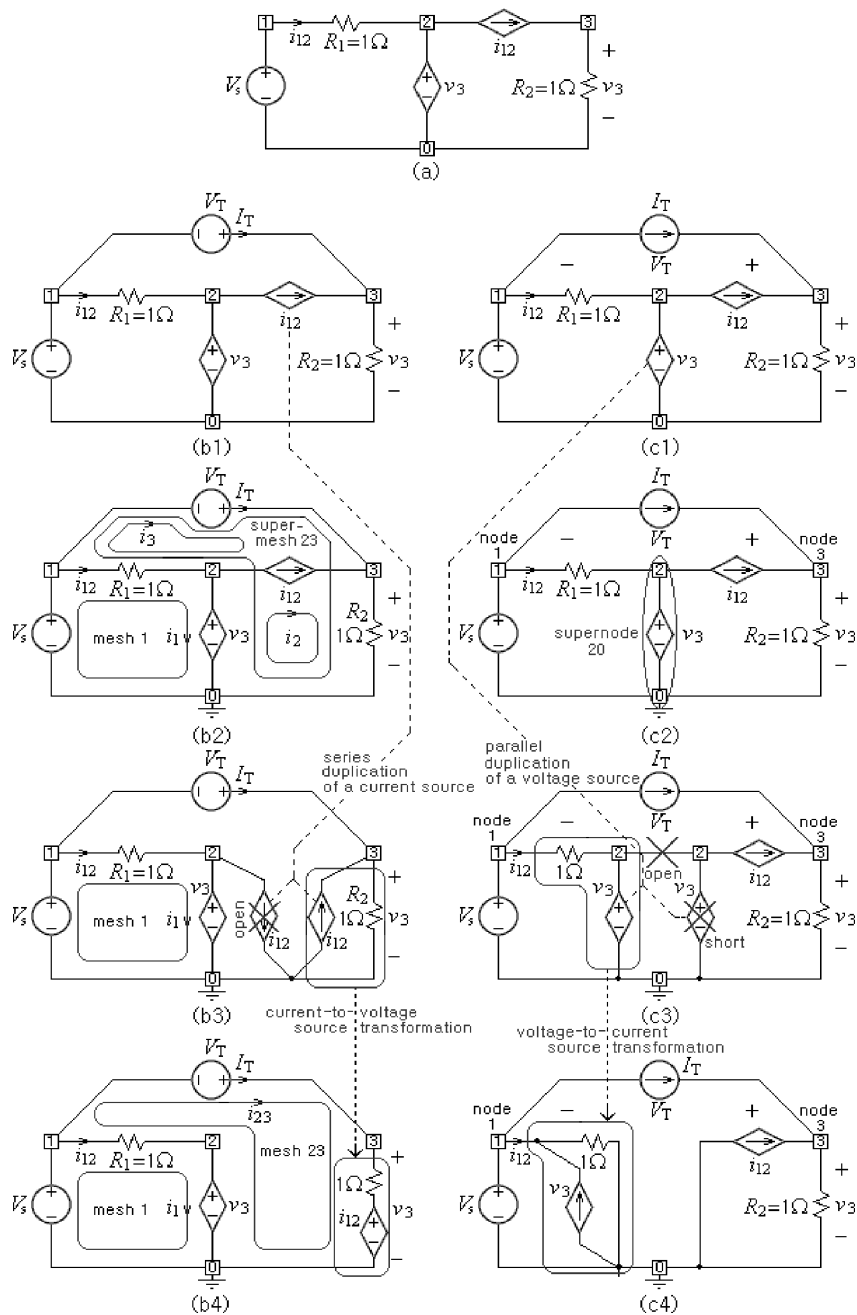


Figure 2.28 Thevenin equivalent of a circuit containing two dependent sources (Example 2.21)

Naturally, this is the same result as Equation (E2.21.2) obtained above.

- (b) Apply a test current source I_T through terminals 1 to 3 as depicted in Figure 2.28(c1) and find the relationship between the voltage V_T (across I_T) and I_T using the node analysis. At first, to avoid the (voltage-to-current) source transformation, KCL is applied to node 3 to write the node equation, which yields the same result:

$$\frac{v_3}{R_2} = I_T + i_{12} = I_T + \frac{V_s - v_3}{R_1}; \quad 2v_3 = I_T + V_s; \quad v_3 = \frac{1}{2}I_T + \frac{1}{2}V_s \quad (\text{E2.21.5})$$

$$V_T = v_3 - v_1 \stackrel{(\text{E2.21.5})}{=} \frac{1}{2}I_T + \frac{1}{2}V_s - V_s = \frac{1}{2}I_T - \frac{1}{2}V_s \quad (\text{E2.21.6})$$

(Question) Did KCL have to be applied to node 1 or supernode 20 in the circuit of Figure 2.28(c2)?

(Answer) No, because their node voltages are determined by the voltage sources as V_s and v_3 , respectively. Consequently, only one node equation (with a single unknown node voltage v_3) is needed.

Now, in order to set up the node equations from visual inspection, it is advisable to take the trouble of transforming the v_3 [V] voltage source into the current source(s). To do so, it is duplicated to make two copies in parallel so that each one can be associated with the (left) resistor $R_1 = 1 \Omega$ and the i_{12} current source, respectively, as depicted in Figure 2.28(c3). The one associated with the i_{12} [A] current source can be neglected without affecting the analysis result of the rest part (Remark 1.3) and the other associated with the resistor R_1 can be transformed into a $v_3/R_1 = v_3$ [A] current source, as depicted in Figure 2.28(c4). Then the node equation can be set up for node 3 and solved to get the same result.

Note. For this problem, this systematic approach using the source transformation has no advantage over the supernode method because the circuit has just a single node to which KCL should be applied.

2.8 Superposition Principle and Linearity

A circuit is said to be *linear* if the *superposition principle* holds in the sense that it satisfies the following properties:

1. *Additivity.* The output (such as the value of a voltage or current) of the circuit excited by more than one independent source is the algebraic sum of its outputs to each of the input sources applied individually.
2. *Homogeneity.* The output of the circuit excited by a single independent source is proportional to the input source.

This superposition principle holds for any linear circuit containing inductors, capacitors, and dependent sources as well as resistors and independent sources.

Based on this superposition principle, the output of a linear circuit containing several independent sources can be found by breaking down the circuit into simpler subcircuits, each of which has only one source, solving the individual subcircuits, and adding the outputs of all the subcircuits. This superposition approach allows the relative contributions from several sources to be compared. Let us take a look at the following example.

(Example 2.22) Superposition Principle for a Linear Circuit

Consider the circuit of Figure 2.8(a) or Figure 2.29(a), which was dealt with in Example 2.3. Let us solve this circuit again for the node voltage v_2 by using the superposition principle. Figures 2.29(b), (c), and (d) show the subnetworks, each with one of the three sources of 5 V, 4 A, and 1 V, respectively. The node voltage v_2 for each subnetwork is

$$\text{For the subnetwork in (b),} \quad v_{2b} = 5 \times \frac{1/5}{1 + 1/5} = \frac{5}{6} \quad (\text{voltage divider}) \quad (\text{E2.22.1})$$

$$\text{For the subnetwork in (c),} \quad (1 + 2 + 3)v_{2c} = 4; \quad v_{2c} = \frac{2}{3} \quad (\text{E2.22.2})$$

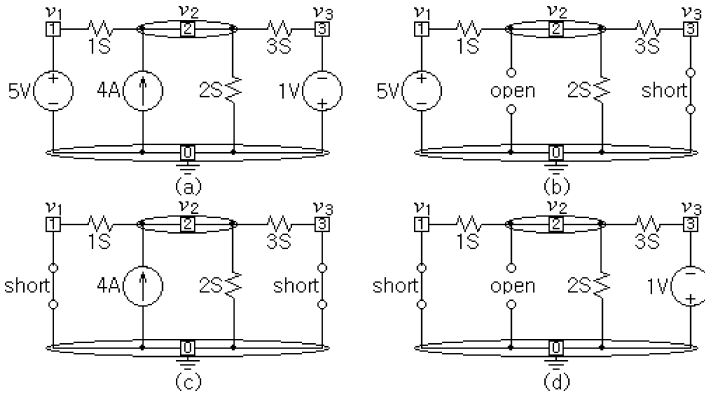


Figure 2.29 Circuits for Example 2.22 showing the superposition principle

$$\text{For the subnetwork in (d), } v_{2d} = (-1) \times \frac{1/3}{1/3 + 1/3} = -\frac{1}{2} \quad (\text{voltage divider}) \quad (\text{E2.22.3})$$

These individual outputs are added to obtain the overall output, i.e. the node voltage v_2 of the circuit excited by the three sources as

$$v_2 = v_{2b} + v_{2c} + v_{2d} = \frac{5}{6} + \frac{4}{6} - \frac{3}{6} = 1 \text{ V} \quad (\text{E2.22.4})$$

This result agrees with that obtained in Example 2.3.

2.9 OP Amp Circuits With Resistors

Several important properties of the OP Amp (operational amplifier) were introduced in Section 1.3.3. In this section the most basic OP Amp configurations are discussed, i.e. the inverting, noninverting, and buffer amplifiers with negative feedback, and, additionally, the inverting and noninverting amplifiers with positive feedback. Throughout this book, a black-box approach will be taken to analyze OP Amp circuits; in other words, only the terminal behavior will be discussed without paying much attention to the internal characteristic of the OP Amp.

2.9.1 Inverting OP Amp Circuit

Figure 2.30(a) shows an inverting OP Amp circuit, where the overall input v_i is applied to the negative (inverting) input terminal through a resistor R_1 and another resistor R_f makes a connection between the output terminal and the negative input terminal, providing a negative feedback path for the OP Amp. For a novice in OP Amp circuits, the OP Amp is replaced with the ideal OP Amp model of Figure 1.10(f) to yield a common circuit having a dependent voltage source, as depicted in Figure 2.30(b). For this single-mesh circuit, the mesh analysis method is chosen, the mesh current labeled i , and the controlling variable ($v_+ - v_-$) expressed in terms of the mesh current as

$$v_+ - v_- = 0 - (v_i - R_1 i) = -v_i + R_1 i \quad (\because v_+ = 0 : \text{grounded}) \quad (2.15)$$

where we have used the fact that the positive input terminal is grounded so that $v_+ = 0$. Then the mesh equation

$$(R_1 + R_f)i = v_i - A(v_+ - v_-) \stackrel{(2.15)}{=} v_i + A(v_i - R_1 i)$$

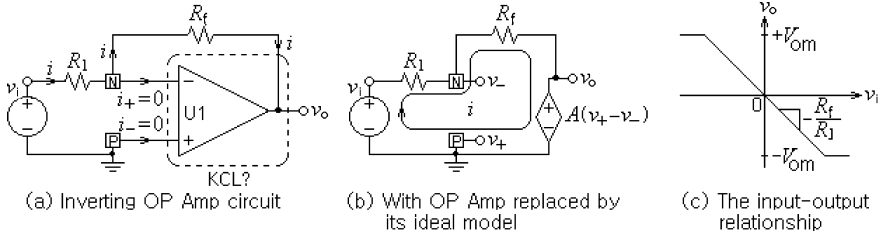


Figure 2.30 Inverting OP Amp circuit

is set up, which can be solved for the mesh current i as

$$i = \frac{A + 1}{A R_1 + (R_1 + R_f)} v_i \quad (2.16)$$

Finally, the output voltage v_o is found by subtracting the voltage drop across R_1 and R_f from the input voltage v_i as

$$v_o = v_i - (R_1 + R_f)i \stackrel{(2.16)}{=} \frac{A R_1 + (R_1 + R_f) - (A + 1)(R_1 + R_f)}{A R_1 + (R_1 + R_f)} v_i = \frac{-A R_f}{A R_1 + (R_1 + R_f)} v_i \quad (2.17)$$

$$v_o = -\frac{R_f/R_1}{1 + (1 + R_f/R_1)/A} v_i \rightarrow v_o \cong -\frac{R_f}{R_1} v_i \quad \text{under the assumption that } 1 + \frac{R_f}{R_1} \ll A \quad (2.18)$$

where it is reasonable to assume that $1 + R_f/R_1 \ll A$ since the open-loop gain A of an OP Amp is very large. In fact, the value of the open-loop gain A is not certain in that it varies with temperature and time, as well as from one sample to another. That is why it is good to see that A has disappeared in the (approximate) output voltage expression (2.18).

Now, as a more practical approach, we will use the virtual short principle (Remark 1.2(2)), which can be applied to an OP Amp with negative feedback between the output terminal and the negative input terminal, such as the one contained in the circuit of Figure 2.30(a). By the virtual short principle, we have

$$v_- \stackrel{(1.26)}{\cong} v_+ \stackrel{\text{grounded}}{=} 0 \quad (\text{virtual ground}) \quad (2.19)$$

where the negative input terminal is called a ‘virtual ground’ in the sense that its node voltage is zero, giving the illusion of being grounded. Thus the current through R_1 can be found to be

$$i = \frac{v_i - v_-}{R_1} \stackrel{(2.19)}{=} \frac{v_i}{R_1} \quad (2.20)$$

Noting that all this current flows through R_f to the output terminal since no current flows into or out of the negative input terminal by the virtual open principle (Remark 1.2(1)), the voltage drop across R_f is subtracted from the negative input terminal voltage v_- to get the output voltage v_o as

$$v_o = v_- - R_f i \stackrel{(2.19), (2.20)}{=} -\frac{R_f}{R_1} v_i \quad (2.21)$$

which agrees with Equation (2.18). This input–output relationship is described by the *transfer characteristic curve* in Figure 2.30(c), which implies that the *voltage gain*, i.e. the ratio of the output voltage to the input voltage of the circuit, is

$$\frac{v_o}{v_i} = -\frac{R_f}{R_1} \quad (2.22)$$

as long as v_o is not saturated into $\pm V_{om}$. This is referred to as the *closed-loop gain* due to the fact that it is obtained when the feedback path makes a closed loop consisting of the OP Amp and R_f .

Several points worth mentioning about the OP Amp circuit are

1. The closed-loop gain is determined by the external components (R_g and R_f) and can easily be customized to a particular application.
2. The OP Amp circuit is called an *inverting amplifier* because the closed-loop gain is negative and basically the input voltage is applied to the negative input terminal.
3. The input impedance is R_1 and the output impedance is $R_o = 0$, as shown in Figure 2.30(b) with the assumption of an ideal OP Amp.

2.9.2 Noninverting OP Amp Circuit

Figure 2.31(a) shows a noninverting OP Amp circuit, where the overall input v_i is applied to the positive (noninverting) input terminal and another resistor R_f makes a connection between the output terminal and the negative input terminal, providing a negative feedback path for the OP Amp. For a novice in OP Amp circuits, the OP Amp is replaced with the ideal OP Amp model of Figure 1.10(f) to get a common circuit having a dependent voltage source as depicted in Figure 2.31(b). For this single-mesh circuit, the mesh analysis method is chosen, the mesh current labeled i (in the counterclockwise direction), and the controlling variable ($v_+ - v_-$) expressed in terms of the mesh current as

$$v_+ - v_- = v_i - R_1 i \quad (2.23)$$

where we have used the fact that the positive input terminal is directly connected to the input voltage source v_i so that $v_+ = v_i$. Then the mesh equation is set up and solved as

$$\begin{aligned} (R_1 + R_f)i &= A(v_+ - v_-) \stackrel{(2.23)}{=} A(v_i - R_1 i) \\ i &= \frac{A}{A R_1 + (R_1 + R_f)} v_i \end{aligned} \quad (2.24)$$

Finally, the output voltage v_o is found by using Equation (1.22) together with Equation (2.23) or by summing the voltage drops across R_1 and R_f as

$$v_o = (R_1 + R_f)i \stackrel{(2.24)}{=} \frac{A(R_1 + R_f)}{A R_1 + (R_1 + R_f)} v_i = \frac{(R_1 + R_f)/R_1}{1 + (R_1 + R_f)/R_1/A} v_i \quad (2.25)$$

$$v_o = \frac{(R_1 + R_f)/R_1}{1 + (R_1 + R_f)/R_1/A} v_i \rightarrow v_o \cong \frac{R_1 + R_f}{R_1} v_i \quad \text{under the assumption that } 1 + \frac{R_f}{R_1} \ll A \quad (2.26)$$

where it is reasonable to assume that $1 + R_f/R_1 \ll A$, as mentioned above.

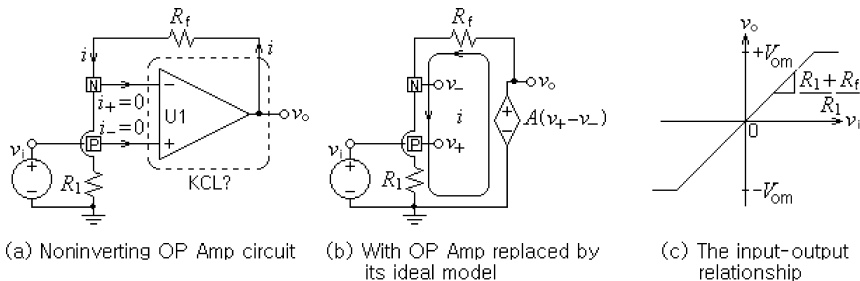


Figure 2.31 Noninverting OP Amp circuit

Now, let us take a more practical approach, which is to use the virtual short principle (Remark 1.2(2)). Noting that the OP Amp in the circuit of Figure 2.31(a) has a negative feedback between the output terminal and the negative input terminal and that the positive input terminal is directly connected to the input voltage source v_i so that $v_+ = v_i$, we have

$$v_- \stackrel{(1.26)}{\cong} v_+ = v_i$$

Thus the current through R_1 can be found to be

$$i = \frac{v_- - 0}{R_1} = \frac{v_i}{R_1} \quad (2.27)$$

Noting that all of this current flows through R_f from the output terminal since no current flows into or out of the negative input terminal by the virtual open principle (Remark 1.2(1)), the voltage drops across R_1 and R_f are summed to get the output voltage v_o as

$$v_o = (R_1 + R_f)i \stackrel{(2.27)}{=} \frac{R_1 + R_f}{R_1} v_i \quad (2.28)$$

which agrees with Equation (2.26). This input–output relationship is described by the *transfer characteristic curve* in Figure 2.31(c), which implies that the *voltage gain*, i.e. the ratio of the output voltage to the input voltage of the circuit is

$$\frac{v_o}{v_i} = \frac{R_1 + R_f}{R_1} \quad (2.29)$$

as long as v_o is not saturated into $\pm V_{om}$. This is referred to as the *closed-loop gain* due to the fact that it is obtained when the feedback path makes a closed loop consisting of the OP Amp and R_f .

Several points worth mentioning about the OP Amp circuit are

1. The closed-loop gain is determined by the external components (R_i and R_f) and can easily be customized to a particular application.
2. The OP Amp circuit is called a *noninverting amplifier* because the closed-loop gain is positive and basically the input voltage is applied to the positive input terminal.
3. The input impedance is infinitely large (∞) as the input impedance of an ideal OP Amp itself and the output impedance is $R_o = 0$, as shown in Figure 2.31(b), with the assumption of an ideal OP Amp.

Note. Whether an OP Amp circuit is inverting or noninverting depends on which of its negative/positive input terminals the input is applied to. It determines the sign of the voltage gain (2.22) and (2.29), and the negative/positive slope of the transfer characteristic curves depicted in Figures 2.30(c) and 2.31(c).

(Question) You may have tried to apply KCL to the OP Amp or the group of its two input nodes and output node or the closed surface denoted by the dotted line in Figure 2.30(a) or 2.31(a) to write the KCL equation as follows. Do they hold?

$$i_+ + i_- + i \stackrel{?}{=} 0, \quad i_+ + i_- - i \stackrel{?}{=} 0$$

(Answer) No, they do not hold since $i_+ = 0$, $i_- = 0$, and $i = v_i/R_1$. Nevertheless, it cannot be a counterexample contradicting KCL because KCL has just been misapplied. Referring to Figure 2.32, recall that the (dual) power supplies for OP Amps are omitted in the circuit diagram as mentioned in Section 1.3.3. If KCL is to be applied to the

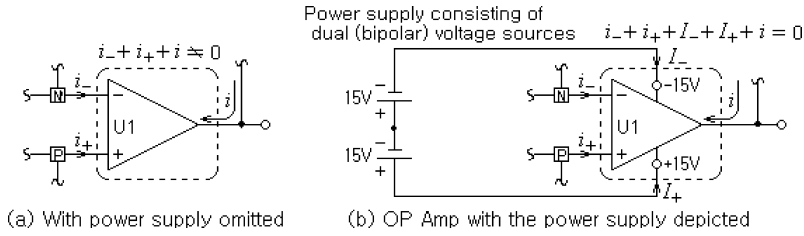


Figure 2.32 Application of KCL to the output node of OP Amp

OP Amp output node or any group of nodes including it, the omitted power supplies should be restored back (Figure 2.32(b)) or the OP Amp replaced with its model having a dependent voltage source (Figure 1.10(e) or (f)). In practice, this rarely needs to be done for analysis of OP Amp circuits.

[Remark 2.2] Practical Analysis Rules of OP Amp Circuits with Negative Feedback

1. Apply the ‘virtual open’ principle that $i_+ = 0$ and $i_- = 0$, which is mentioned as one of the ideal OP Amp conditions in Remark 1.2(1).
2. Apply the ‘virtual short’ principle that $v_+ \cong v_-$, which is mentioned in Remark 1.2(2). However, this rule may be preempted (invalidated) by positive feedback, if there is any. Besides, KCL must be applied to each node to write the node equations in unknown node voltages, except for the output node of an OP Amp or any group of nodes (closed surface) including it.

2.9.3 Voltage Follower

Figure 2.33(a) shows a noninverting OP Amp circuit with R_1 open-circuited and R_f short-circuited, whose voltage gain (2.29) will be unity.

$$\frac{v_o}{v_i} \stackrel{(2.29)}{=} \frac{R_1 + R_f}{R_1} \xrightarrow{R_1=\infty, R_f=0} \frac{v_o}{v_i} = 1; \quad v_o = v_i \quad (2.30)$$

This circuit is called a unity-gain noninverting amplifier or a *voltage follower* in the sense that the output voltage follows the input voltage. Note that its input impedance is infinitely large and its output impedance is zero as the impedance of a (dependent) voltage source.

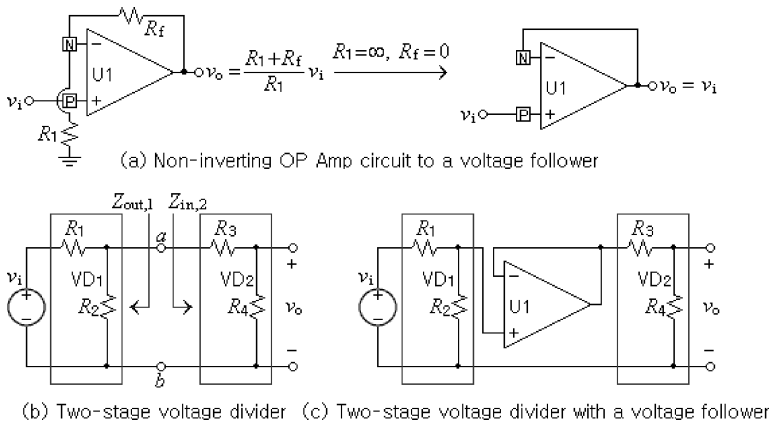


Figure 2.33 Voltage follower for removing or reducing the load effect

The circuit can be used as a voltage buffer for eliminating the interstage loading effect since it blockades the flow of current while presenting a virtual short connection between the input and output of the OP Amp in terms of the voltage. For example, let us compare the two circuits shown in Figures 2.33(b) and (c) in terms of their voltage gains:

$$\begin{aligned} \frac{v_o}{v_i} &= \frac{R_2 \parallel (R_3 + R_4)}{R_1 + [R_2 \parallel (R_3 + R_4)]} \frac{R_4}{R_3 + R_4} = \frac{R_2 (R_3 + R_4) \times R_4 / (R_3 + R_4)}{R_1 (R_2 + R_3 + R_4) + R_2 (R_3 + R_4)} \\ &= \frac{R_2}{R_1 + R_2} \frac{R_4}{R_1 R_2 / (R_1 + R_2) + (R_3 + R_4)} \end{aligned} \quad (2.31)$$

$$\frac{v_o}{v_i} = \frac{R_2}{R_1 + R_2} \frac{R_4}{R_3 + R_4} \quad (2.32)$$

This indicates that the voltage gain (Equation (2.31)) of the two-stage voltage divider with no voltage follower in Figure 2.33(b) is smaller than that (Equation (2.32)) of the two-stage voltage divider with a voltage follower between the two stages in Figure 2.33(c), which can be noticed from its larger denominator. What causes this difference in voltage gain? It arises from the *loading effect* of the second-stage voltage divider (VD2) on the first-stage one (VD1) in Figure 2.33(b), while in Figure 2.33(c) such a loading effect is eliminated by the voltage follower having infinitely large input impedance and zero output impedance.

The difference can be made small by making the following inequality satisfied by a wide margin:

$$Z_{\text{out},1} = \frac{R_1 R_2}{R_1 + R_2} \ll R_3 + R_4 = Z_{\text{in},2} \quad (2.33)$$

Note that the left-hand side (LHS) is the output impedance of the first voltage divider (at the source side) and the right-hand side (RHS) is the input impedance of the second one (at the load side) seen from the terminals *a-b*.

2.9.4 More Exact Analysis of OP Amp Circuits

For more exact analysis of the inverting OP Amp circuit of Figure 2.30(a), the OP Amp can be replaced with a practical model of Figure 1.10(e), as depicted in Figure 2.34(a), where the input impedance of the OP Amp is assumed to be infinitely large for simplicity. Applying KCL to the output node and the negative input node of the OP Amp yields the node equations as

$$\begin{aligned} \text{Node N: } \quad & \frac{v_o}{R_L} = \frac{v_i - v_o}{R_1 + R_f} + \frac{A(v_+ - v_-) - v_o}{R_o} \quad (v_+ = 0: \text{grounded}) \\ \text{Node O: } \quad & \frac{v_- - v_i}{R_1} + \frac{v_- - v_o}{R_f} = 0 \\ & \begin{bmatrix} \frac{1}{R_1 + R_f} + \frac{1}{R_o} + \frac{1}{R_L} & \frac{A}{R_o} \\ -\frac{1}{R_f} & \frac{1}{R_1} + \frac{1}{R_f} \end{bmatrix} \begin{bmatrix} v_o \\ v_- \end{bmatrix} = \begin{bmatrix} \frac{v_i}{R_1 + R_f} \\ \frac{v_i}{R_1} \end{bmatrix} \end{aligned} \quad (2.34)$$

This equation for v_o can be solved as

$$\begin{aligned} v_o &= \frac{\left(\frac{1}{R_1} + \frac{1}{R_f}\right) \frac{1}{R_1 + R_f} - \frac{A}{R_o} \frac{1}{R_1}}{\left(\frac{1}{R_1 + R_f} + \frac{1}{R_o} + \frac{1}{R_L}\right) \left(\frac{1}{R_1} + \frac{1}{R_f}\right) + \frac{A}{R_o} \frac{1}{R_f}} v_i \\ &= \frac{R_L (R_o - A R_f)}{R_o R_L + (R_o + R_L) (R_1 + R_f) + A R_1 R_L} v_i \xrightarrow{A \rightarrow \infty} -\frac{R_f}{R_1} v_i \end{aligned} \quad (2.35)$$

This naturally agrees with Equation (2.18) or (2.21).

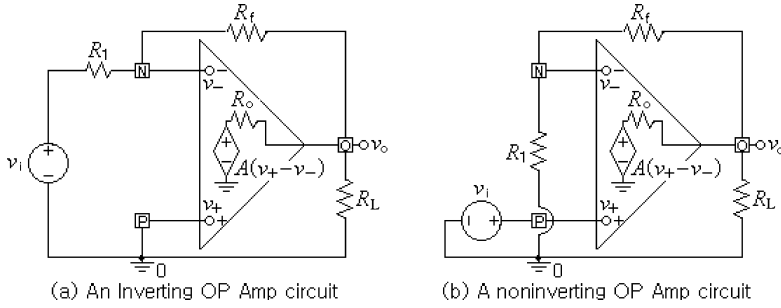


Figure 2.34 Inverting and noninverting OP Amp circuits with a practical model

For a more exact analysis of the noninverting OP Amp circuit of Figure 2.31(a), the OP Amp can be replaced with a practical model of Figure 1.10(e), as depicted in Figure 2.34(b), where the input impedance of the OP Amp is assumed to be infinitely large for simplicity. Applying KCL to the output node of the OP Amp yields the node equation as

$$\frac{v_o}{(R_1 + R_f) \parallel R_L} = \frac{A(v_+ - v_-) - v_o}{R_o} \quad (2.36)$$

This equation can be solved for v_o as

$$\begin{aligned} v_o &= \frac{(R_1 + R_f) \parallel R_L}{R_o + [(R_1 + R_f) \parallel R_L]} A(v_+ - v_-) \quad \text{with } v_+ = v_i \text{ and } v_- = \frac{R_1}{R_1 + R_f} v_o \\ &= \frac{(R_1 + R_f) R_L}{R_o (R_1 + R_f + R_L) + (R_1 + R_f) R_L} A \left(v_i - \frac{R_1}{R_1 + R_f} v_o \right) \\ \left[1 + \frac{AR_1 R_L}{R_o (R_1 + R_f + R_L) + (R_1 + R_f) R_L} \right] v_o &= \frac{A(R_1 + R_f) R_L}{R_o (R_1 + R_f + R_L) + (R_1 + R_f) R_L} v_i \quad (2.37) \\ v_o &= \frac{A(R_1 + R_f) R_L}{AR_1 R_L + R_o (R_1 + R_f + R_L) + (R_1 + R_f) R_L} v_i \xrightarrow{A \rightarrow \infty} \frac{R_1 + R_f}{R_1} v_i \end{aligned}$$

As expected, this result agrees with Equation (2.26) or (2.28).

Equations (2.35) and (2.37) allow us to examine the effects of the load impedance R_L and the output impedance R_o as well as the open-loop gain A on the closed-loop gains of the basic inverting and noninverting OP Amp circuits, respectively.

2.9.5 OP Amp Circuits with Positive Feedback

As discussed in Section 1.3.3, a negative feedback path between the output terminal and the negative input terminal of an OP Amp has a *stabilization* effect on the differential input voltage ($v_+ - v_-$) and the output voltage v_o so that $v_+ - v_- = v_o/A \cong 0$. Thus v_o may stay at some nominal value V_o between the negative/positive maximum output or saturation (limit) voltages $-V_{om}$ and $+V_{om}$. In contrast, a *positive feedback* path between the output terminal and the positive input terminal of an OP Amp has a *destabilization* effect on the differential input voltage ($v_+ - v_-$) and the output voltage v_o . As a result, v_o may diverge till it reaches $-V_{om}$ or $+V_{om}$ to be confined there.

2.9.5.1 Inverting Positive Feedback OP Amp Circuit

Figure 2.35(a) shows an inverting OP Amp circuit with a positive feedback path (via R_2) between the output terminal and the positive input terminal. Due to the destabilization effect of the positive feedback,

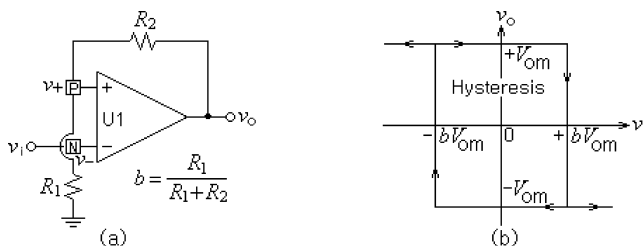


Figure 2.35 Inverting positive feedback OP Amp circuit and its input–output relationship (transfer characteristic)

the virtual short principle does not apply to this circuit and besides, the ideal or practical OP Amp model does not work for this circuit. Noting that

- (a) its output voltage will be either $v_o = +V_{om}$ or $-V_{om}$, depending on which one of the two input terminals is of higher voltage, and
- (b) the voltage of the positive input terminal is determined by the voltage divider rule as $bv_o = \pm bV_{om}$ with $b = R_1/(R_1 + R_2)$,

suppose that the output voltage at some instant is $v_o = +V_{om}$. Then the $+$ input terminal of the OP Amp has the node voltage of

$$v_+ = +b V_{om} \quad \text{with } b = \frac{R_1}{R_1 + R_2} \quad (2.38)$$

and this state will be maintained as long as $v_i = v_- < v_+ = +b V_{om}$. If the input voltage v_i somehow rises above $v_+ = +b V_{om}$, the voltages at the output and $+$ input terminals will go down to

$$v_o = -V_{om} \quad \text{and} \quad v_+ = -b V_{om}$$

respectively, and this state will be maintained as long as $v_i = v_- > v_+ = -b V_{om}$. If the input voltage v_i somehow goes below $v_+ = -b V_{om}$, the voltages at the output and $+$ input terminals will go up to

$$v_o = +V_{om} \quad \text{and} \quad v_+ = +b V_{om}$$

respectively, and this state will be maintained as long as $v_i = v_- < v_+ = +b V_{om}$. The output becomes $-V_{om}$ for an input above the upper (higher) threshold value $V_{TH} = +b V_{om}$ and $+V_{om}$ for an input below the lower threshold value $V_{TL} = -b V_{om}$, while it stays at the current state for an input between V_{TL} and V_{TH} . This input–output relationship can be described by the following equation and the transfer characteristic curve in Figure 2.35(b):

$$v_o = \begin{cases} +V_{om} & \text{for } v_i < -b V_{om} \\ \text{keep the current state} & \text{for } -b V_{om} \leq v_i \leq +b V_{om} \\ -V_{om} & \text{for } v_i > +b V_{om} \end{cases} \quad \text{with } b = \frac{R_1}{R_1 + R_2} \quad (2.39)$$

2.9.5.2 Noninverting Positive Feedback OP Amp Circuit

Figure 2.36(a) shows a noninverting OP Amp circuit with a positive feedback path (via R_2) between the output terminal and the positive input terminal. Note the following:

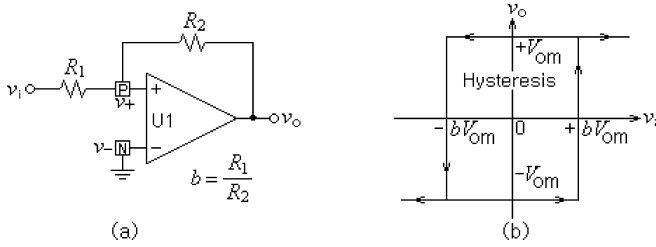


Figure 2.36 Noninverting positive feedback OP Amp circuit and its input–output relationship (transfer characteristic)

1. Its output voltage will be either $v_o = +V_{om}$ or $-V_{om}$ depending on whether the voltage at the positive input terminal is higher or lower than zero, i.e. the voltage at the negative input terminal that is grounded.
2. The value of the input voltage causing the voltage, v_+ , at the positive input terminal to be zero is related with the output voltage v_o as

$$v_i = -\frac{R_1}{R_2} v_o = -b v_o \quad \text{with } b = \frac{R_1}{R_2} \quad (2.40)$$

This implies that once $v_o = +V_{om}$, it changes into $-V_{om}$ only when $v_i < -bV_{om}$ and once $v_o = -V_{om}$, it changes into $+V_{om}$ only when $v_i > +bV_{om}$. Thus the input–output relationship can be written that is described by the following equation and the transfer characteristic curve in Figure 2.36(b):

$$v_o = \begin{cases} +V_{om} & \text{for } v_i > +bV_{om} \\ \text{keep the current state} & \text{for } -bV_{om} \leq v_i \leq +bV_{om} \\ -V_{om} & \text{for } v_i < -bV_{om} \end{cases} \quad \text{with } b = \frac{R_1}{R_2} \quad (2.41)$$

[Remark 2.3] Destabilization Effect of Positive Feedback and a Bistable Multivibrator

1. If an OP Amp has a positive feedback and no negative feedback, changes in the output voltage v_o and the differential input ($v_+ - v_-$) help each other synergistically so that even the slightest change in v_o instantly results in $v_o = +V_{om}$ or $-V_{om}$. That is why the positive feedback OP Amp circuits operate in saturation with their output voltages at one of the two (extreme) states $\pm V_{om}$ most of the time. This is called the ‘destabilization effect’ of positive feedback.
2. The positive feedback OP Amp circuits in Figures 2.35(a) and 2.36(a) are called the ‘bistable multivibrator’ since they have two stable outputs $\pm V_{om}$ (positive or negative saturation voltages) for an input voltage v_i as long as $-bV_{om} \leq v_i \leq +bV_{om}$. Just like a flip-flop, they are said to have a memory since they keep the previous state, i.e. their outputs are either $+V_{om}$ or $-V_{om}$ depending on the (previous) state that they were in, unless they are triggered by any (possibly of short duration) input of magnitude greater than bV_{om} . This presents the reason why the circuits are referred to as inverting/noninverting *Schmitt triggers*.
3. As can be seen from the overall input–output relationships that are described by the transfer characteristic curves in Figures 2.35(b) and 2.36(b), the positive feedback OP Amp circuits have two different output-changing paths since there are two different threshold values of input to change the output depending on whether their inputs are increasing or decreasing. For this reason, they are said to exhibit a *hysteresis* or *deadband* characteristic.
4. The hysteresis or deadband characteristic can be used to reduce the number of contact bounces in an on–off switch for a microprocessor or a temperature control system (see Example 2.32). It also has an important application to periodic wave generation, as will be discussed in Section 3.5.2.

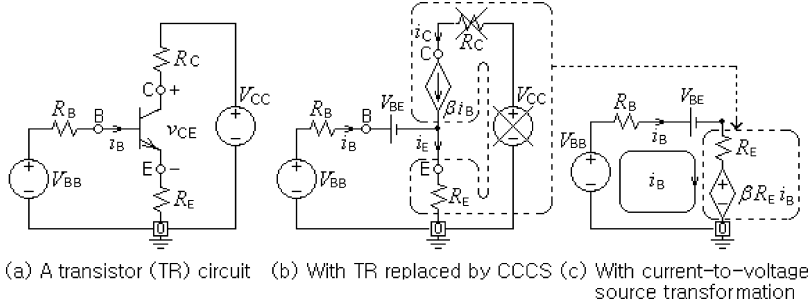


Figure 2.37 A transistor circuit and its equivalents

2.10 Transistor Circuits

Consider the transistor circuit shown in Figure 2.37(a). To analyze the circuit, the NPN-BJT (bipolar junction transistor) can be replaced with the CCCS (current-controlled current source) model of Figure 1.11.1(b), as depicted in Figure 2.37(b). Referring to Remark 1.3, V_{CC} and R_C are removed (short-circuited) without making any difference to the analysis of the rest of the circuit because they are connected in series with a current source, and then the βi_B current source in parallel with R_E is transformed into a $\beta R_E i_B$ voltage source in series with R_E , as depicted in Figure 2.37(c). Noting that the mesh current of this single-mesh circuit is simply the base current i_B , the mesh equation can be written and solved as

$$(R_B + R_E) i_B = V_{BB} - V_{BE} - \beta R_E i_B; \quad [R_B + (\beta + 1)R_E] i_B = V_{BB} - V_{BE} \quad (2.42)$$

$$i_B = \frac{V_{BB} - V_{BE}}{R_B + (\beta + 1)R_E} \quad (2.43)$$

If the voltage v_{CE} across the terminals C and E is needed, we should go back to the circuit in Figure 2.37(b) with V_{CC} and R_C undeleted and subtract the voltage drops across R_C and R_E from V_{CC} to get

$$v_{CE} = V_{CC} - R_C i_C - R_E i_E \quad i_C = \beta i_B, i_E = i_B + i_C = (\beta + 1) i_B \quad V_{CC} - \beta R_C i_B - (\beta + 1) R_E i_B \quad (2.44)$$

2.11 Loading Effect and Input/Output Resistance

To re-examine the relationship among the output impedance of a source, the input impedance of a load, and the loading effect discussed in Sections 2.2.1 and 2.9.3, consider the typical model of a voltage amplifier shown in Figure 2.38(a), where the voltage amplifier having the input impedance R_I , the output impedance R_o , and the open-circuit voltage gain A_{oc} is connected to a voltage source v_s in series with R_s at its input port and to a load R_L at its output port. The voltage divider rule can be applied at the input and output ports to obtain the *overall voltage gain*, i.e. the ratio of the output load voltage to the input source voltage as

$$\begin{aligned} v_L &= A_{oc} v_1 \frac{R_L}{R_o + R_L} = A_{oc} \frac{R_I}{R_s + R_I} v_s \frac{R_L}{R_o + R_L} \\ \frac{v_L}{v_s} &= \frac{R_I}{R_s + R_I} A_{oc} \frac{R_L}{R_o + R_L} = \frac{1}{1 + (R_s/R_I)} A_{oc} \frac{1}{1 + (R_o/R_L)} \end{aligned} \quad (2.45)$$

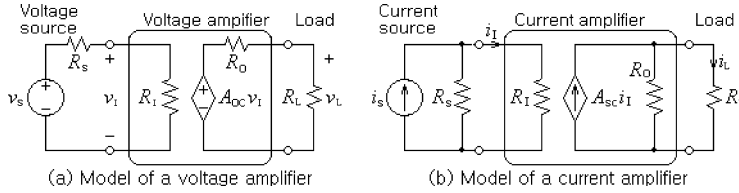


Figure 2.38 Models of voltage/current amplifiers

This implies that the loading effect of decreasing this overall voltage gain can be made small by making the following inequalities satisfied by a wide margin:

$$R_s \ll R_1 \quad \text{and} \quad R_o \ll R_L \quad (2.46)$$

Likewise, for the typical model of a current amplifier with the short-circuit current gain A_{sc} shown in Figure 2.38(b), the current divider rule can be applied at the input and output ports to obtain the *overall current gain* as

$$\begin{aligned} i_L &= A_{sc} i_1 \frac{R_o}{R_o + R_L} = A_{sc} \frac{R_s}{R_s + R_1} i_s \frac{R_o}{R_o + R_L} \\ \frac{i_L}{i_s} &= \frac{R_s}{R_s + R_1} A_{sc} \frac{R_o}{R_o + R_L} = \frac{1}{1 + (R_1/R_s)} A_{sc} \frac{1}{1 + (R_L/R_o)} \end{aligned} \quad (2.47)$$

This implies that the loading effect of decreasing this overall current gain can be made small by making the following inequalities satisfied by a wide margin:

$$R_s \gg R_1 \quad \text{and} \quad R_o \gg R_L \quad (2.48)$$

These inequalities (2.46) and (2.48) mean that it is helpful in reducing the *loading effect* in terms of the voltage/current gain to make the *output impedance* of the source side much smaller/larger than the *input impedance* of the load side at each stage.

[Remark 2.4] Loading Effect and Input/Output Resistances (Impedances)

1. For the purpose of transferring the voltage from the source side to the load side, it helps reduce the loading effect to make the output impedance of the source side much smaller than the input impedance of the load side. In contrast, for the purpose of transferring the current from the source side to the load side, it helps reduce the loading effect to make the output impedance of the source side much larger than the input impedance of the load.
2. To reduce loading effect is desirable not only because it reduces the variation of the voltage or current gain caused by the change of the source and load, but also because it allows a multistage amplifier to be analyzed and designed stage by stage.

2.12 Load Line Analysis of Nonlinear Resistor Circuits

The v - i characteristic of a nonlinear resistor such as a diode or a transistor is often described by a curve on the i - v plane rather than by a mathematical relation. The v - i characteristic curve can be obtained by

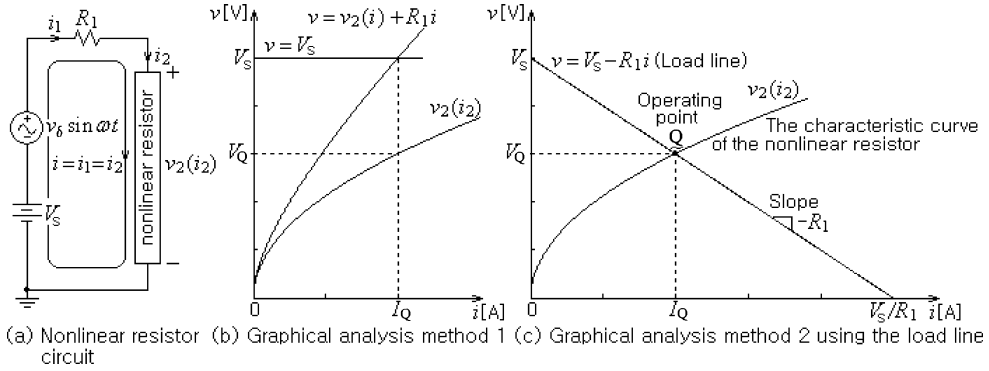


Figure 2.39 Graphic analysis of a nonlinear resistor circuit

using a curve tracer. To analyze circuits containing a nonlinear resistor, the *load line analysis* should be used. To grasp the concept of a load line, consider the graphical analysis of the circuit in Figure 2.39(a), which consists of a linear resistor R_1 , a nonlinear resistor R_2 , a DC voltage source V_s , and an AC voltage source of small amplitude v_δ . Let the v - i relationship of R_2 be denoted by $v_2(i)$ and represented by the characteristic curve in Figure 2.39(b). A graphical method will be considered that yields the *operating point* (I_Q , V_Q), i.e. the pair of the current through and the voltage across R_2 for $v_\delta = 0$. KVL can be applied around the mesh to write down the mesh equation as

$$R_1 i + v_2(i) = V_s \quad (2.49)$$

Since no specific mathematical expression of $v_2(i)$ is given, no analytical method can be used to solve this equation, which is why a graphical method will be used. First, it may be considered to try plotting the graph for the LHS (left-hand side) of Equation (2.49) and finding its intersection with a horizontal line for the RHS (right-hand side), i.e. $v = V_s$, as depicted in Figure 2.39(b). Another way is to leave only the nonlinear term on the LHS and move the other terms into the RHS to rewrite the equation as

$$v_2(i) = V_s - R_1 i \quad (2.50)$$

and find the intersection, called the *operating point* and denoted by Q (quiescent point), of the graphs for both sides, as depicted in Figure 2.39(c). The straight line with the slope of $-R_1$ is called the *load line*. This graphical method is better than the first one because it does not require a new curve to be plotted for $v_2(i) + R_1 i$. That is why it is widely used to analyze nonlinear resistor circuits in the name of ‘load line analysis’. Let us take a look at the following example.

Note. All resistors appearing in this book except in this section are linear in the sense that their voltages are linearly proportional to their currents so that their voltage–current relationships are described by Ohm’s law (Equation (1.6a)) and, consequently, their v - i characteristics are described by straight lines passing through the origin with the slopes corresponding to their resistances on the i - v plane. However, they may have been modeled or approximated to be linear just for simplicity and convenience, because all physical resistors more or less exhibit some nonlinear characteristics. The problem is whether or not the modeling is valid in the range of practical operations so that it may yield the solution with sufficient accuracy to serve the objective of the analysis and design.

Note. A *curve tracer* is an instrument that displays the v - i characteristic curve of an electric element on a cathode ray tube (CRT) when the element is inserted into an appropriate receptacle (Reference [F-1]).

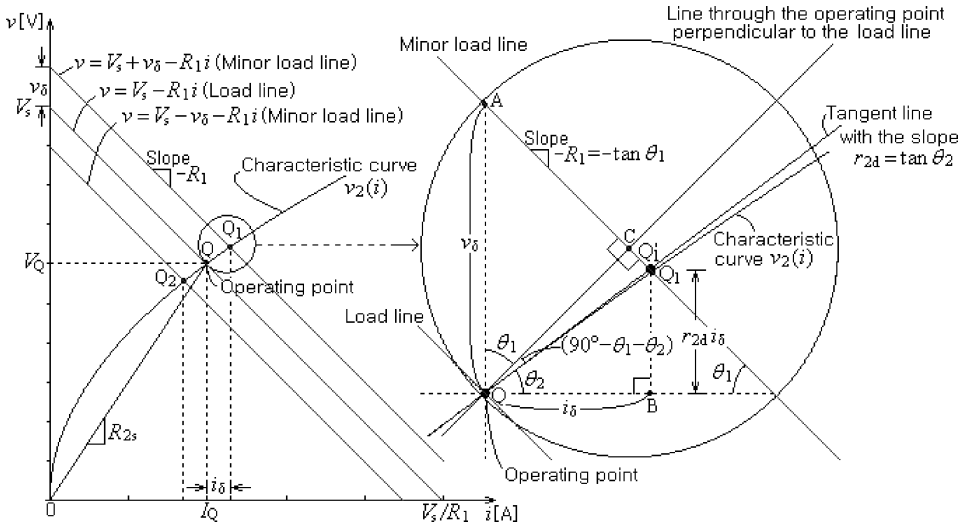


Figure 2.40 Variation of the voltage and current of a nonlinear resistor around the operating point

(Example 2.23) Small-Signal (AC) Analysis of Nonlinear Circuit

Consider the circuit in Figure 2.39(a), where a linear resistor R_1 and a nonlinear resistor R_2 in series are driven by a DC voltage source V_s in series with a small-amplitude AC voltage source producing the virtual voltage v_δ

$$v_s(t) = V_s + v_\delta \sin \omega t \quad (\text{E2.23.1})$$

The voltage–current relationship, $v_2(i_2)$, of the nonlinear resistor R_2 is described by the characteristic curve in Figure 2.40.

As depicted in Figure 2.40, the upper/lower limits as well as the equilibrium value of the current i through the circuit can be obtained from the three operating points, i.e. the intersections (Q_1 , Q , and Q_2) of the characteristic curve with the following three load lines:

$$v = V_s + v_\delta - R_1 i \quad (\text{E2.23.2a})$$

$$v = V_s - R_1 i \quad (\text{E2.23.2b})$$

$$v = V_s - v_\delta - R_1 i \quad (\text{E2.23.2c})$$

Although this approach gives the exact solution, no insight into the solution is gained from it. Instead, a rather approximate approach is taken, which consists of the following two steps:

1. Find the equilibrium (I_Q , V_Q) at the major operating point Q , which is the intersection of the characteristic curve with the DC load line (E2.23.2b).
2. Find the two approximate minor operating points Q'_1 and Q'_2 from the intersections of the tangent to the characteristic curve at Q with the two minor load lines (E2.23.2a) and (E2.23.2c).

Then the current will be obtained as

$$i(t) = I_Q + i_\delta \sin \omega t \quad (\text{E2.23.3})$$

With the *dynamic* or *small-signal* or *AC resistance* r_{2d} defined to be the slope of the tangent to the characteristic curve at Q as

$$r_{2d} = \left. \frac{dv_2}{di} \right|_Q \quad (\text{E2.23.4})$$

Now find the analytical expressions of I_Q and i_δ in terms of V_s and v_δ . Referring to the encircled area around the operating point in Figure 2.40, i_δ can be expressed in terms of v_δ as

$$\begin{aligned} i_\delta &= \overline{QB} \stackrel{\Delta QQ'_1 B}{=} \overline{QQ'_1} \cos \theta_2 \stackrel{\Delta QQ'_1 C}{=} \frac{\overline{QC} \cos \theta_2}{\cos(90^\circ - \theta_1 - \theta_2)} \stackrel{\Delta AQC}{=} \frac{\overline{AQ} \cos \theta_1 \cos \theta_2}{\sin(\theta_1 + \theta_2)} \\ &\stackrel{(\text{E.5})}{=} v_\delta \frac{\cos \theta_1 \cos \theta_2}{\sin \theta_1 \cos \theta_2 + \cos \theta_1 \sin \theta_2} = v_\delta \frac{1}{\tan \theta_1 + \tan \theta_2} \end{aligned} \quad (\text{E2.23.5})$$

This corresponds to approximating the characteristic curve in the operation range by its tangent at the operating point. Noting that:

- (a) the load line and the tangent to the characteristic curve at Q are at angles of $(180^\circ - \theta_1)$ and θ_2 to the positive i axis,
- (b) the slope of the load line is $\tan(180^\circ - \theta_1) = -\tan \theta_1$ and must be $-R_1$, which is the proportionality coefficient in i of the load line equation (E2.23.2); $\tan \theta_1 = R_1$, and
- (c) the slope of the tangent to the characteristic curve at Q is the dynamic resistance r_{2d} defined by (E2.23.4); $\tan \theta_2 = r_{2d}$,

Equation (E2.23.5) can be written as

$$i_\delta = \frac{v_\delta}{R_1 + r_{2d}} \quad (\text{E2.23.6})$$

Now the *static* or *DC resistance* of the nonlinear resistor R_2 is defined to be the ratio of the voltage V_Q to the current I_Q at the operating point Q as

$$R_{2s} = \frac{V_Q}{I_Q} = \frac{V_s - R_1 I_Q}{I_Q} \quad (\text{E2.23.7})$$

so that the DC component of the current, I_Q , can be written as

$$I_Q = \frac{V_s}{R_1 + R_{2s}} \quad (\text{E2.23.8})$$

Finally, the above results are combined to write the current through and the voltage across the nonlinear resistor R_2 as follows:

$$i(t) = I_Q + i_\delta \sin \omega t = \frac{V_s}{R_1 + R_{2s}} + \frac{v_\delta}{R_1 + r_{2d}} \sin \omega t \quad (\text{2.51})$$

$$v(t) = R_{2s} I_Q + r_{2d} i_\delta \sin \omega t = \frac{R_{2s}}{R_1 + R_{2s}} V_s + \frac{r_{2d}}{R_1 + r_{2d}} v_\delta \sin \omega t \quad (\text{2.52})$$

This result implies that the nonlinear resistor exhibits twofold resistance, i.e. the *static resistance* R_{2s} to a DC input and the *dynamic resistance* r_{2d} to an AC input of small amplitude. That is why r_{2d} is also called the (small-signal) AC resistance, while R_{2s} is called the DC resistance.

[Remark 2.5] Operating Point and Static/Dynamic Resistances of a Nonlinear Resistor

1. For a nonlinear resistor R_2 connected with linear resistors in a circuit excited by a DC source and a small-amplitude AC source, its operating point $Q = (I_Q, V_Q)$ is the intersection of its characteristic curve $v(i)$ and the load line.

2. The v intercept of the load line ($v = V_s - R_1 i$) is determined by the DC component (V_s) of the voltage source. The slope of the load line is determined by the equivalent resistance (R_1) of the linear part seen from the pair of terminals of the nonlinear resistor. (See Problem 2.29.)
3. The static or DC resistance (R_{2s}) is the ratio of the voltage V_Q to the current I_Q at the operating point Q.
4. The dynamic or small-signal or AC resistance (r_{2d}) is the slope of the tangent to the characteristic curve at Q.
5. Once R_1 , R_{2s} , and r_{2d} are obtained, the above formulas (2.51) and (2.52) can be used to find the voltage and current of the nonlinear resistor.

Note. The static or dynamic resistance are not used for linear resistors since they are identical.

Note. The relationship between the AC (small-signal) components of voltage across and current through the nonlinear resistor can be attributed to the Taylor series expansion of its VCR (voltage–current relationship) $v_2(i)$ up to the first-order term around the operating point $Q = (I_Q, V_Q)$:

$$v(i) \simeq V_Q + \left. \frac{dv_2}{di} \right|_Q (i - I_Q); \quad v_\delta \simeq V_Q + r_d i_\delta \quad \text{with} \quad r_d = \left. \frac{dv_2}{di} \right|_Q$$

2.13 More Examples of Resistor Circuits

The following examples, which have a slight emphasis on design and application, illustrate the usefulness of understanding circuits and highlights the importance of studying circuit theory. While the solution of an analysis problem is normally unique, that of a design problem is not in general. There may be various circuit configurations meeting given specifications. In most cases, the design solution must satisfy not only technical specifications on voltage/current/power but also various explicit and implicit constraints on the production cost, safety, etc.

(Example 2.24) Design and Evaluation of an Interface Network

Design an interface in a series configuration and another in a parallel configuration, as depicted in Figures 2.41(a) and (b), such that the load of $R_L = 90\,\Omega$ can operate with its rated voltage of 9 V.

(a) Series Interface

In the circuit of Figure 2.41(a), the load current $i_{R_L} = 9\text{ V}/90\,\Omega = 0.1\text{ A}$ flows through the interface resistor R_{S_x} as well as the voltage source and its resistance R_s . We can apply KVL around the mesh to write the mesh equation and solve it for R_{S_x} as

$$(R_s + R_{S_x}) i_{R_L} + v_{R_L} = 12; \quad (15 + R_{S_x}) 0.1 + 9 = 12; \quad R_{S_x} = 1.5/0.1 = 15\,\Omega \quad (\text{E2.24.1})$$

(b) Parallel Interface

In the circuit of Figure 2.41(b), the load current $i_{R_L} = 9\text{ V}/90\,\Omega = 0.1\text{ A}$ adds to the current, i_{R_p} , through the interface resistor R_{P_x} to make the current, i_{R_s} , through R_s . We can apply KCL to the top node of R_{P_x} to write the node equation and solve it for R_{P_x} as

$$\frac{9}{R_{P_x}} + \frac{9}{R_{R_L}} - \frac{12 - 9}{R_s} = 0; \quad \frac{9}{R_{P_x}} = -\frac{9}{90} + \frac{12 - 9}{15} = 0.1; \quad R_{P_x} = 90\,\Omega \quad (\text{E2.24.2})$$

(c) Comparison of Two Interfaces in Terms of Power and Voltage Variation w.r.t. the Load

Powers dissipated in the interface circuits are found to be

$$P_{S_x} = R_{S_x} i_{R_L}^2 = 15 \times 0.1^2 = 0.15\text{ W} \quad \text{and} \quad P_{P_x} = v_{R_L}^2 / R_{P_x} = 9^2 / 90 = 0.9\text{ W} \quad (\text{E2.24.3})$$

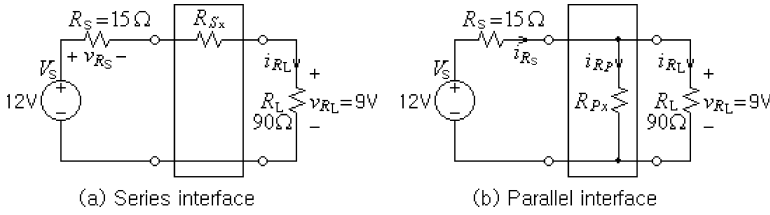


Figure 2.41 Interface design

We can write the output voltages in terms of a variable load resistance R_L and find their variations as

$$v_{o,S} = \frac{R_L}{R_S + R_S + R_L} V_S = \frac{R_L}{30 + R_L} 12 \quad (\text{E2.24.4})$$

$$\left. \frac{d}{dR_L} v_{o,S} \right|_{R_L=90} = 12 \frac{30}{(30 + R_L)^2} \Big|_{R_L=90} = 0.025 = 2.5\% \quad (\text{E2.24.5})$$

$$v_{o,P} = \frac{R_P \parallel R_L}{R_S + (R_P \parallel R_L)} V_S = \frac{90R_L/(90 + R_L)}{15 + 90R_L/(90 + R_L)} 12 = \frac{90R_L}{1350 + 105R_L} 12 \quad (\text{E2.24.6})$$

$$\left. \frac{d}{dR_L} v_{o,P} \right|_{R_L=90} = 90 \times 12 \frac{1350}{(1350 + 105R_L)^2} \Big|_{R_L=90} = 0.012 = 1.2\% \quad (\text{E2.24.7})$$

This result indicates that the series interface is advantageous in the aspect of power dissipation, while the parallel interface is advantageous in the aspect of voltage variation.

(Example 2.25) Design of a Ladder Network

As a design procedure of the ladder network of Figure 2.42, determine the values of the resistors R_1 , R_2 , R_3 , R_4 , and R_5 such that the voltages at nodes 2,3,4,5, and 6 are 9, 6, 3, 2, and 1 V, respectively.

Starting from the part farthest from the source, we proceed back toward the source as follows:

$$i_{R_5} = i_{56} = (v_5 - v_6)/R = (2 - 1) \text{ V} / 1 \text{ k}\Omega = 1 \text{ mA}; R_5 = v_6 / i_{R_5} = 1 \text{ V} / 1 \text{ mA} = 1 \text{ k}\Omega$$

$$i_{R_4} = i_{45} - i_{56} = (v_4 - v_5)/R - i_{56} = (3 - 2) \text{ V} / 1 \text{ k}\Omega - 1 \text{ mA} = 0; R_4 = v_5 / i_{R_4} = 2/0 = \infty \text{ (open)}$$

$$i_{R_3} = i_{34} - i_{45} = (v_3 - v_4)/R - i_{45} = (6 - 3) \text{ V} / 1 \text{ k}\Omega - 1 \text{ mA} = 2 \text{ mA}; R_3 = v_3 / i_{R_3} = 6 \text{ V} / 2 \text{ mA} = 3 \text{ k}\Omega$$

$$i_{R_2} = i_{23} - i_{34} = (v_2 - v_3)/R - i_{34} = (9 - 6) \text{ V} / 1 \text{ k}\Omega - 3 \text{ mA} = 0; R_2 = v_3 / i_{R_2} = 6/0 = \infty \text{ (open)}$$

$$i_{R_1} = i_{12} - i_{23} = (v_1 - v_2)/R - i_{23} = (12 - 9) \text{ V} / 1 \text{ k}\Omega - 3 \text{ mA} = 0; R_1 = v_2 / i_{R_1} = 9/0 = \infty \text{ (open)}$$

This indicates that the values of the resistors must be

$$R_1 = \infty \text{ (open)}, \quad R_2 = \infty \text{ (open)}, \quad R_3 = 3 \text{ k}\Omega, \quad R_4 = \infty \text{ (open)}, \quad R_5 = 1 \text{ k}\Omega$$

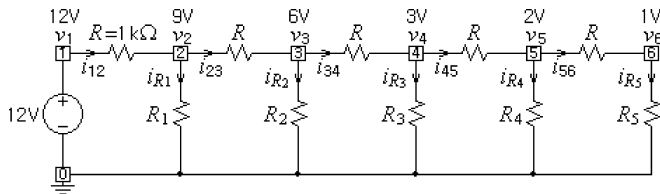


Figure 2.42 Resistive ladder network

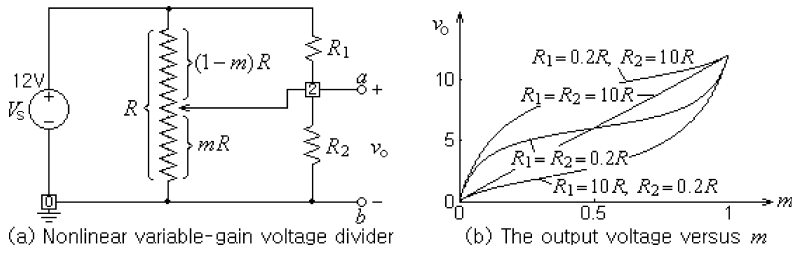


Figure 2.43 The circuit for Example 2.26

(Example 2.26) Nonlinear Variable-Gain Voltage Divider

Consider the circuit of Figure 2.43(a). Find the output voltage v_o in terms of m , R , R_1 , and R_2 , and plot v_o versus $m = 0:0.001:1$ for the following cases: (a) $R_1 = R_2 = 10R$; (b) $R_1 = R_2 = 0.2R$; (c) $R_1 = 10R$; $R_2 = 0.2R$; (d) $R_1 = 0.2R$, $R_2 = 10R$.

First, the output voltage can be written as

$$v_o = \frac{12 \frac{m R R_2}{m R + R_2}}{\frac{(1-m) R R_1}{(1-m) R + R_1} + \frac{m R R_2}{m R + R_2}} = \frac{12 m R_2 [(1-m) R + R_1]}{m(1-m) R (R_1 + R_2) + R_1 R_2} \quad (\text{E2.26.1})$$

Then the following MATLAB program `cir02e26.m` is composed and run to get Figure 2.43(b).

```
%cir02e26.m
m=0:0.001:1; R=1;
for i=1:4
    if i==1, R1=10*R; R2=10*R;
    elseif i==2, R1=0.2*R; R2=0.2*R;
    elseif i==3, R1=10*R; R2=0.2*R;
    else i==4, R1=0.2*R; R2=10*R;
    end
    vo = 12*m*R2.*((1-m)*R+R1)./(m.*(1-m)*R*(R1+R2)+R1*R2); % Eq. (E2.26.1)
    plot(m,vo), hold on, pause
end
```

(Example 2.27) Design of Dependent Source Parameter for Desired Resistance

Find the output resistance of the circuit of Figure 2.44(a) seen from terminals a - b two times, once by applying a test voltage source V_T and once by applying a test current source I_T . Then determine the values of K such that the output resistance is zero and infinity. Note that the independent voltage source V_s can be removed to find the equivalent resistance.

(a) Applying a Test Voltage Source V_T

Referring to Figure 2.44(a), the (right) 2Ω resistor is removed by open-circuiting it without making any difference in the analysis of the rest of the circuit since it is in parallel with a (test) voltage source. Then, noting that the controlling variable i_b can be expressed in terms of the mesh currents as $i_b = i_2 - i_1$, the mesh equation is set up and solved as

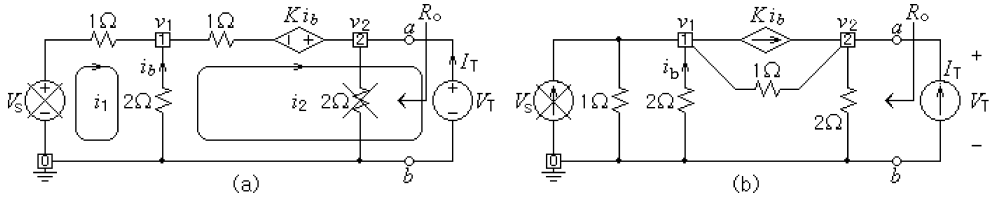


Figure 2.44 The circuit for Example 2.44

$$\begin{bmatrix} 1+2 & -2 \\ -2 & 1+2 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} 0 \\ K i_b - V_T \end{bmatrix} = \begin{bmatrix} 0 \\ K(i_2 - i_1) - V_T \end{bmatrix}; \quad \begin{bmatrix} 3 & -2 \\ K-2 & 3-K \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} 0 \\ -V_T \end{bmatrix} \quad (\text{E2.27.1})$$

$$\begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \frac{1}{5-K} \begin{bmatrix} 3-K & 2 \\ 2-K & 3 \end{bmatrix} \begin{bmatrix} 0 \\ -V_T \end{bmatrix} = \frac{V_T}{K-5} \begin{bmatrix} 2 \\ 3 \end{bmatrix} \quad (\text{E2.27.2})$$

This result can be used to get the expression of the test voltage in terms of the test current as

$$I_T = -i_2 = \frac{3}{5-K} V_T; \quad V_T = \frac{5-K}{3} I_T \quad (\text{E2.27.3})$$

Matching this relation with Equation (2.14) yields

$$Z_{Th} = (5-K)/3 \quad (\text{E2.27.4})$$

Finally, we take the parallel combination of Z_{Th} with the omitted resistance 2Ω to get the output resistance as

$$R_o = Z_{Th} \parallel 2 \stackrel{(\text{E2.27.4})}{=} \frac{(5-K)/3 \times 2}{(5-K)/3 + 2} = \frac{2(5-K)}{11-K} \quad (\text{E2.27.5})$$

(b) Applying a Test Current Source I_T

Referring to Figure 2.44(b), a test current source is applied and the voltage source(s) transformed into the equivalent current sources. Then, noting that the controlling variable i_b can be expressed in terms of the node voltages as $i_b = -v_1/2$, the node equation is set up and solved as

$$\begin{bmatrix} 1+1/2+1 & -1 \\ -1 & 1+1/2 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} -K i_b \\ K i_b + I_T \end{bmatrix} = \begin{bmatrix} K v_1/2 \\ -K v_1/2 + I_T \end{bmatrix}; \quad \begin{bmatrix} 5-K & -2 \\ K-2 & 3 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} 0 \\ 2I_T \end{bmatrix} \quad (\text{E2.27.6})$$

$$\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \frac{1}{11-K} \begin{bmatrix} 3 & 2 \\ 2-K & 5-K \end{bmatrix} \begin{bmatrix} 0 \\ 2I_T \end{bmatrix} = \frac{2I_T}{11-K} \begin{bmatrix} 2 \\ 5-K \end{bmatrix} \quad (\text{E2.27.7})$$

This result can be used to obtain the expression of the test voltage in terms of the test current as

$$V_T = v_2 = \frac{2(5-K)}{11-K} I_T \quad (\text{E2.27.8})$$

Matching this relation with Equation (2.14) yields the output resistance as

$$R_o = Z_{Th} = \frac{2(5-K)}{11-K} \quad (\text{E2.27.9})$$

Finally, we can find the conditions for $R_o = 0$ and $R_o = \infty$ as

$$K = 5 \quad \text{and} \quad K = 11 \quad (\text{E2.27.10})$$

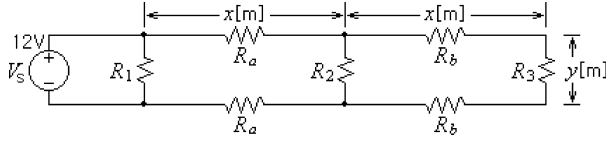


Figure 2.45 A resistor circuit for a defroster

(Example 2.28) Design of a Window Defroster for Uniform Heating

Consider the resistor circuit for a window defroster shown in Figure 2.45. Find the expressions of R_a , R_b , R_1 , and R_2 in terms of R_3 and $\sigma = x/y$ such that the power dissipated by each resistor is proportional to the length of the grid element that it is responsible for heating; in other words, the powers dissipated by the resistors R_a and R_b must be $\sigma = x/y$ times as much as that dissipated by R_1 , R_2 , or R_3 .

Noting that the circuit is a kind of ladder network as discussed in Example 2.25, a start is made from the part farthest from the source, i.e. the R_b - R_3 - R_b series connection. Since the three resistors carry the same current, say i_3 , the condition for their powers can be written as

$$\frac{R_b i_3^2}{R_3 i_3^2} = \frac{x}{y} = \sigma; \quad \frac{R_b}{R_3} = \sigma; \quad R_b = \sigma R_3 \quad (\text{E2.28.1})$$

Since $2R_b$ - R_3 and R_2 in parallel have a common voltage, say v_2 , the condition for their powers can be written as

$$\begin{aligned} \frac{v_2^2/(2R_b + R_3)}{v_2^2/R_2} &= \frac{2x + y}{y} = 2\sigma + 1; & \frac{R_2}{2R_b + R_3} &= 2\sigma + 1; \\ R_2 &= (2\sigma + 1)(2R_b + R_3) \stackrel{(\text{E2.28.1})}{=} (2\sigma + 1)^2 R_3 \end{aligned} \quad (\text{E2.28.2})$$

Note also that the parallel combination of $2R_b$ - R_3 and R_2 is

$$R_{23} = (2R_b + R_3) || R_2 \stackrel{(\text{E2.28.2})}{=} (2\sigma + 1)R_3 || (2\sigma + 1)^2 R_3 = \frac{(2\sigma + 1)^2}{2(\sigma + 1)} R_3 \quad (\text{E2.28.3})$$

Further, the power condition for the series $2R_a$ - R_{23} connection can be written as

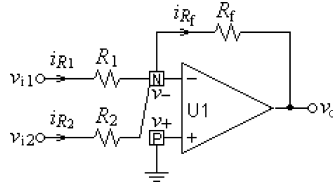
$$\begin{aligned} \frac{R_{23} i_a^2}{R_a i_a^2} &= \frac{2x + 2y}{x} = \frac{2(\sigma + 1)}{\sigma} \stackrel{(\text{E2.28.3}), (\text{E2.28.1})}{\rightarrow} \frac{(2\sigma + 1)^2 R_3}{2R_a(\sigma + 1)} = \frac{2(\sigma + 1)}{\sigma}; \\ R_a &= \frac{\sigma(2\sigma + 1)^2}{4(\sigma + 1)^2} R_3 \end{aligned} \quad (\text{E2.28.4})$$

Note that the series combination of $2R_a$ and R_{23} is

$$R_{a23} = 2R_a + R_{23} \stackrel{(\text{E2.28.3}), (\text{E2.28.4})}{=} 2 \frac{(2\sigma + 1)^2 \sigma}{4(\sigma + 1)^2} R_3 + \frac{(2\sigma + 1)^2}{2(\sigma + 1)} R_3 = \frac{(2\sigma + 1)^3}{2(\sigma + 1)^2} R_3 \quad (\text{E2.28.5})$$

Since $2R_a$ - R_{23} and R_1 in parallel have the common voltage V_s , the condition for their powers can be written as

$$\begin{aligned} \frac{V_s^2/(2R_a + R_{23})}{V_s^2/R_1} &= \frac{2(2x + y)}{y}; & \frac{R_1}{2R_a + R_{23}} &= 2(2\sigma + 1); \\ R_1 &= 2(2\sigma + 1)(2R_a + R_{23}) \stackrel{(\text{E2.28.5})}{=} \frac{(2\sigma + 1)^4}{(\sigma + 1)^2} R_3 \end{aligned} \quad (\text{E2.28.6})$$

**Figure 2.46** Summing amplifier circuit**(Example 2.29) Summing Amplifier – OP Amp Application**

Find the expression of the output voltage v_o in terms of the two input voltages v_{i1} and v_{i2} of the OP Amp circuit shown in Figure 2.46. Since the OP Amp has a negative feedback path with no positive feedback path, the practical analysis rules (Remark 2.2) can be used for this OP Amp circuit. Since the $+$ input terminal is grounded so that $v_+ = 0$, the virtual short principle can be applied to write

$$v_- = v_+ = 0 \quad (\text{E2.29.1})$$

Thus the currents through R_1 and R_2 are found to be

$$i_{R1} = \frac{v_{i1} - v_-}{R_1} \stackrel{(\text{E2.29.1})}{=} \frac{1}{R_1} v_{i1}, \quad i_{R2} = \frac{v_{i2} - v_-}{R_2} \stackrel{(\text{E2.29.1})}{=} \frac{1}{R_2} v_{i2} \quad (\text{E2.29.2})$$

Since by the virtual open principle no current flows into or out of the negative input terminal of the OP Amp, all these currents flow through the feedback resistor R_f to the output node so that the output voltage v_o can be obtained by subtracting the voltage drop across R_f from v_- as

$$v_o = v_- - R_f(i_{R1} + i_{R2}) \stackrel{(\text{E2.29.1})(\text{E2.29.2})}{=} 0 - R_f \left(\frac{1}{R_1} v_{i1} + \frac{1}{R_2} v_{i2} \right) = -\frac{R_f}{R_1} v_{i1} - \frac{R_f}{R_2} v_{i2} \quad (\text{E2.29.3})$$

This is an experience-based logical approach, which may become difficult to use for more complex OP Amp circuits. A systematic approach is to apply KCL to as many nodes as unknown node voltages. For the circuit of Figure 2.46, the output voltage is only one unknown so a single node equation is needed that can be written by applying KCL to the node N, i.e. the negative input terminal:

$$-i_{R1} - i_{R2} + i_{Rf} = 0; \quad \frac{v_- - v_{i1}}{R_1} + \frac{v_- - v_{i2}}{R_2} + \frac{v_- - v_o}{R_f} \stackrel{(\text{E2.29.1})}{=} \frac{-v_{i1}}{R_1} + \frac{-v_{i2}}{R_2} + \frac{-v_o}{R_f} = 0 \quad (\text{E2.29.4})$$

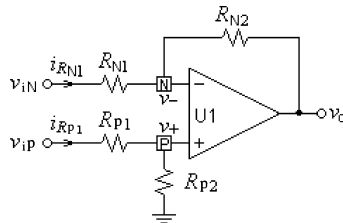
Solving this equation for v_o yields the same solution as (E2.29.3).

(Example 2.30) Difference Amplifier – OP Amp Application

Consider the OP Amp circuit of Figure 2.47 in which the average and difference of two input voltages v_{iP} and v_{iN} are defined as the common and differential modes, respectively, as follows:

$$\text{Common mode : } v_{cm} = \frac{1}{2} (v_{iP} + v_{iN}) \quad (\text{E2.30.1})$$

$$\text{Differential mode : } v_{dm} = v_{iP} - v_{iN} \quad (\text{E2.30.2})$$

**Figure 2.47** Difference amplifier circuit

- (a) Find the expression of the output voltage v_o in terms of the two input voltages v_{iP} and v_{iN} . Note that by the virtual open principle no current flows into or out of the positive input terminal of the OP Amp so that the node voltage v_+ at the + input terminal (node P) is determined by the voltage divider rule as

$$v_+ = \frac{R_{P2}}{R_{P1} + R_{P2}} v_{iP} \quad (\text{E2.30.3})$$

Since the OP Amp has a negative feedback path via R_{N2} , but no positive feedback path, the virtual short principle says that the negative input voltage is (almost) equal to the positive input voltage

$$v_- = v_+ \stackrel{(\text{E2.30.3})}{=} \frac{R_{P2}}{R_{P1} + R_{P2}} v_{iP} \quad (\text{E2.30.4})$$

so that the current through R_{N1} is found to be

$$i_{R_{N1}} = \frac{v_{iN} - v_-}{R_{N1}} \stackrel{(\text{E2.30.4})}{=} \frac{1}{R_{N1}} \left(v_{iN} - \frac{R_{P2}}{R_{P1} + R_{P2}} v_{iP} \right) \quad (\text{E2.30.5})$$

Since by the virtual open principle no current flows into or out of the negative input terminal (node N) of the OP Amp, all this current flows through R_{N2} to the output node so that the output voltage v_o can be obtained by subtracting the voltage drop across R_{N2} from v_- as

$$v_o = v_- - R_{N2} i_{R_{N1}} \stackrel{(\text{E2.30.4}), (\text{E2.30.5})}{=} \frac{R_{P2}}{R_{P1} + R_{P2}} v_{iP} - \frac{R_{N2}}{R_{N1}} \left(v_{iN} - \frac{R_{P2}}{R_{P1} + R_{P2}} v_{iP} \right) \quad (\text{E2.30.6})$$

This output voltage can be written in terms of v_{cm} and v_{dm} as

$$v_o = \frac{R_{P2}(R_{N1} + R_{N2})}{R_{N1}(R_{P1} + R_{P2})} v_{iP} - \frac{R_{N2}}{R_{N1}} v_{iN} = A_{cm} v_{cm} + A_{dm} v_{dm} \quad (\text{E2.30.7})$$

where

$$\text{Common mode gain : } A_{cm} = \frac{R_{N1} R_{P2} - R_{N2} R_{P1}}{R_{N1}(R_{P1} + R_{P2})} \quad (\text{E2.30.8a})$$

$$\text{Differential mode gain : } A_{dm} = \frac{R_{P2}(R_{N1} + R_{N2}) + R_{N2}(R_{P1} + R_{P2})}{2R_{N1}(R_{P1} + R_{P2})} \quad (\text{E2.30.8b})$$

A systematic approach is to apply KCL to nodes N and P to write the node equations in the two unknown node voltages v_o and $v_- = v_+$ (see Problem 2.18).

- (b) Find the condition under which only the differential mode appears at the output. The common mode gain can be set to zero to find the condition as

$$A_{cm} = \frac{R_{N1} R_{P2} - R_{N2} R_{P1}}{R_{N1}(R_{P1} + R_{P2})} = 0; \quad R_{N1} R_{P2} - R_{N2} R_{P1} = 0; \quad \frac{R_{P2}}{R_{P1}} = \frac{R_{N2}}{R_{N1}} = K \quad (\text{E2.30.9})$$

This yields the value of the differential mode gain as

$$A_{dm} = \frac{R_{N2}(R_{P1} + R_{P2}) + R_{N2}(R_{P1} + R_{P2})}{2R_{N1}(R_{P1} + R_{P2})} = \frac{R_{N2}}{R_{N1}} = K \quad (\text{E2.30.10})$$

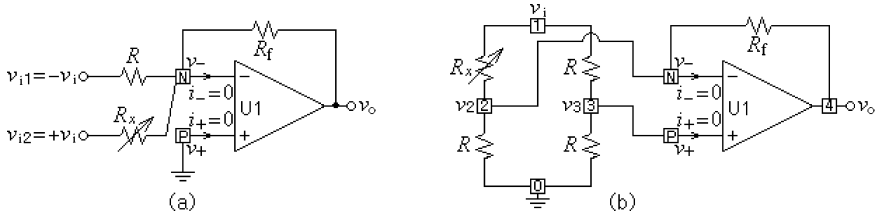


Figure 2.48 Circuits for Example 2.31 (From Reference [T-1]. Source: © Prentice-Hall)

(Example 2.31) Design and Evaluation of the OP Amp Resistor Circuit (Source: R. E. Thomas and A. J. Rosa, *The Analysis and Design of Linear Circuits*, 1994. Source: © Prentice-Hall)

The OP Amp circuits shown in Figure 2.48 contain a photo resistor, R_x , the resistance of which varies with the intensity of illumination (Reference [T-1] or [T-2]).

- (a) Express the output voltage v_o of the circuit in Figure 2.48(a) in terms of the input voltage v_i . Noting that the circuit is a summing amplifier like the one shown in Figure 2.46 and dealt with in Example 2.29, Equation (E2.29.3) can be used to find the output voltage as

$$v_o \stackrel{(E2.29.3)}{=} -\frac{R_f}{R} v_{i1} - \frac{R_f}{R_x} v_{i2} = \left(\frac{1}{R} - \frac{1}{R_x} \right) R_f v_i \quad (E2.31.1)$$

- (b) Express the output voltage v_o of the circuit in Figure 2.48(b) in terms of the input voltage v_i . Since the circuit has a negative feedback, but no positive feedback, the practical analysis rules summarized in Remark 2.2 can be applied. Noting that $v_2 = v_- = v_+ = v_3$ (by the virtual short principle) and $i_- = i_+ = 0$ (by the virtual open principle), KCL can be applied to nodes 2 and 3 to write the node equations in the two unknown voltages v_2 and v_o as

$$\text{Node 2 : } \frac{v_2 - v_i}{R_x} + \frac{v_2 - 0}{R} + \frac{v_2 - v_o}{R_f} = 0 \quad (E2.31.2)$$

$$\text{Node 3 : } \frac{v_2 - v_i}{R} + \frac{v_2 - 0}{R} = 0; \quad 2v_2 - v_i = 0; \quad v_2 = \frac{v_i}{2} \quad (E2.31.3)$$

In fact, Equation (E2.31.3) can easily be obtained by applying the voltage divider rule to the path of node 1– R –node 3– R –node 0. Equation (E2.31.3) can be substituted into Equation (E2.31.2) to find v_o as

$$v_o \stackrel{(E2.31.2), (E2.31.3)}{=} \frac{1}{2} \left(1 - \frac{R_f}{R_x} + \frac{R_f}{R} \right) v_i \quad (E2.31.4)$$

- (c) Let the input voltage be $v_i = 10 \text{ V}$ and the resistance of the photo resistor be

$$R_x = \begin{cases} 1 \text{ k}\Omega & \text{in daylight} \\ 3 \text{ k}\Omega & \text{at night} \end{cases} \quad (E2.31.5)$$

Determine the values of R and R_f of the circuits in Figures 2.48(a) and (b) such that the output voltage becomes -10 V and $+10 \text{ V}$ in daylight and at night, respectively.

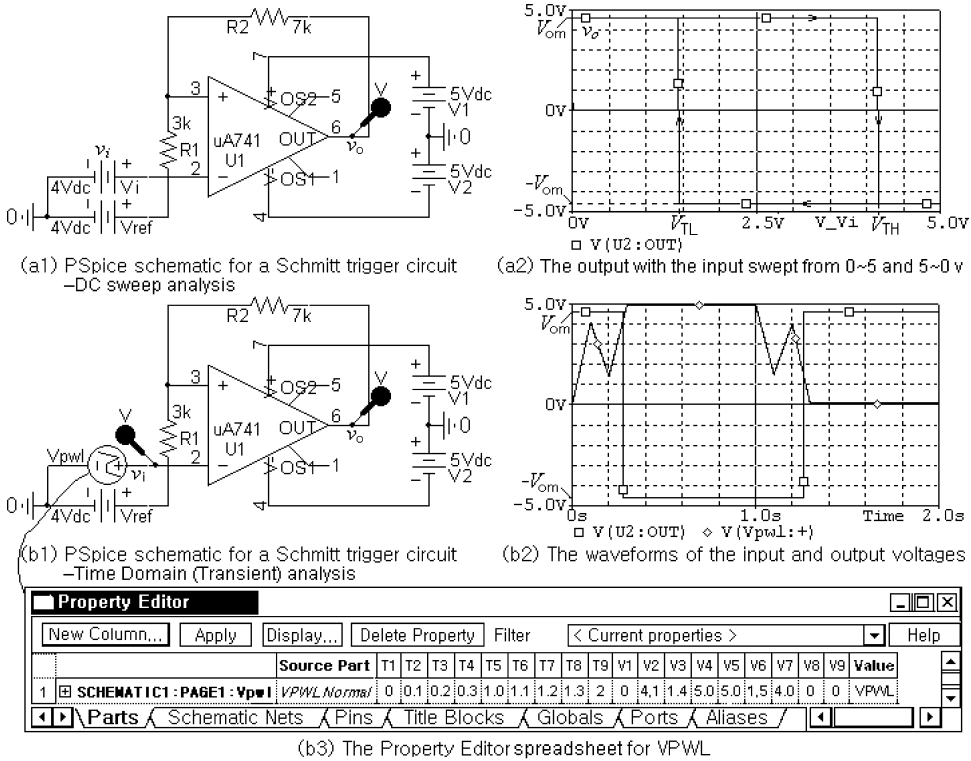


Figure 2.49 PSpice schematics and simulation results for Example 2.31

With Equation (E2.31.1) for Figure 2.48(a), this condition can be written as

$$\left. \begin{aligned} \left(\frac{1}{R} - \frac{1}{1} \right) R_f \times 10 = -10 : (1) \quad (2) - (1) : \frac{2}{3} R_f \times 10 = 20; R_f = 3 \text{ k}\Omega : (3) \\ \left(\frac{1}{R} - \frac{1}{3} \right) R_f \times 10 = 10 : (2) \quad (1) \xrightarrow{(3)} \left(\frac{1}{R} - \frac{1}{1} \right) 3 \times 10 = -10; R = \frac{3}{2} \text{ k}\Omega \end{aligned} \right\} \quad (\text{E2.31.6a})$$

With Equation (E2.31.4), this condition can be written as

$$\left. \begin{aligned} \frac{1}{2} \left(1 - \frac{R_f}{1} + \frac{R_f}{R} \right) 10 = -10 : (1) \quad (2) - (1) : \frac{1}{3} R_f \times 10 = 20; R_f = 6 \text{ k}\Omega : (3) \\ \frac{1}{2} \left(1 - \frac{R_f}{3} + \frac{R_f}{R} \right) 10 = 10 : (2) \quad (1) \xrightarrow{(3)} \frac{1}{2} \left(1 - \frac{6}{1} + \frac{6}{R} \right) 10 = -10; R = 2 \text{ k}\Omega \end{aligned} \right\} \quad (\text{E2.31.6b})$$

- (d) Compare the two configurations in Figures 2.48(a) and (b) in the aspects of number of parts and power dissipation. Since (a) needs fewer resistors than (b), it is better in the number of parts. Noting that the voltages across R and R_x are 10 V in (a), while they are 5 V in (b), their power dissipations can be written as

$$P_a = \frac{v_i^2}{R} + \frac{v_i^2}{R_x} + \frac{(\pm 10)^2}{R_f} = \frac{10^2}{3/2} + \frac{10^2}{1 \sim 3} + \frac{10^2}{3} = \left(\frac{2}{3} + \frac{1}{1 \sim 3} + \frac{1}{3} \right) 100 = \frac{400}{3} \sim 200 \text{ W} \quad (\text{E2.31.7a})$$

$$P_b = 3 \frac{(v_i/2)^2}{R} + \frac{(v_i/2)^2}{R_x} + \frac{(v_i/2 \pm 10)^2}{R_f} = 3 \frac{5^2}{2} + \frac{5^2}{1 \sim 3} + \frac{25 \sim 225}{6} = 50 \sim \frac{500}{6} \text{ W} \quad (\text{E2.31.7b})$$

This indicates that the power dissipation in (b) is less than 50% of that in (a) on the average.

(Example 2.32) Simulation of OP Amp Circuit with Positive Feedback – Schmitt Trigger

Figure 2.49(a1) shows the PSpice schematic for a Schmitt trigger circuit, which is slightly different from the circuit of Figure 2.35(a) in that a DC voltage source of $V_{\text{ref}} = 4 \text{ V}$ is connected in series with R_1 . The node voltage at the + input terminal of the OP Amp determining the threshold voltages also differs from Equation (2.38) as

$$v_+ = V_{\text{ref}} + b(v_o - V_{\text{ref}}) = (1 - b)V_{\text{ref}} + bv_o \quad \text{with } b = \frac{R_1}{R_1 + R_2} \quad (\text{E2.32.1})$$

- (a) Find the two (higher/lower) threshold values of the input voltage to reverse the output voltage. The two (higher/lower) threshold values are obtained by substituting $v_o = \pm V_{\text{om}}$ into Equation (E2.32.1) as

$$V_{\text{TH}}, V_{\text{TL}} = (1 - b)V_{\text{ref}} \pm bV_{\text{om}} \quad (\text{E2.32.2})$$

- (b) With $\pm V_{\text{om}} = \pm 4.6 \text{ V}$, find the values of b and V_{ref} such that the higher/lower threshold values are $V_{\text{TH}} = 4.18 \text{ V}$ and $V_{\text{TL}} = 1.42 \text{ V}$, respectively. Then

$$\begin{aligned} V_{\text{TH}} &= (1 - b)V_{\text{ref}} + 4.6b = 4.18 \\ V_{\text{TL}} &= (1 - b)V_{\text{ref}} - 4.6b = 1.42 \end{aligned} \quad (\text{E2.32.3})$$

is solved to get $b = 0.3$ and $V_{\text{ref}} = 4 \text{ V}$.

- (c) The following steps are taken to perform the PSpice simulation (for the DC Sweep analysis) in order to get the transfer characteristic curve describing the overall input–output relationship shown in Figure 2.49(a2):
1. Draw the PSpice schematic as depicted in Figure 2.49(a1).
 2. Set the Analysis type to DC Sweep, the sweep variable to V_i , the sweep type to Linear, and the sweep values to (Start value: 0, End value: 5, Increment: 0.01) in the Simulation Settings dialog box and run the simulation.
 3. Change the sweep values to (Start value: 5, End value: 0, Increment: -0.01) in the Simulation Settings dialog box and run the simulation.
- (d) The following steps are taken to perform the PSpice simulation (Transient analysis) to get the input and output waveforms shown in Figure 2.49(b2), which shows the *debouncing* feature of the Schmitt trigger due to its *deadband* characteristic.
1. Draw the PSpice schematic as depicted in Figure 2.49(b1).
 2. Double-click the VPWL (piecewise linear voltage source) to open the Property Editor spreadsheet and set the values of the parameters (T1,V1), (T2,V2), $\dots$, as shown in Figure 2.49(b3). If needed, click the New Column button to create new columns like (T9,V9).
 3. Set the Analysis type to Time Domain (Transient), Run_to_time to 2 s, and Maximum step to 0.1 ms in the Simulation Settings dialog box and run the simulation.

Problems

2.1 Series and Parallel Combination of Resistors

- (a) Among the following set of resistance values, identify those that cannot be obtained from any connection of the resistor array in Figure P2.1(a): $\{1/4, 1/3, 2/5, 1/2, 2/3, 3/4, 4/5, 1, 5/4, 4/3, 3/2, 2, 5/2, 3, 4\}$.

Note. If you do not like to enumerate every possible combination of the resistors and compute their combined resistance, you may save the following three routines in three M-files each named `possible_comb.m`, `contain.m`, and `rotate_r.m`, together with the `parallel_comb.m` given in Section 2.2 in a directory that MATLAB can search, and then type the following statements into the MATLAB command window.

```
>>format rat, possible_comb([], [1 1 1 1])
```

```
function Set=possible_comb(Set,A)
NA=length(A);
if NA==1; Set= contain(Set,A);
elseif NA==2
    Set= contain(Set,A(1)); Set= contain(Set,A(2));
    Set= contain(Set,sum(A));
    Set= contain(Set,parallel_comb(A));
else
    for i=1:length(A)
        Set= contain(Set,A(i));
        Set= possible_comb(Set,[sum(A(1:2)) A(3:end)]);
        Set= possible_comb(Set,[parallel_comb(A(1:2)) A(3:end)]);
        A= rotate_r(A,1);
    end
end

function A=contain(A,a)
contained=0;
for i=1:length(A)
    if abs(a-A(i))<eps, contained=1; break; end
end
if contained==0, A=[A a]; end

function x1=rotate_r(x,M)
N=size(x,2); M=mod(M,N);
x1=[x(:,end-M+1:end) x(:,1:end-M)];
```

- (b) Among the following set of combinations, identify those that cannot be obtained from any connection of the resistor array in Figure P2.1(b): $\{R_1 + R_2 + R_3, (R_1 + R_2) \parallel R_3, (R_1 \parallel R_2) + R_3, R_1 \parallel R_2 \parallel R_3\}$.
- (c) Do the same job as in (b) for the resistor array in Figure P2.1(c).

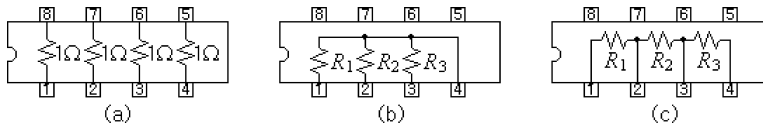


Figure P2.1

2.2 Design of a Voltage/Current Divider for a Multiscale Voltmeter/Ammeter

- (a) Let the 1 mA ammeter (current meter) have an internal resistance of $R_M = 50 \Omega$ and a full-scale current of $I_{FS} = 1$ mA in the configuration of Figure P2.2(a). Determine the values of R_1 and R_2 such that the ammeter can be used to measure a voltage with dual full scales of 50 V or 5 V depending on which side the switch is connected to.
- (b) Let the 1 mA ammeter (current meter) have an internal resistance of $R_M = 10\,989 \Omega$ and a full-scale current of $I_{FS} = 1$ mA in the configuration of Figure P2.2(b). Determine the values of R_a

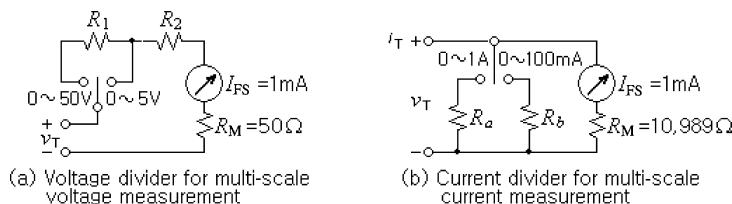


Figure P2.2

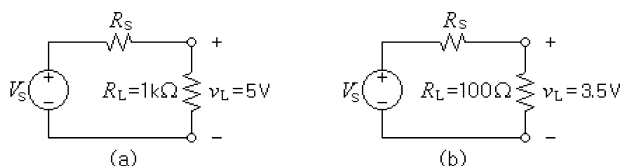


Figure P2.3

and R_b such that the ammeter can be used to measure a current with dual full scales of 1 A or 100 mA depending on which side the switch is connected to.

2.3 Determination of Source Impedance (Internal Resistance) of a Voltage Source

Suppose a DC voltage source supplies a voltage $v_L = 5\text{ V}$ for a load of resistance $R_L = 1\text{ k}\Omega$ and $v_L = 3.5\text{ V}$ for $R_L = 100\Omega$, as depicted in Figure P2.3. Find the value of the source impedance R_s .

2.4 Design of an Interface with Given Output Voltage and Output Resistance

- For the configuration shown in Figure P2.4(a), determine the values of R_1 and R_2 such that the output voltage is 10 V and the output resistance is $5\text{ k}\Omega$.
- For the configuration shown in Figure P2.4(b), determine the values of R_1 and R_2 such that the output voltage is 10 V and the output resistance is $(5/6)\text{ k}\Omega$.

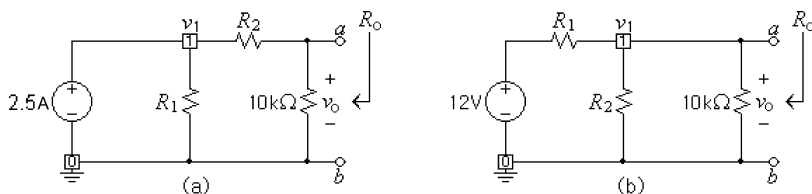


Figure P2.4

2.5 Resistance Combination Using Y- Δ (T-II) Conversion

- Find the equivalent resistance of the bridge circuit of Figures P2.5(a) or (b) (without the source) seen from terminals 1 and 0 using a Y- Δ (T-II) conversion.

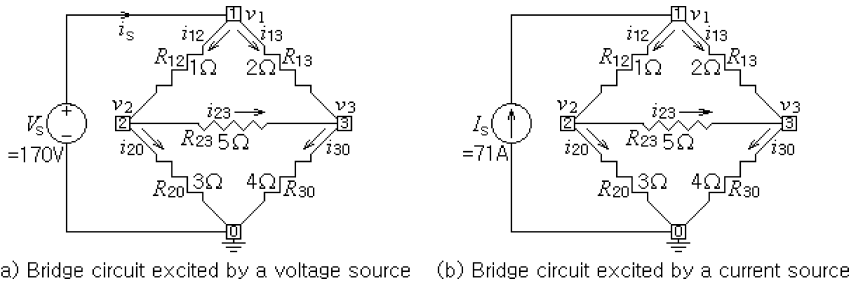


Figure P2.5

- (b) Find the node voltages, v_1 , v_2 , and v_3 , and the branch currents, i_{12} , i_{13} , i_{20} , i_{30} , and i_{23} , in the circuits of Figures P2.5(a) and (b).

2.6 Node Analysis and Mesh Analysis

- Make the voltage-to-current source transformation of the 170 V voltage source and use the node analysis to find the current i_{23} through the 5Ω resistor in the circuit of Figure P2.5(a).
- Use the node analysis to find the current i_{23} through the 5Ω resistor in the circuit of Figure P2.5(b).
- Use the mesh analysis to find the current i_{23} through the 5Ω resistor in the circuit of Figure P2.5(a).
- Make the current-to-voltage source transformation of the 71 A current source and use the mesh analysis to find the current i_{23} through the 5Ω resistor in the circuit of Figure P2.5(b).

2.7 Node Analysis and Mesh Analysis

- Use the node analysis to find the node voltage v_2 and the mesh current i_1 in the circuit of Figure P2.7.
- Use the mesh analysis to find the node voltage v_2 and the mesh current i_1 in the circuit of Figure P2.7.

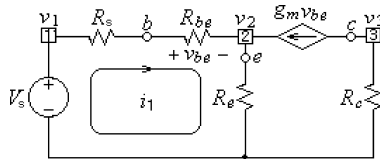


Figure P2.7

2.8 Millman's Theorem on Equivalent Circuits

Consider the circuit shown in Figure P2.8(a).

- Use the node analysis to find the node voltage v_1 at node 1.
- Use the mesh analysis to find the mesh currents i_1 and i_2 and use the result to find v_1 .
- Referring to Figures P2.8(b) and (c), find the Norton and Thevenin equivalents.

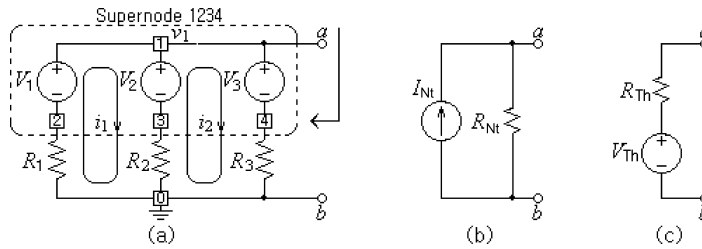


Figure P2.8 Millman's theorem

2.9 Thevenin Equivalent Circuit

- Find the Thevenin equivalent seen from terminals 2 and 3 of the circuit of Figure P2.5(a) without the 5Ω resistor. Use the Thevenin equivalent to find the current i_{23} through the 5Ω resistor.
- Find the Thevenin equivalent seen from terminals 2 and 3 of the circuit of Figure P2.5(b) without the 5Ω resistor. Use the Thevenin equivalent to find the current i_{23} through the 5Ω resistor.

2.10 Thevenin Equivalent Circuit

Find the Thevenin equivalent of the circuit of Figure P2.10 seen from terminals 2 and 3, where ΔR is assumed to be much less than R .

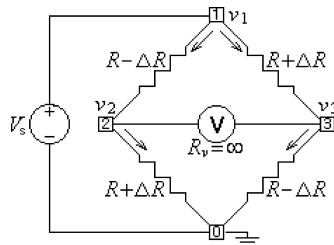


Figure P2.10

2.11 Charley's Thinking on the Thevenin Equivalent Circuit

As discussed in Example 2.17, there is generally a need for an adjustable and readable voltage source (with a voltmeter) and an ammeter to find the Thevenin equivalent of a network, especially if it contains a dependent source. If the network has an independent source, only a voltmeter and an ammeter are needed since the Thevenin voltage source is just the open-circuit voltage $V_{Th} = V_{oc}$ and the Thevenin equivalent impedance can be obtained from dividing the open-circuit voltage by the short-circuit current as $Z_{Th} = V_{oc}/I_{sc}$ (see Equation (2.13)). However, Charley, who had previously burnt out an ammeter when measuring a large current over its rated current and received a warning from the lab T.A. that he would fail the lab course with another mistake, is worried that short-circuiting the terminals of the network results in a large current exceeding the rated current of the ammeter, which may cause the ammeter to be damaged. Therefore, instead of using an ammeter, he connected a 20Ω resistor across the terminals and measured the terminal voltage $V_{ab} = 2\text{ V}$ after

measuring the open-circuit voltage $V_{oc} = 5\text{ V}$ (see Figure P2.11). Find the equivalent resistance of the network based on his data, unless you disagree with him.

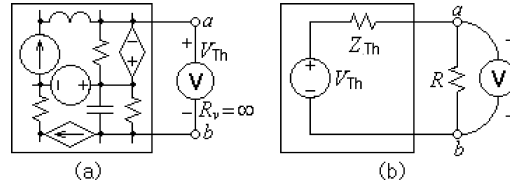


Figure P2.11 Finding the Thevenin equivalent

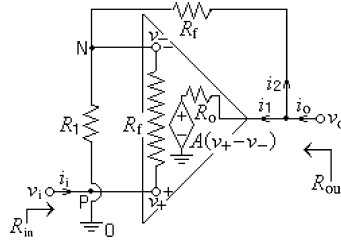


Figure P2.12 A noninverting OP Amp circuit

2.12 Equivalent of a Circuit Containing a Dependent (Controlled) Source

Consider the OP Amp circuit in Figure P2.12, where the OP Amp is replaced with a practical model.

(a) Assuming that $R_1 = \infty$, verify that the output resistance seen from the output port is as follows:

$$R_{out} = \left. \frac{v_o}{i_o} \right|_{v_+ = v_i = 0} = \frac{v_o}{i_1 + i_2} = \frac{v_o}{[v_o - (-Av_-)]/R_o + v_o/(R_1 + R_f)}$$

$$= \frac{1}{\frac{1}{R_o} + \frac{AR_1}{R_o(R_1 + R_f)} + \frac{1}{R_1 + R_f}} = \frac{R_o}{1 + \frac{AR_1}{R_1 + R_f} + \frac{R_o}{R_1 + R_f}} \quad (\text{P2.12.1})$$

$$\stackrel{R_o \ll R_1, R_f}{\cong} \frac{R_o}{1 + AR_1/(R_1 + R_f)} \ll R_o \quad (\text{P2.12.2})$$

(b) Verify that the input resistance seen from the input port is obtained as follows:

$$\frac{v_- - v_+}{R_1} + \frac{v_- - A(v_+ - v_-)}{R_o + R_f} + \frac{v_-}{R_1} = 0 \quad (\text{with } v_+ = v_i); \quad (\text{P2.12.3})$$

$$\left(\frac{1}{R_1} + \frac{A+1}{R_o + R_f} + \frac{1}{R_1} \right) v_- = \left(\frac{1}{R_1} + \frac{A}{R_o + R_f} \right) v_i$$

$$v_- = \frac{R_1(R_o + R_f + AR_1)}{R_1(R_o + R_f) + (A+1)R_1R_1 + R_1(R_o + R_f)} v_i \quad (\text{P2.12.4})$$

$$i_i = \frac{v_i - v_-}{R_1} = \frac{R_1 + R_o + R_f}{R_1(R_o + R_f) + (A+1)R_1R_1 + R_1(R_o + R_f)} v_i \quad (\text{P2.12.5})$$

$$R_{in} = \frac{v_i}{i_i} = \frac{R_1(R_o + R_f) + (A + 1)R_1 R_1 + R_1(R_o + R_f)}{R_1 + R_f + R_o} \quad (\text{P2.12.6})$$

$$R_o \ll R_1, R_f \ll R_1 \quad \frac{(A + 1)R_1 + R_f}{R_1 + R_f} R_1 = \left(1 + \frac{A R_1}{R_1 + R_f}\right) R_1 \gg R_1 \quad (\text{P2.12.7})$$

Note. These results indicate that the input impedance of a noninverting OP Amp circuit is much larger than that of the OP Amp itself and its output impedance is much smaller than that of the OP Amp itself.

2.13 Equivalent Circuit

Find the Thevenin equivalent of the circuit seen from terminals *c-d* in Figure P2.13.

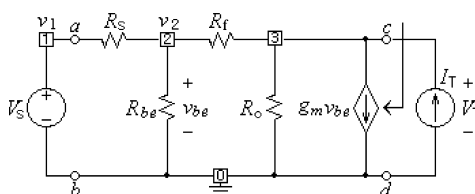


Figure P2.13

2.14 Equivalent Circuit

Find the Thevenin equivalent of the circuit seen from terminals B and 0 in Figure P2.14 so that the circuit can be transformed equivalently to the one in Figure 2.37(a).

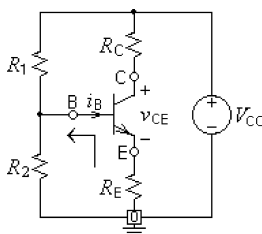


Figure P2.14

2.15 Equivalent of a Circuit Containing a Dependent (Controlled) Source

Consider the circuit of Figure P2.15.

- Find the Thevenin equivalent of the circuit seen from terminals 1 and 0 with the switch closed.
- Find the Thevenin equivalent of the circuit seen from terminals 1 and 0 with the switch open.

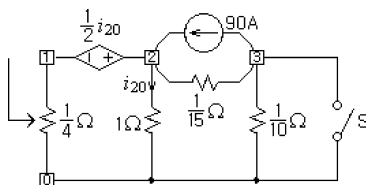


Figure P2.15

2.16 Equivalent of a Circuit Containing more than One Dependent (Controlled) Source

Find the Thevenin equivalent of the circuit seen from terminals 2 and 0 in Figure P2.16.

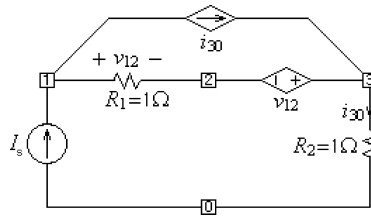


Figure P2.16

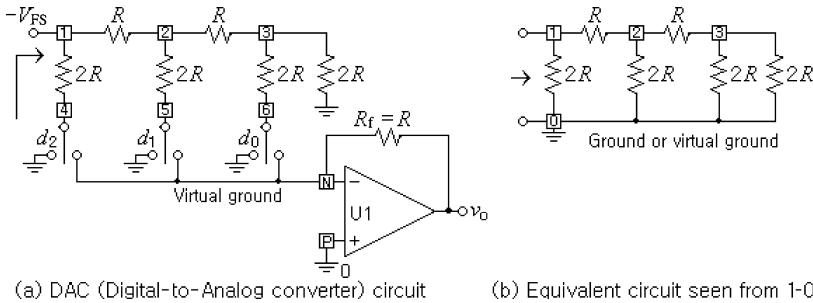


Figure P2.17

2.17 DAC (Digital-to-Analog Converter) Using OP Amp – Equivalent Circuit

Consider the OP Amp circuit of Figure P2.17(a) in which each of the three transfer switches is connected to the left side (the real ground) or the right side (node N), which is a virtual ground, depending on whether the value of the corresponding bit of the binary-coded digital input signal $d_2d_1d_0$ is 0 or 1. Note that no matter which side each transfer switch is connected to, the voltage at nodes 4, 5, and 6 is zero and, consequently, the currents flowing from nodes 1, 2, and 3 to the (negative) voltage source $-V_{FS}$ are always constant. What really matters is whether the current flows from the real ground or the output terminal through the feedback resistor R_f to make a contribution toward raising the output voltage v_o .

- Referring to Figure P2.17(b), find the equivalent resistance between node 1 and the real or virtual ground.
- Show that the currents flowing from nodes 4, 5, and 6 to the (negative) voltage source $-V_{FS}$ are always constant as $V_{FS}/(2R)$, $V_{FS}/(4R)$, and $V_{FS}/(8R)$, respectively.
- Show that the (analog) output voltage becomes $V_{FS}/2$, $V_{FS}/4$, and $V_{FS}/8$ for the digital input signal $d_2d_1d_0 = 100$, 010 , and 001 , respectively.

2.18 Systematic Approach to the Difference Amplifier of Example 2.30 (Figure 2.47)

Consider again the OP Amp circuit, which was shown in Figure 2.47 and dealt with in Example 2.30. As a systematic approach, apply KCL to nodes N and P to set up the node equations in the two unknown node voltages, v_o and $v_- = v_+$, and check whether they yield the same expression for the output voltage v_o as Equation (E2.30.6).

2.19 Difference Amplifier Using OP Amp

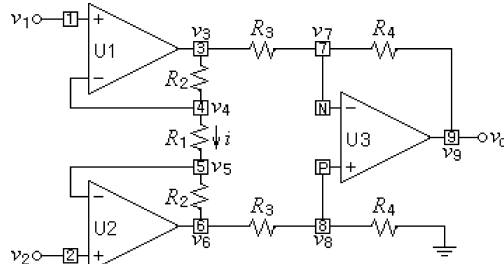


Figure P2.19 Difference amplifier

Consider the OP Amp circuit shown in Figure P2.19.

- (a) Show that the current through R_2 – R_1 – R_2 is

$$i = (v_1 - v_2)/R_1 \quad (\text{P2.19.1})$$

- (b) Show that the output voltage v_o is expressed in terms of the difference between the two input voltages as

$$v_o = \frac{R_4}{R_3} \left(1 + \frac{2R_2}{R_1} \right) (v_2 - v_1) \quad (\text{P2.19.2})$$

2.20 Analysis and Design of General Combiner Using an OP Amp

- (a) Verify the following for the OP Amp circuit of Figure P2.20(a):

– The voltage at the positive input terminal is

$$v_+ = (R_{P1} || R_{P2}) \left(\frac{v_{P1}}{R_{P1}} + \frac{v_{P2}}{R_{P2}} \right) \quad (\text{P2.20.1})$$

– With $v_{N1} = v_{N2} = 0$, the output voltage is

$$v_{oP} = (R_{P1} || R_{P2}) \left(\frac{v_{P1}}{R_{P1}} + \frac{v_{P2}}{R_{P2}} \right) \left(1 + \frac{R_f}{R_{N1} || R_{N2}} \right) \quad (\text{P2.20.2})$$

– With $v_{P1} = v_{P2} = 0$, the output voltage is

$$v_{oN} = - \left(\frac{R_f}{R_{N1}} v_{N1} + \frac{R_f}{R_{N2}} v_{N2} \right) \quad (\text{P2.20.3})$$

- (b) Verify that connecting one more resistor R_{P3} between the positive input terminal and the ground changes only Equation (P2.20.2) as follows (see Figure P2.20(b))

$$v_{oP} = (R_{P1} || R_{P2} || R_{P3}) \left(\frac{v_{P1}}{R_{P1}} + \frac{v_{P2}}{R_{P2}} \right) \left(1 + \frac{R_f}{R_{N1} || R_{N2}} \right) \quad (\text{P2.20.4})$$

Does it decrease the output voltage v_o ?

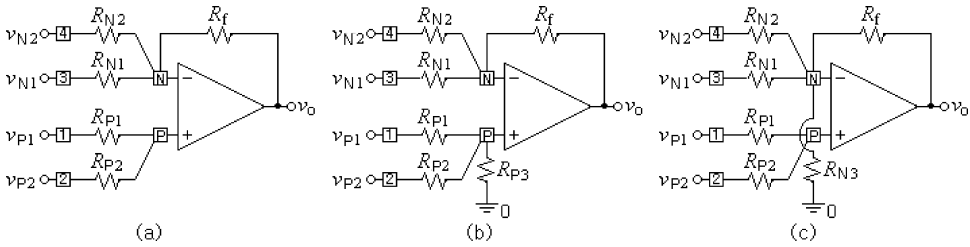


Figure P2.20

- (c) Verify that connecting one more resistor R_{N3} between the negative input terminal and the ground changes only Equation (P2.20.2) as follows (see Figure P2.20(c)):

$$v_{oP} = (R_{P1} \parallel R_{P2}) \left(\frac{v_{P1}}{R_{P1}} + \frac{v_{P2}}{R_{P2}} \right) \left(1 + \frac{R_f}{R_{N1} \parallel R_{N2} \parallel R_{N3}} \right) \quad (\text{P2.20.5})$$

Does it increase the output voltage v_o ?

- (d) Design a linear combiner realizing a given linear combination of several input signals, say,

$$v_o = a_1 v_{P1} + a_2 v_{P2} - b_1 v_{N1} - b_2 v_{N2} \quad \text{with } a_1, a_2, b_1, b_2 > 0 \quad (\text{P2.20.6})$$

as one of the two configurations in Figures P2.20(b) and (c) by the following procedure:

- Step 1. After choosing an appropriate value of the feedback resistance R_f , determine the values of R_{N1} and R_{N2} to be inversely proportional to the negative coefficients, b_1 and b_2 , such that the negative terms are implemented as Equation (P2.20.3).
- Step 2. Determine the values of R_{P1} and R_{P2} to be inversely proportional to the positive coefficients, a_1 and a_2 , such that the positive terms are implemented as Equation (P2.20.2).
- Step 3. If the coefficients obtained from Equation (P2.20.2) with the chosen values of resistors are larger/smaller than a_1 and a_2 , connect another resistor between the $+/-$ input terminal and the ground so that the positive coefficients obtained from Equations (P2.20.4)/(P2.20.5) become as small/large as required.

This procedure is cast into the following MATLAB function `design_combiner()`, which produces the values of the resistances to be connected to the $+/-$ input terminals for the positive/negative coefficients as $\mathbf{as} = [a_1 \ a_2 \ \dots]$ and $\mathbf{bs} = [b_1 \ b_2 \ \dots]$ given together with the value, R_f , of the feedback resistance R_f and the minimum value, R_{Pmin} , of R_{Pi} values.

```
function [RPs,RNs]=design_combiner(as,bs,Rf,RPmin)
% Design a linear combiner to realize vo = as*vPs - bs*vNs
% Input: positive coefficient vector as=[a1 a2...]
%       negative coefficient vector bs=[b1 b2...]
%       Rf: Feedback resistance
%       RPmin: Minimum value among RP1,RP2,...
% Output: resistances to be connected to the +/- input terminal[k ohm]
% Copyright: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only
if nargin<4, RPmin=1; end
if nargin<3, Rf=1; end
RNs= Rf./bs; amax=max(as); RPs= RPmin*amax./as;
```



```

composite_RPs=parallel_comb(RPs); composite_RNs=parallel_comb(RNs);
as_t=composite_RPs./RPs*(1+Rf/composite_RNs);
ratio=max(as_t)/amax
if ratio>1
    Rl=1/(ratio-1)*composite_RPs; RPs0=RPs; RPs=[RPs Rl];
    P_coefficients=parallel_comb(RPs)./RPs0*(1+Rf/composite_RNs);
elseif ratio<1
    scale=ratio*Rf/((1-ratio)*composite_RNs+Rf);
    Rl=scale/(1-scale)*composite_RNs; RNs=[RNs Rl];
    P_coefficients=composite_RPs./RPs*(1+Rf/parallel_comb(RNs));
end
combiner_coefficients=[P_coefficients -Rf./RNs(1:length(bs))]%Realized

```

For instance, a linear combiner can be designed that realizes

$$v_o = 2v_{P1} + v_{P2} - 2v_{N1} - v_{N2} \quad (\text{P2.20.7})$$

by typing the following statements into the MATLAB command window:

```

>>[RPs,RNs]=design_combiner([2 1],[2 1],2,1) % with Rf=2, RPsmin=1
RPs = 1.0000      2.0000      2.0000
RNs = 1           2

```

Note that the feedback resistance R_f and the minimum value of R_{Pi} values are given as $2 \text{ k}\Omega$ and $1 \text{ k}\Omega$. This result indicates that the designed values of the resistances are

$$R_{P1} = 1 \text{ k}\Omega, \quad R_{P2} = 2 \text{ k}\Omega, \quad R_{P3} = 2 \text{ k}\Omega, \quad R_{N1} = 1 \text{ k}\Omega, \quad \text{and} \quad R_{N2} = 2 \text{ k}\Omega$$

where R_{P3} is the resistance that is connected additionally in the configuration of Figure P2.20(b). Design a linear combiner that realizes

$$v_o = 4v_{P1} + 2v_{P2} - v_{N1} - 2v_{N2} \quad (\text{P2.20.8})$$

2.21 Fahrenheit-to-Centigrade Converter Using OP Amp

Consider the OP Amp circuit shown in Figure P2.21, where a constant voltage of 9.7 V is applied to node 1 and a variable voltage v_F to node 3. Find the output voltage v_{o3} of the OP Amp U3 in terms of v_F .

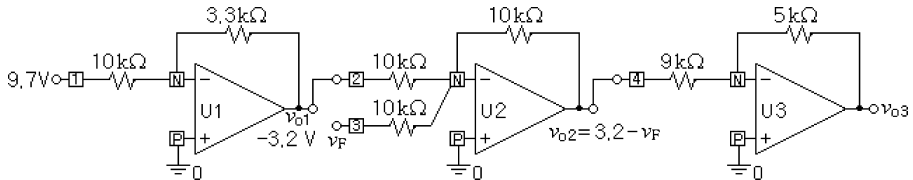


Figure P2.21 Fahrenheit-to-centigrade converter using OP Amps

2.22 Voltage Amplifier Using OP Amp

Consider the OP Amp circuit shown in Figure P2.22.

(a) Show that the output voltage v_o of the OP Amp can be expressed in terms of the input voltage v_i as

$$v_o = -\frac{R_2}{R_1} \left(1 + \frac{R_3}{R_2} + \frac{R_3}{R_4} \right) v_i \quad (\text{P2.22.1})$$

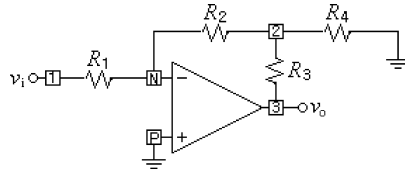


Figure P2.22 Voltage amplifier

- (b) With $R_1 = 1 \text{ k}\Omega$, $R_2 = 10 \text{ k}\Omega$, and $R_3 = 90 \text{ k}\Omega$, find the value of the resistance R_4 needed to attain a voltage gain $v_o/v_i = -1000$. Compare it with the value of R_f needed to attain the same voltage gain in the basic inverting OP Amp circuit of Figure 2.30, where $R_1 = 1 \text{ k}\Omega$.

Note. Compared with the basic inverting amplifier, this circuit can achieve the same voltage gain with smaller-valued resistors, while requiring more resistors.

2.23 Voltage Polarity Changer Using OP Amp

Consider the OP Amp circuit shown in Figure P2.23.

- (a) Find the voltage gain v_o/v_i when the switch is open.
 (b) Find the voltage gain v_o/v_i when the switch is closed.

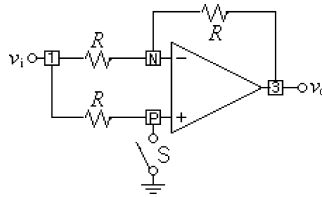


Figure P2.23 Voltage polarity changer

2.24 Feedforward–Feedback OP Amp Circuit

Show that the voltage gain of the OP Amp circuit shown in Figure P2.24 is

$$v_o = \frac{G_1 G_4 - G_3 G_5}{G_3 G_6 - G_2 G_4} v_i \quad (\text{P2.24.1})$$

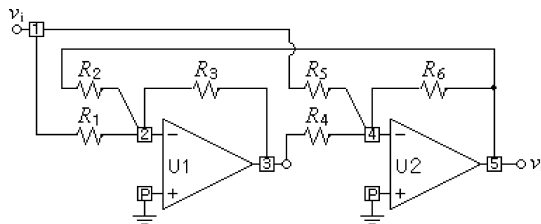


Figure P2.24 Feedforward–feedback OP Amp circuit

2.25 Load Current Controller Using OP Amp

Consider the OP Amp circuit of Figure P2.25. Show that the current through the load resistor R_L can be controlled by moving the sliding contact (wiper) of the potentiometer, which is a kind of user-adjustable resistance, as

$$i_{R_L} = m \frac{V_R}{R_1} \quad (\text{P2.25.1})$$

Note. Note that this load current does not vary with the load resistance R_L .

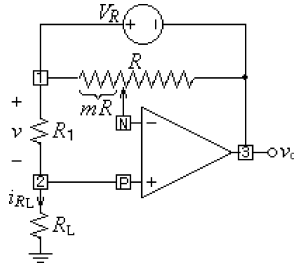


Figure P2.25 Load current controller

2.26 Voltage-to-Current Converter Using OP Amp – Howland Circuit

Consider the OP Amp circuit of Figure P2.26.1(a), which is known as the *Howland circuit*.

- (a) Noting that the circuit has a negative feedback, the virtual short principle and the voltage divider rule can be used to write

$$v_3 = \frac{R_1 + R_2}{R_1} v_- \stackrel{\text{virtual short}}{=} \frac{R_1 + R_2}{R_1} v_+ = \frac{R_1 + R_2}{R_1} v_o \quad (\text{P2.26.1})$$

Show that the output voltage v_o can be expressed in terms of the input voltage v_i as

$$v_o = \frac{1}{1 - R_2 R_3 / R_1 R_4 + R_3 / R_L} v_i \quad (\text{P2.26.2})$$

- (b) Referring to Figure P2.26.1(b), show that the output resistance is

$$R_o = \frac{R_4}{R_4 / R_3 - R_2 / R_1} \quad (\text{P2.26.3})$$

- (c) Find the condition on which the output resistance R_o is infinitely large. Also, show that if the condition is satisfied, the load current through R_L is

$$i_L = \frac{v_o}{R_L} = \frac{v_i}{R_3} \quad (\text{P2.26.4})$$

Note. This load current does not vary with the load resistance R_L and can be controlled by adjusting the input voltage v_i . That is why the circuit is called a voltage-to-current converter.

- (d) To see if the load current can really be controlled by adjusting the input voltage v_i regardless of the load resistance R_L , perform the PSpice simulation of the circuit for 0.05 s three times, where $R_1 = R_2 = 10 \text{ k}\Omega$ and $R_3 = R_4 = 5 \text{ k}\Omega$: once with $v_i(t) = \sin 2\pi 60t [\text{V}]$ and $R_L = 10 \text{ k}\Omega$, once

with $v_i(t) = \sin 2\pi 60t$ [V] and $R_L = 20 \text{ k}\Omega$, and once with $v_i(t) = 2 \sin 2\pi 60t$ [V] and $R_L = 10 \text{ k}\Omega$, referring to Figure P2.26.2(a).

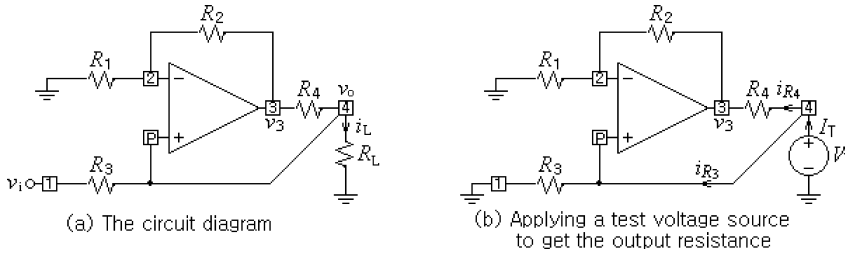


Figure P2.26.1 Voltage-to-current converter

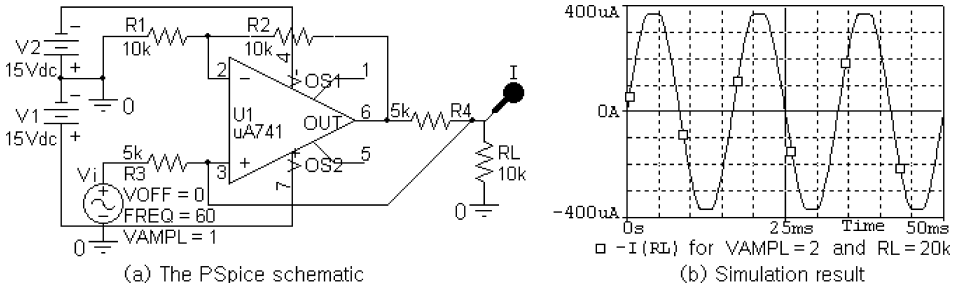


Figure P2.26.2 PSpice simulation of the voltage-to-current converter

- (e) Make use of Equation (P2.26.2) to get the load current for $v_i = 2 \text{ V}$ in the Howland circuit with $R_1 = R_2 = 10 \text{ k}\Omega$, $R_3 = R_4 = 5 \text{ k}\Omega$, and $R_L = 20 \text{ k}\Omega$. Perform the PSpice simulation with $v_i(t) = 2 \sin 2\pi 60t$ [V] and $R_L = 20 \text{ k}\Omega$, which will yield the load current waveform depicted in Figure P2.26.2(b). Does the simulation result agree with the analytical result? If not, discuss why $i_L(t)$ is distorted.

Hint. Find v_3 at the OP Amp output terminal for $v_i = 2 \text{ V}$ and check whether it exceeds the maximum output (saturation) voltage V_{om} .

2.27 Realization of Negative Resistance Using OP Amp

- (a) Show that the input resistance of the circuit in Figure P2.27(a) is

$$R_{in} = \frac{v_i}{i_i} = \frac{v_i}{(v_i - v_o)/R} = -\frac{R_1}{R_2}R \quad (\text{P2.27.1})$$

- (b) To see if this negative resistance can really be realized by the OP Amp circuit of Figure P2.27(a), perform the PSpice simulation of the circuit with $R_1 = 10 \text{ k}\Omega$, $R_2 = 20 \text{ k}\Omega$, $R = 2 \text{ k}\Omega$, and $v_i(t) = 4 \sin 0.2\pi t$ [V] for 10 s and have the input voltage and current waveforms plotted in the PSpice A/D (Probe) window, as depicted in Figure P2.27(b). Find the ratio of the input voltage to the input current obtained from the simulation and compare it with the resistance obtained from the analytical formula (P2.27.1).

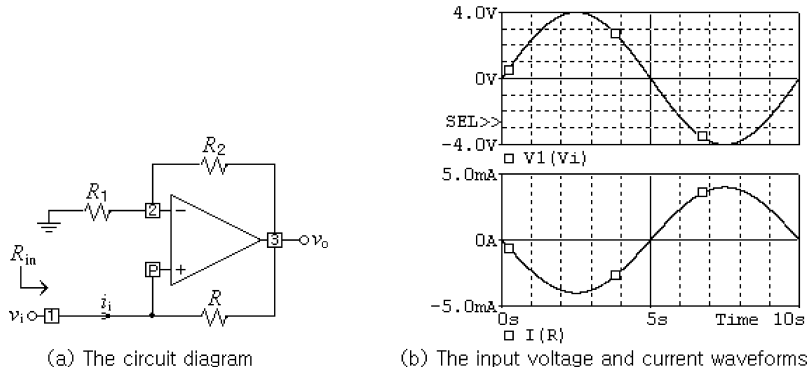


Figure P2.27 Negative resistance

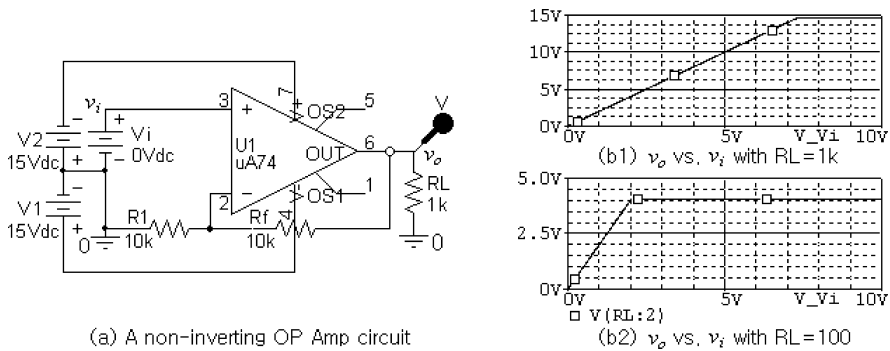


Figure P2.28.1

2.28 The Limits on Output Voltage and Current of OP Amp (Reference [H-1],

Referring to Figure P2.28.1(a), perform the PSpice simulation for the DC sweep analysis of a noninverting OP Amp circuit with $R_1 = 10k\Omega$ and $R_f = 10k\Omega$ to get the plot of v_o versus v_i in Figure P2.28.1(b1) (with $R_L = 1k\Omega$) and Figure P2.28.1(b2) (with $R_L = 100\Omega$).

- Consider Figure P2.28.1(b1). Does the voltage gain of the circuit agree with what you expect from Equation (2.26)? How can you explain the saturation of the output voltage at about 14 V?
Hint. Read the typical value of the output voltage swing of a $\mu A741$ OP Amp denoted by V_{om} or V_{opp} in the datasheet in Reference [W-7] or [W-8].
- Consider Figure P2.28.1(b2). How can you explain the saturation of the output voltage at about 4 V?
Hint. Put the Current Marker at the OP Amp output terminal to measure the output current of the OP Amp and read the maximum value of the (short-circuit) output current of a $\mu A741$ OP Amp denoted by I_{os} in the datasheet in Reference [W-7] or [W-8].
- Figure P2.28.2(a) shows a noninverting OP Amp circuit using an NPN-BJT as a current booster. Perform the PSpice simulation two times, once with the feedback from the OP Amp output and once with the feedback from the emitter terminal of the BJT (as depicted by the dotted line), to ensure that the corresponding input–output relationships are as depicted in Figures P2.28.2(b1) and (b2), respectively. Has the overload current problem been remedied by the BJT boosting its

base current, which is the OP Amp output current, by a factor of $(\beta + 1)$? Which feedback connection is better in realizing the voltage gain of $v_o/v_i = 2$?

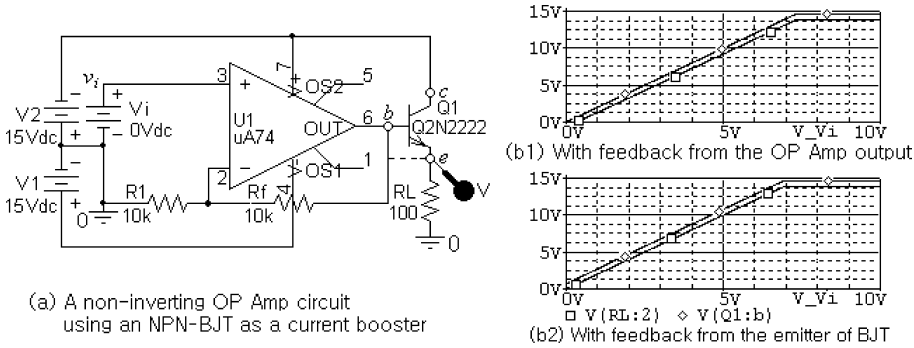


Figure P2.28.2

Note. The typical value of the DC current gain $\beta(h_{FE})$ is greater than 50 for the BJT 2N2222, as can be seen at the website (<http://www.alldatasheet.com/view.jsp?Searchword=2N2>).

2.29 Load Line Analysis of a Nonlinear Resistor Circuit

Consider the circuit of Figure P2.29(a), which contains a nonlinear resistor whose voltage–current characteristic curve is depicted in Figure P2.29(b).

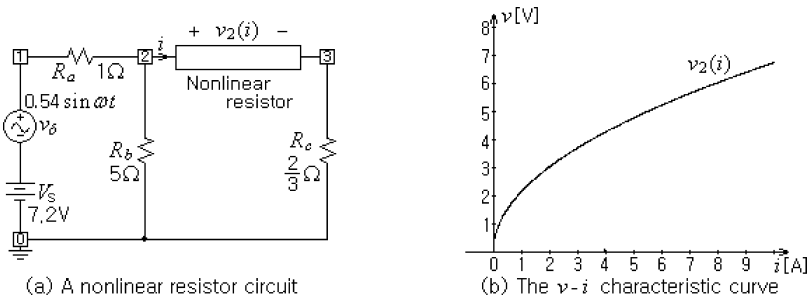


Figure P2.29

- Find the Thevenin equivalent of the circuit seen from nodes 2 and 3.
- Draw the load line on the characteristic curve of the nonlinear resistor to find the operating point (I_Q, V_Q) .
- Noting that $v_2(t) = 1.5\sqrt{2}i$, find the static and dynamic resistances of the nonlinear resistor at the operating point Q by using Equations (E2.23.7) and (E2.23.4).
- Using Equations (2.51) and (2.52), express the current through and the voltage across the nonlinear resistor in terms of the DC/AC components of the input voltage V_s and v_s .

3

First-Order Circuits

In contrast to the previous chapter, this and subsequent chapters deal with circuits having not only resistors but also electric energy storage elements such as inductors and capacitors that exhibit time-dependent properties and are thereby called dynamic elements. Unlike resistor-only circuits, circuits containing inductors and/or capacitors show a dynamic behavior where their voltages and currents vary with time, even to only DC sources. The equations for such dynamic circuits take the form of integro-differential equations and are solved using the Laplace transform method. Further, by transforming the dynamic circuits into the s -domain, many of the techniques developed for resistor networks can be used to analyze them. Readers who have never been exposed to the Laplace transform are strongly recommended to study it from Appendix A or by any other means.

This chapter deals only with first-order circuits, which are described by a first-order differential equation. A formula will be presented that can be used to obtain solutions of DC-excited first-order circuits even without setting up the circuit equations, rather than resorting to the Laplace transform. It will also be shown that some first-order circuits containing an OP Amp with positive feedback can be used to generate a periodic waveform. The 555 timer circuit introduced in one of the examples illustrates the practicability of a first-order OP Amp circuit.

3.1 Characteristics of Inductors and Capacitors

3.1.1 Inductor

As given by Equations (1.12a) and (1.12b), the voltage–current relationship of an inductor (depicted in Figure 3.1(a)) is described by the following equations:

$$v_L(t)[V] = N \frac{d\phi(t)}{dt} = L \frac{di_L(t)}{dt} [\text{H} \cdot \text{A/s}] \quad (3.1a)$$

$$i_L(t) = \frac{1}{L} \int_{-\infty}^t v_L(\tau) d\tau \quad (3.1b)$$

where L is the value of the inductance in henries (H).

Equation (3.1a) implies that if the inductor current changed abruptly enough to cause $di_L(t)/dt \rightarrow \infty$, the voltage across the inductor would become so high that other elements connected to the inductor might be damaged or at least an infinite amount of power is required; in other words, if the inductor current needs to be changed discontinuously, an (impulse-like) infinitely high voltage should be applied across the inductor, which is not usually permitted. This is called the *continuity rule of inductor currents* and is expressed by

$$i_L(t_0^-) = i_L(t_0) = i_L(t_0^+) \quad \text{for any instant } t_0 \quad (3.2)$$

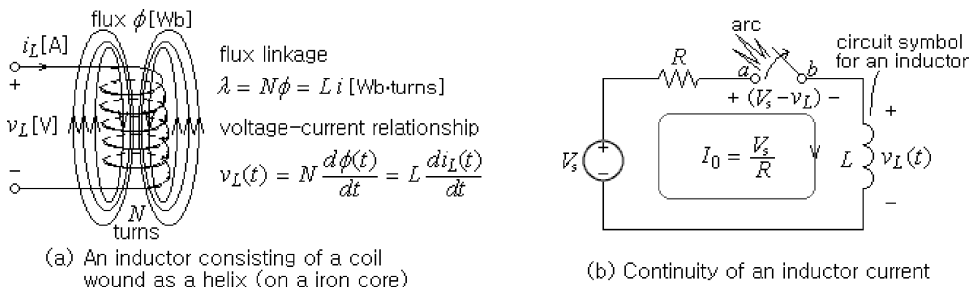


Figure 3.1 An inductor

While this holds in general for any instant t_0 , this rule is usually applied for some instant when input sources are connected/disconnected or the network structure is changed by switching operations. This is summarized in the following remark.

[Remark 3.1] Continuity of Inductor Currents – Memory/Inertia of an Inductor on Its Current

1. As long as the voltage across an inductor is not allowed to be infinitely high, its current is not supposed to have discontinuity at any instant. In this context, an inductor tends to keep its current, especially when the condition of the circuit on the input source and/or the structure is changed by switching operations; the current at such a switching instant is called the initial inductor current and should be considered for the transient analysis of circuits containing inductors.

Note. Inductors are called ‘chokes’ to reflect their habit to resist sudden changes in current.

2. In cases where the continuity rule is to be broken, say overwritten by the KCL applied to a node that is connected only to inductors and/or current sources (see Section 3.6.1), the voltage across the inductor may become infinitely high instantaneously, like an impulse. Such cases can be analyzed by applying the principle of *flux linkage conservation*, which states that an inductive circuit tends to conserve its total flux linkage:

$$\sum \lambda_i|_{t_0^-} = \sum \lambda_i|_{t_0} = \sum \lambda_i|_{t_0^+}; \quad \sum L_i I_i|_{t_0^-} = \sum L_i I_i|_{t_0} = \sum L_i I_i|_{t_0^+} \quad (3.3)$$

There are two implications of this continuity rule on inductor currents. One is a phenomenon called *spark arcing* where the inductor current initially continues to flow in the air gap across a switch on an inductive circuit when someone opens it. In order to understand this more concretely, consider the R – L circuit in Figure 3.1(b) where the switch is to be opened at $t = 0$ so that the inductor current is about to drop to zero from I_0 suddenly. Then the voltage–current relationship (3.1a) claims that this situation of $di_L/dt \rightarrow -\infty$ induces a negative high voltage ($v_L = L di_L/dt \rightarrow -\infty$) across the inductor and this will make the voltage between the contacts a and b of the switch high enough to exceed the insulating strength of the air gap (about 3 kV/mm), eventually providing an ionized path that appears to be a spark arcing. (You might feel like applauding the electrons for being so brave as to cross the arcing space and for being faithful to the continuity rule.) Through this arc across the switch contacts, the inductor current continues to flow until the switch contacts become distant enough to extinguish the arc. This is the way the continuity rule of the inductor current survives a jeopardy situation. Actually, switching inductive circuits is an important engineering problem, because arcing and voltage surges may cause equipment damage or at least corrosion of switch contacts. A free-wheeling diode is often connected in parallel with an inductor to provide a bypass for the strenuous current (see Section 3.6.4). It is interesting that such a

seemingly dangerous arcing phenomenon can be of practical use, as in the ignition system of an internal combustion engine (see Example 4.2).

The above-mentioned implication of the continuity rule on inductor currents is for the situation where an inductor having a nonzero initial current is to be blocked off from the source. Now consider the situation where an inductor having no initial current is to be connected to a source. This leads to another implication of the continuity rule on inductor currents, which says that an inductor having no initial current does not let any current flow through it, just like an open switch instantly after it is connected to a source. By contrast, an inductor allows any amount of current to flow through it with a zero induced voltage, just like a closed switch or a shorted element, if only the current is constant so that $di_L/dt \rightarrow 0$, which is also implied by Equation (3.1a). These two rules about the initial/final states of an inductor in DC circuits are restated in the following remark.

[Remark 3.2] The Initial/Final States of an Inductor

1. The instant an inductor having no initial current is connected to a source, it does not allow any current to flow through it, just like an open switch, no matter how high its voltage is. In other words, the initial state of an inductor having no initial current is an open element.
2. As long as the inductor current is constant (DC), the voltage across the inductor is zero as if it were shorted, no matter how large its current is. In other words, the final state of an inductor is a shorted element; i.e. an inductor acts as if it is shorted in the DC steady state.

Note. A circuit is said to be in the *DC steady state* if it has no time-varying quantities, voltages, or currents. Any circuit having no AC source is supposed to reach the DC steady state in a sufficiently long time.

3.1.2 Capacitor

As given by Equations (1.18a) and (1.18b), the voltage–current relationship of a capacitor (depicted in Figure 3.2(a)) is described by the following equations:

$$i_C(t)[A] = \frac{dq(t)}{dt} = C \frac{dv_C(t)}{dt} [F \cdot V/s] \quad (3.4a)$$

$$v_C(t) = \frac{1}{C} \int_{-\infty}^t i_C(\tau) d\tau \quad (3.4b)$$

where C is the value of the capacitance in farads (F).

Equation (3.4a) implies that if the capacitor voltage changed abruptly enough to cause $dv_C(t)/dt \rightarrow \infty$, the current across the capacitor would become so large that other elements connected

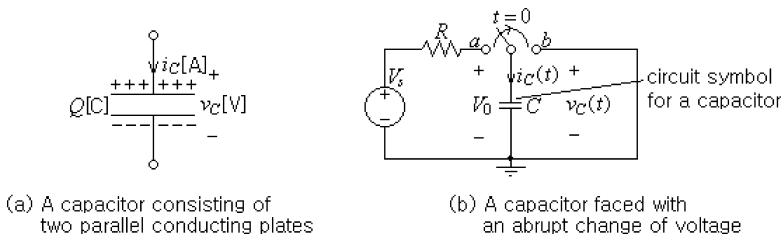


Figure 3.2 A capacitor

to the capacitor might be damaged or at least an infinite amount of power is required; in other words, if the capacitor voltage needs to be changed discontinuously, an (impulse-like) infinitely large current should be applied through the capacitor, which is not usually permitted. This is called the *continuity rule of capacitor voltages* and is expressed by

$$v_C(t_0^-) = v_C(t_0) = v_C(t_0^+) \quad \text{for any instant } t_0 \quad (3.5)$$

While this holds in general for any instant t_0 , this rule is usually applied for some instant when input sources are connected/disconnected or the network structure is changed by switching operations. This is summarized in the following remark.

[Remark 3.3] Continuity of Capacitor Voltages – Memory/Inertia of a Capacitor on Its Voltage

1. As long as the current through a capacitor is not allowed to be infinitely large, its voltage is not supposed to have discontinuity at any instant. In this context, a capacitor tends to keep its voltage, especially when the condition of the circuit on the input source and/or the structure is changed by switching operations; the voltage at such a switching instant is called the initial capacitor voltage and should be considered for the transient analysis of circuits containing capacitors.
2. In cases where the continuity rule is to be broken, say overwritten by the KVL applied to a loop that contains only capacitors and/or voltage sources (see Section 3.6.2), the current through the capacitor may become infinitely large instantaneously, like an impulse. Such cases can be analyzed by applying the principle of *charge conservation*, which states that a capacitive circuit tends to conserve its total charges:

$$\sum q_i|_{t_0^-} = \sum q_i|_{t_0} = \sum q_i|_{t_0^+}, \quad \sum C_i V_i|_{t_0^-} = \sum C_i V_i|_{t_0} = \sum C_i V_i|_{t_0^+} \quad (3.6)$$

What is the implication of this continuity rule on capacitor voltages for the situation where a capacitor having a nonzero initial voltage is to be shorted, as depicted in Fig. 3.2(b)? Fortunately it is not necessary to pay attention to such a singular case (that will be discussed in Section 3.6.2), because a physical capacitor has resistance in its dielectric and leads and so can be modeled as connected in series with a resistor. Now consider the situation where a capacitor having no initial voltage is to be connected to a source. The continuity rule on capacitor voltages says that a capacitor having no initial voltage tends to keep its voltage zero, just like a closed element instantly after it is connected to a source. By contrast, a capacitor allows no DC current to flow through it, just like an open switch, if only the voltage is constant so that $dv_C(t)/dt \rightarrow 0$, which is also implied by Equation (3.4a). These two rules about the initial/final states of a capacitor in DC circuits are restated as follows:

[Remark 3.4] The Initial/Final States of a Capacitor

1. Instantly after a capacitor having no initial voltage is connected to a source, it keeps the voltage zero, just like a closed switch, no matter how large its current is. In other words, the initial state of a capacitor having no initial voltage is a shorted element.
2. As long as the capacitor voltage is constant (DC), the current through the capacitor is zero as if it were opened, no matter how high its voltage is. In other words, the final state of a capacitor is an open element, i.e. a capacitor acts as if it is an open element in the DC steady state.

Table 3.1 Characteristics of inductors and capacitors

	Inductor with no initial current	Capacitor with no initial voltage
Voltage–current relationship	$v_L(t) = L \frac{di_L(t)}{dt}$ $i_L(t) = \frac{1}{L} \int_{-\infty}^t v_L(\tau) d\tau$ $= i_L(t_0) + \frac{1}{L} \int_{t_0}^t v_L(\tau) d\tau$	$i_C(t) = C \frac{dv_C(t)}{dt}$ $v_C(t) = \frac{1}{C} \int_{-\infty}^t i_C(\tau) d\tau$ $= v_C(t_0) + \frac{1}{C} \int_{t_0}^t i_C(\tau) d\tau$
Initial state	$i_L(t_0) = 0$ (open)	$v_C(t_0) = 0$ (short)
Final state (DC steady state)	$v_L(\infty) = 0$ (short)	$i_C(\infty) = 0$ (open)
Cannot change abruptly	$i_L(t_0^-) = i_L(t_0) = i_L(t_0^+)$	$v_C(t_0^-) = v_C(t_0) = v_C(t_0^+)$
For a singular circuit	Flux linkage conservation $\sum \lambda_i _{t_0^-} = \sum \lambda_i _{t_0} = \sum \lambda_i _{t_0^+}$ $\sum L_i i_i _{t_0^-} = \sum L_i i_i _{t_0} = \sum L_i i_i _{t_0^+}$	Charge conservation $\sum q_i _{t_0^-} = \sum q_i _{t_0} = \sum q_i _{t_0^+}$ $\sum C_i V_i _{t_0^-} = \sum C_i V_i _{t_0} = \sum C_i V_i _{t_0^+}$
Power	$p_L(t) = L i_L(t) \frac{di_L(t)}{dt}$	$p_C(t) = C v_C(t) \frac{dv_C(t)}{dt}$
Energy	$W_L(t) = \frac{1}{2} L i_L^2(t)$	$W_C(t) = \frac{1}{2} C v_C^2(t)$

Table 3.1 summarizes the characteristics of inductors and capacitors.

Note. Inductors are widely used in transformers, motors, relays, microphones and speakers, radios, TVs, power supplies, and high-frequency devices, but they are rarely used in modern electronics because their bulkiness and heaviness makes it difficult to be fabricated in an IC (integrated circuit).

Note. While resistors dissipate energy, inductors/capacitors never dissipate energy; they just store (borrow) energy and return the stored energy. However, this is true only for the ideal (mathematical) models and not for physical devices. Real-world inductors dissipate some energy because of the series-model (winding) resistance of the wire used for winding the coil and core losses in the magnetic core. Real-world capacitors also dissipate some energy because of the parasitic effect of the parallel-model leakage resistance.

Note. For electric materials, the voltage rating, or working voltage, should be considered, i.e. the maximum voltage that can safely be applied to them without causing any damage.

3.2 Series–Parallel Combination of Inductors/Capacitors

In this section we determine the equivalent inductance for series and parallel combinations of inductors/capacitors, which are analogous to the equivalent resistance/conductance.

3.2.1 Series–Parallel Combination of Inductors

Figure 3.3(a) shows a series combination of N inductors each of inductance L_n . Since all the inductors in series have the same current $i_1 = i_2 = \dots = i_N = i$, the voltage of each inductor can be written as

$$v_1(t) = L_1 \frac{di(t)}{dt}, \quad v_2(t) = L_2 \frac{di(t)}{dt}, \dots, \quad v_N(t) = L_N \frac{di(t)}{dt}$$

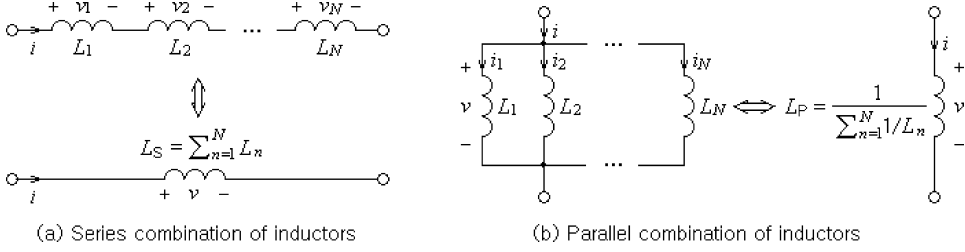


Figure 3.3 Series/parallel combination of inductors

and all the inductor voltages can be summed to obtain the overall voltage

$$v(t) = \sum_{n=1}^N v_n(t) = \sum_{n=1}^N L_n \frac{di(t)}{dt} = \left(\sum_{n=1}^N L_n \right) \frac{di(t)}{dt} = L_S \frac{di(t)}{dt}$$

This implies that the equivalent inductance of N inductors in series is the sum of all the individual inductances:

$$L_S = \sum_{n=1}^N L_n \quad (3.7)$$

Figure 3.3(b) shows a parallel combination of N inductors each of inductance L_n . Since all the inductors in parallel have the same voltage $v_1 = v_2 = \dots = v_N = v$, the current of each inductor can be written as

$$i_1(t) = \frac{1}{L_1} \int_{-\infty}^t v(\tau) d\tau, \quad i_2(t) = \frac{1}{L_2} \int_{-\infty}^t v(\tau) d\tau, \dots, \quad i_N(t) = \frac{1}{L_N} \int_{-\infty}^t v(\tau) d\tau$$

and all the inductor currents can be summed to obtain the overall current

$$i(t) = \sum_{n=1}^N i_n(t) = \sum_{n=1}^N \frac{1}{L_n} \int_{-\infty}^t v(\tau) d\tau = \left(\sum_{n=1}^N \frac{1}{L_n} \right) \int_{-\infty}^t v(\tau) d\tau = \frac{1}{L_P} \int_{-\infty}^t v(\tau) d\tau$$

This implies that the equivalent inverse inductance of N inductors in parallel is the sum of all the individual inverse inductances:

$$\frac{1}{L_P} = \sum_{n=1}^N \frac{1}{L_n} \quad (3.8)$$

3.2.2 Series–Parallel Combination of Capacitors

Figure 3.4(a) shows a series combination of N capacitors each of capacitance C_n . Since all the capacitors in series have the same current $i_1 = i_2 = \dots = i_N = i$, the voltage of each capacitor can be written as

$$v_1(t) = \frac{1}{C_1} \int_{-\infty}^t i(\tau) d\tau, \quad v_2(t) = \frac{1}{C_2} \int_{-\infty}^t i(\tau) d\tau, \dots, \quad v_N(t) = \frac{1}{C_N} \int_{-\infty}^t i(\tau) d\tau$$

and all the capacitor voltages can be summed to obtain the overall voltage

$$v(t) = \sum_{n=1}^N v_n(t) = \sum_{n=1}^N \frac{1}{C_n} \int_{-\infty}^t i(\tau) d\tau = \left(\sum_{n=1}^N \frac{1}{C_n} \right) \int_{-\infty}^t i(\tau) d\tau = \frac{1}{C_S} \int_{-\infty}^t i(\tau) d\tau$$

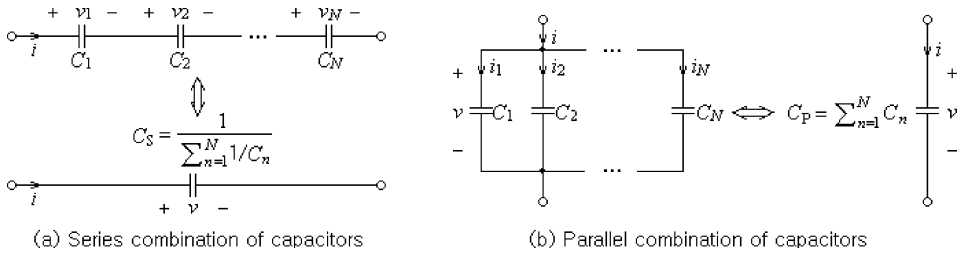


Figure 3.4 Series/parallel combination of capacitors

This implies that the equivalent inverse capacitance of N capacitors in series is the sum of all the individual inverse capacitances:

$$\frac{1}{C_S} = \sum_{n=1}^N \frac{1}{C_n} \quad (3.9)$$

Figure 3.4(b) shows a parallel combination of N capacitors each of capacitance C_n . Since all the capacitors in parallel have the same voltage $v_1 = v_2 = \dots = v_N = v$, the current of each capacitor can be written as

$$i_1(t) = C_1 \frac{dv(t)}{dt}, \quad i_2(t) = C_2 \frac{dv(t)}{dt}, \quad \dots, \quad i_N(t) = C_N \frac{dv(t)}{dt}$$

and all the capacitor currents can be summed to obtain the overall current

$$i(t) = \sum_{n=1}^N i_n(t) = \sum_{n=1}^N C_n \frac{dv(t)}{dt} = \left(\sum_{n=1}^N C_n \right) \frac{dv(t)}{dt} = C_P \frac{dv(t)}{dt}$$

This implies that the equivalent capacitance of N capacitors in parallel is the sum of all the individual capacitances:

$$C_P = \sum_{n=1}^N C_n \quad (3.10)$$

3.3 Circuit Analysis Using the Laplace Transform

In this section we discuss how to use the Laplace transform for the description as well as the analysis of circuit systems. The mesh/node equation is first set up in the form of an integro-differential equation which is then solved using the Laplace transform. Later it is found to be more efficient to bypass the time (t)-domain circuit equation and transform the circuit directly to set up the s -domain circuit equation. This technique allows the circuits containing such dynamic elements as inductors/capacitors to be dealt with as if they were resistor circuits discussed in Chapter 2.

Readers who are not familiar with the Laplace transform need not worry as a review of Appendix A prior studying this section will be sufficient in order to proceed. As illustrated in Figure 3.5, the procedure of applying the Laplace transform to solve a differential equation consists of the following three steps:

1. Take the Laplace transform of the differential equation by using the differentiation/integration properties of the Laplace transform.
2. Solve the transformed algebraic equation to get the s -domain solution.
3. Take the inverse Laplace transform of the transformed solution to get the t -domain solution.

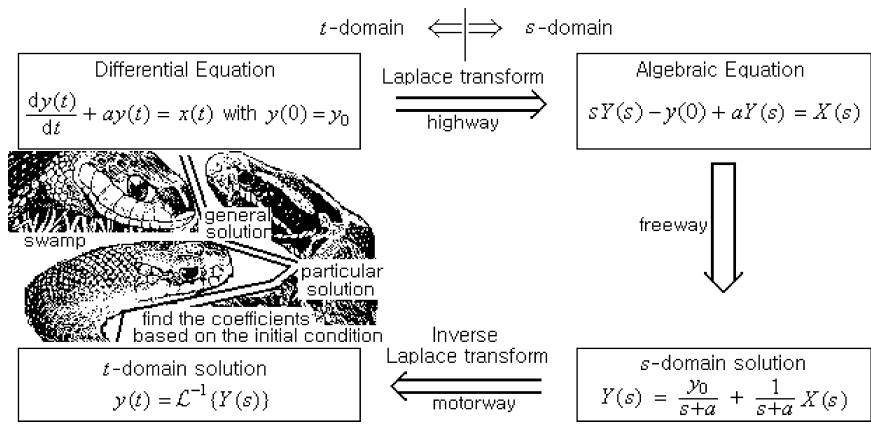


Figure 3.5 The Laplace transform method for solving differential equations

The Laplace transforms of several time functions and some properties of the Laplace transform are listed in Tables 3.2 and 3.3 (or Tables A.1 and A.2 in Appendix A), respectively.

3.3.1 The Laplace Transform for a First-Order Differential Equation

Let us use the Laplace transform to solve the following first-order differential equation:

$$\frac{dy(t)}{dt} + ay(t) = x(t) \quad \text{with the initial condition } y(0) = y_0 \quad (3.11)$$

Taking the Laplace transform of both sides and using the differentiation property (Table 3.3(5)) of the Laplace transform yields

$$\begin{aligned} sY(s) - y(0) + aY(s) &= X(s) \\ (s + a)Y(s) &= y_0 + X(s) \end{aligned}$$

Table 3.2 Laplace transforms of basic functions

$x(t)$	$X(s)$
(1) Unit impulse function $\delta(t)$	1
(2) Unit step function $u_s(t)$	$\frac{1}{s}$
(3) Exponential function $e^{-at}u_s(t)$	$\frac{1}{s+a}$
(4) $\cos \omega t u_s(t)$	$\frac{s}{s^2 + \omega^2}$
(5) $\sin \omega t u_s(t)$	$\frac{\omega}{s^2 + \omega^2}$
(6) $e^{-at} \cos \omega t u_s(t)$	$\frac{s+a}{(s+a)^2 + \omega^2}$
(7) $e^{-at} \sin \omega t u_s(t)$	$\frac{\omega}{(s+a)^2 + \omega^2}$

Table 3.3 Properties of the Laplace transform

Definition	$X(s) = \mathcal{L}\{x(t)\} = \int_0^\infty x(t) e^{-st} dt; \quad x(t) \leftrightarrow X(s)$
(1) Linearity	$\alpha x(t) + \beta y(t) \leftrightarrow \alpha X(s) + \beta Y(s)$
(2) Time shifting	$x(t - t_1)u_s(t - t_1), \quad t_1 > 0 \leftrightarrow e^{-st_1} X(s) \text{ for } x(t) = 0 \quad \forall t < 0$
(3) Frequency shifting	$e^{st_1} x(t) \leftrightarrow X(s - s_1)$
(4) Real convolution	$g(t) * X(t) \leftrightarrow G(s) X(s)$
(5) Time derivative	$X'(t) = \frac{d}{dt} x(t) \leftrightarrow sX(s) - x(0)$
(6) Time integral	$\int_{-\infty}^t x(\tau) d\tau \leftrightarrow \frac{1}{s} X(s) + \frac{1}{s} \int_{-\infty}^0 x(\tau) d\tau$
(7) Initial value theorem	$x(0) = \lim_{s \rightarrow \infty} sX(s)$
(8) Final value theorem	$x(\infty) = \lim_{s \rightarrow 0} sX(s)$

This algebraic equation is solved to get the s -domain solution

$$Y(s) = \frac{y_0}{s+a} + \frac{1}{s+a} X(s) \quad (3.12)$$

and take its inverse Laplace transform to obtain the t -domain solution

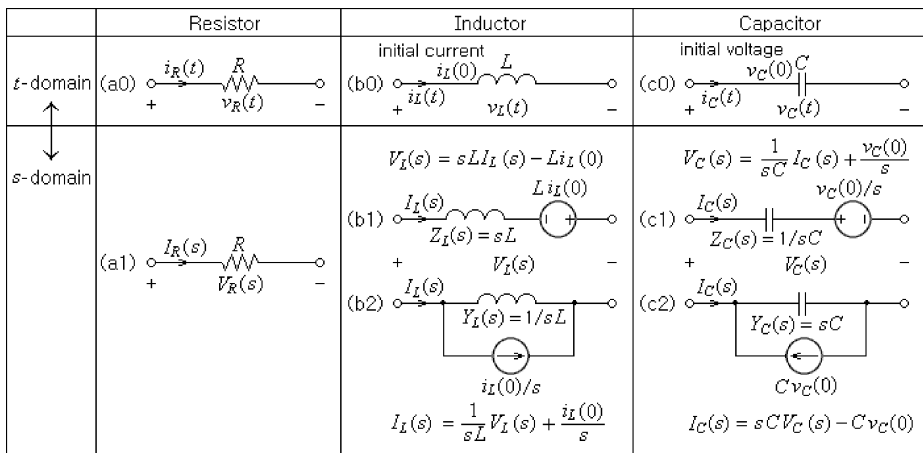
$$y(t) = \mathcal{L}^{-1}\{Y(s)\} \quad (3.13)$$

3.3.2 Transformed Equivalent Circuits for R , L , and C

In order to transform a circuit into the s -domain, we *transformed (s -domain) equivalent* circuits for a resistor, an inductor, and a capacitor, as shown in Figure 3.6. These circuits are obtained by taking the Laplace transform of their voltage–current relationships as follows:

$$\text{Resistor: } v_R(t) = R i_R(t) \rightarrow V_R(s) = R I_R(s) \quad (3.14a)$$

$$Z_R(s) = \frac{V_R(s)}{I_R(s)} = R [\Omega], \quad Y_R(s) = \frac{I_R(s)}{V_R(s)} = G [S] \quad (3.14b)$$

**Figure 3.6** The transformed (s -domain) equivalent circuits of R , L , and C

Inductor: $v_L(t) = L \frac{di_L(t)}{dt} \rightarrow V_L(s) \stackrel{\text{Table 3.3(5)}}{=} L(sI_L(s) - i_L(0)) = sLI_L(s) - Li_L(0)$ (3.15a)

or, equivalently, $I_L(s) = \frac{1}{sL} V_L(s) + \frac{i_L(0)}{s}$ (3.15b)

With no initial current ($i_L(0) = 0$), the ratios of the transformed voltage to the transformed current and its inverse are

$$Z_L(s) = \frac{V_L(s)}{I_L(s)} = sL [\Omega], \quad Y_L(s) = \frac{I_L(s)}{V_L(s)} = \frac{1}{sL} [\text{S}] \quad (3.15c)$$

These are called the *impedance* ('generalized resistance') and *admittance* ('generalized conductance') of an inductor with inductance $L[\text{H}]$, and are measured in ohms (Ω) and siemens (S), respectively.

Capacitor: $i_C(t) = C \frac{dv_C(t)}{dt} \rightarrow I_C(s) \stackrel{\text{Table 3.3(5)}}{=} C[sV_C(s) - v_C(0)] = sCV_C(s) - Cv_C(0)$ (3.16a)

or, equivalently, $V_C(s) = \frac{1}{sC} I_C(s) + \frac{v_C(0)}{s}$ (3.16b)

With no initial voltage ($v_C(0) = 0$), the ratio of the transformed voltage to the transformed current and its inverse is

$$Z_C(s) = \frac{V_C(s)}{I_C(s)} = \frac{1}{sC} [\Omega], \quad Y_C(s) = \frac{I_C(s)}{V_C(s)} = sC [\text{S}] \quad (3.16c)$$

These are called the *impedance* and the *admittance* of a capacitor with capacitance $C[\text{F}]$.

[Remark 3.5] Some Observations about Inductors/Capacitors

1. Once inductors/capacitors are replaced by their s -domain equivalents, shown in Figure 3.6, they can be treated just like resistors and therefore the circuits containing inductors/capacitors can be simplified and solved by the simplification techniques and analysis methods introduced for resistive circuits in Chapter 2.
2. Note that it is better to use the s -domain equivalents representing the initial conditions by voltage sources (Figures 3.6(b1) and (c1)) or current sources (Figures 3.6(b2) and (c2)), depending on which method is going to be used for the circuit analysis, mesh analysis, or node analysis, because the mesh/node analysis prefers to have voltage/current sources, respectively.
3. The s -domain description of inductors/capacitors in terms of impedance or admittance leads to the concept of the transfer function or network function of a circuit, which is the ratio of the transformed output to the transformed input. This will be discussed in Section 4.4.
4. As will be mentioned in Section 4.4.2, inductors and capacitors are dual in the sense that the description of their behaviors is valid if L is interchanged with C , v with i , and series with parallel.

3.4 Analysis of First-Order Circuits

In this section it will be seen how to solve first-order circuits that have a single inductor or a single capacitor (possibly with two or more ones in series/parallel combined into their equivalent) and, as such, are described by a first-order ordinary differential equation.

3.4.1 DC-Excited RL Circuits

Consider the RL circuit in Figure 3.7(a), where the transfer switch has been connected to position b for a long time and then, at $t = 0$, is flipped to position a . Note that the switch is a make-before-break type,

which holds the connection to one position till it makes a connection to another position, thus making sure that the inductor current will never be interrupted abruptly by flipping the switch.

First, let us try the mesh analysis introduced in Section 2.5. In order to do so, KVL is applied to the (single) mesh to write the mesh equation in the mesh current $i(t)$ as

$$v_R(t) + v_L(t) = Ri(t) + L \frac{di(t)}{dt} = v_i(t) = V_i u_s(t) \quad (3.17)$$

where the unit step function $u_s(t)$ is multiplied by V_i to specify that the DC voltage source V_i starts to be applied at the switching instant $t = 0$. Taking the Laplace transform of both sides and using Table 3.3(5) and Table 3.2(2) yield

$$RI(s) + L(sI(s) - i(0)) = V_i(s) = \frac{V_i}{s} \quad (3.18a)$$

$$I(s) = \frac{V_i/s + Li(0)}{R + sL} = \frac{V_i/L}{s(s + R/L)} + \frac{I_0}{s + R/L} \quad (3.18b)$$

The partial fraction expansion and the inverse Laplace transform of this s -domain solution can be taken to obtain the t -domain solution

$$I(s) = \frac{V_i}{R} \left(\frac{1}{s} - \frac{1}{s + R/L} \right) + \frac{I_0}{s + R/L}$$

$$i(t) = \mathcal{L}^{-1}\{I(s)\} \stackrel{\text{Table 3.2(2),(3)}}{=} \left[\frac{V_i}{R} \left(1 - e^{-Rt/L} \right) + I_0 e^{-Rt/L} \right] u_s(t) \quad (3.19)$$

A better way of solving the RL circuit via mesh analysis is to make use of the s -domain equivalent (in Figure 3.6(b1)) to transform the circuit as depicted in Figure 3.7(b). Then, bypassing the differential equation, KVL can be applied to the (single) mesh to get directly the mesh equation (3.18a) in the s -domain mesh current $I(s)$, which yields the same result.

There are two observations. First, substituting $t = 0$ into this solution (3.19) yields

$$i(0) = I_0 \quad (3.20)$$

which is equal to the initial inductor current $i(0^-) = I_0$, as claimed by the continuity rule (3.2). Second, substituting $t = \infty$ into the solution (3.19) yields

$$i(\infty) = \frac{V_i}{R} \quad (3.21)$$

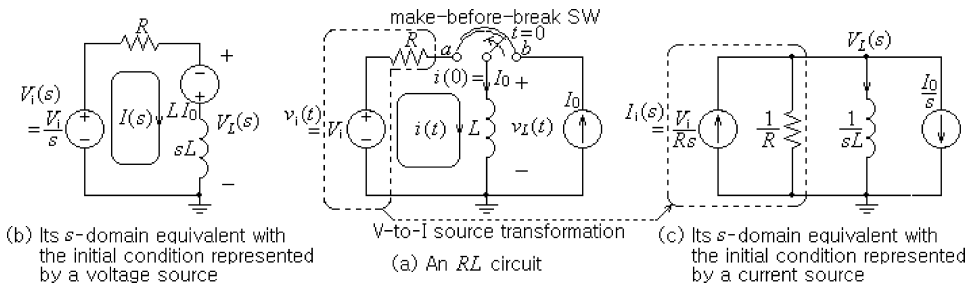


Figure 3.7 An RL circuit and its transformed (s -domain) equivalent circuits

which is equal to the mesh current with the inductor shorted, as stated for the final (DC steady) state of an inductor in Remark 3.2(2).

Now let us try using the node analysis introduced in Section 2.4. In order to do so, KCL is applied to the upper node of the inductor to write the node equation in the node voltage $v_L(t)$ as

$$i_R(t) + i_L(t) = \frac{v_L(t) - v_i(t)}{R} + \frac{1}{L} \int_{-\infty}^t v_L(\tau) d\tau = 0 \quad (3.22)$$

Taking the Laplace transform of both sides and using Table 3.3(6) yields

$$\frac{V_L(s) - V_i(s)}{R} + \frac{1}{L} \left[\frac{1}{s} V_L(s) + \frac{1}{s} \int_{-\infty}^0 v_L(\tau) d\tau \right] = 0 \quad (3.23a)$$

$$\left(\frac{1}{R} + \frac{1}{sL} \right) V_L(s) = \frac{V_i/s}{R} - \frac{1}{s} I_0 \quad \left(\because \frac{1}{L} \int_{-\infty}^0 v_L(\tau) d\tau = i_L(0) = I_0 \text{ and } V_i(s) = \frac{V_i}{s} \right) \quad (3.23b)$$

$$V_L(s) = \frac{LV_i - RL I_0}{R + sL} = \frac{V_i - RI_0}{s + R/L} \quad (3.23c)$$

The inverse Laplace transform of this s -domain solution can be taken to obtain the t -domain solution

$$v_L(t) \stackrel{\text{Table 3.2(3)}}{=} (V_i - RI_0) e^{-Rt/L} u_s(t) \quad (3.24)$$

A better way of solving the RL circuit via the node analysis is to make use of the s -domain equivalent (in Figure 3.6(b2)) to transform the circuit as depicted in Figure 3.7(c). Then, bypassing the differential equation, KCL can be applied to the top node to get directly the node equation (3.23b) in the s -domain node voltage $V_L(s)$, which yields the same result.

There are three observations. First, substituting $t = 0$ into this solution (3.24) yields

$$v_L(0) = V_i - RI_0 \quad (3.25)$$

which is equal to what is obtained by subtracting the voltage drop RI_0 across R from the node voltage V_i at the left node of R . This reflects that the initial inductor current at the switching instant $t = 0$ is equal to $i(0^-) = I_0$, as claimed by the continuity rule (3.2). Second, substituting $t = \infty$ into the solution (3.24) yields

$$v_L(\infty) = 0 \quad (3.26)$$

which reflects the fact that the inductor acts like a short circuit, as stated for the final (DC steady) state of an inductor in Remark 3.2(2). Third, the inductor current obtained by substituting Equation (3.24) into Equation (3.1b) agrees with Equation (3.19), which was obtained from the mesh analysis:

$$\begin{aligned} i_L(t) &\stackrel{(3.1b)}{=} \frac{1}{L} \int_{-\infty}^t v_L(\tau) d\tau = \frac{1}{L} \int_{-\infty}^0 v_L(\tau) d\tau + \frac{1}{L} \int_0^t v_L(\tau) d\tau \\ &\stackrel{(3.24)}{=} i_L(0) - \frac{V_i - RI_0}{L(R/L)} e^{-Rt/L} \Big|_0^t = I_0 + \frac{V_i - RI_0}{R} (1 - e^{-Rt/L}) \\ &\stackrel{(F.33)}{=} \frac{V_i}{R} (1 - e^{-Rt/L}) + I_0 e^{-Rt/L} \stackrel{(3.19)}{=} i(t) \end{aligned} \quad (3.27)$$

3.4.2 DC-Excited RC Circuits

Consider the RC circuit in Figure 3.8(a), where the transfer switch is closed at $t = 0$ when the capacitor has the initial voltage V_0 .

First, in order to try the mesh analysis, KVL is applied to the (single) mesh to write the mesh equation in the mesh current $i(t)$ as

$$v_R(t) + v_C(t) = R i(t) + \frac{1}{C} \int_{-\infty}^t i(\tau) d\tau = v_i(t) = V_i u_s(t) \quad (3.28)$$

Taking the Laplace transform of both sides and using Table 3.3(6) yields

$$R I(s) + \frac{1}{C} \left[\frac{1}{s} I(s) + \frac{1}{s} \int_{-\infty}^0 i(\tau) d\tau \right] = \frac{V_i}{s} \quad (3.29a)$$

$$\left(R + \frac{1}{sC} \right) I(s) = \frac{V_i}{s} - \frac{1}{s} V_0 \quad \left(\because \frac{1}{C} \int_{-\infty}^0 i(\tau) d\tau = v_C(0) = V_0 \right) \quad (3.29b)$$

$$I(s) = \frac{V_i/s - V_0/s}{R + 1/(sC)} = \frac{(V_i - V_0)/R}{s + 1/(RC)} \quad (3.29c)$$

The inverse Laplace transform of this s -domain solution can be taken to obtain the t -domain solution

$$i(t) \stackrel{\text{Table 3.2(3)}}{=} \frac{V_i - V_0}{R} e^{-t/(RC)} u_s(t) \quad (3.30)$$

A better way of solving the RC circuit via the mesh analysis is to make use of the s -domain equivalent (in Figure 3.6(c1)) to transform the circuit as depicted in Figure 3.8(b). Then, bypassing the differential equation, KVL can be applied to the (single) mesh to write directly the mesh equation (3.29b) in the s -domain mesh current $I(s)$ and solve it to get the same result.

There are two observations. First, substituting $t = 0$ into this solution (3.30) yields

$$i(0) = \frac{V_i - V_0}{R} \quad (3.31)$$

which reflects that the initial capacitor voltage at the switching instant $t = 0$ is equal to $v_C(0^-) = V_0$, as claimed by the continuity rule (3.5). Second, substituting $t = \infty$ into the solution (3.30) yields

$$i(\infty) = 0 \quad (3.32)$$

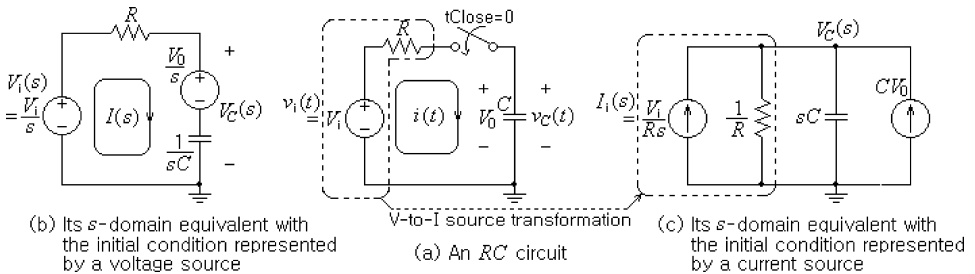


Figure 3.8 An RC circuit and its transformed (s -domain) equivalent circuits

which reflects the fact that the capacitor acts like an open circuit, as stated for the final (DC steady) state of a capacitor in Remark 3.4(2).

Now, to try using the node analysis, KCL is applied to the upper node of the capacitor to write the node equation in the node voltage $v_C(t)$ as

$$i_R(t) + i_C(t) = \frac{v_C(t) - v_i(t)}{R} + C \frac{dv_C(t)}{dt} = 0 \quad (3.33)$$

Taking the Laplace transform of both sides and using Table 3.3(5) yields

$$\left(\frac{1}{R} + sC\right)V_C(s) = C v_C(0) + \frac{V_i(s)}{R} = C v_C(0) + \frac{V_i}{Rs} \quad (3.34a)$$

$$\begin{aligned} V_C(s) &= \frac{C V_0 + V_i/Rs}{1/R + sC} = \frac{V_0}{s + 1/(RC)} + \frac{V_i/(RC)}{s[s + 1/(RC)]} \\ &= \frac{V_0}{s + 1/(RC)} + \frac{V_i}{s} - \frac{V_i}{s + 1/(RC)} \end{aligned} \quad (3.34b)$$

The inverse Laplace transform of this s -domain solution can be taken to obtain the t -domain solution

$$v_C(t) \stackrel{\text{Table 3.2(2),(3)}}{=} \left[V_i \left(1 - e^{-t/(RC)}\right) + V_0 e^{-t/(RC)} \right] u_s(t) \quad (3.35)$$

A better way of solving the RC circuit via the node analysis is to make use of the s -domain equivalent (in Figure 3.6(c2)) to transform the circuit as depicted in Figure 3.8(c). Then, bypassing the differential equation, KCL can be applied to the top node to write directly the node equation (3.34a) in the s -domain node voltage $V_C(s)$, which yields the same result.

There are three observations. First, substituting $t = 0$ into this solution (3.35) yields

$$v_C(0) = V_0 \quad (3.36)$$

which is equal to the initial capacitor voltage just before the switching instant $t = 0$, as claimed by the continuity rule (3.5). Second, substituting $t = \infty$ into the solution (3.35) yields

$$v_C(\infty) = V_i \equiv V_i - R i(\infty) \quad \text{with} \quad i(\infty) = 0 \quad (3.37)$$

which reflects the fact that the capacitor acts like an open circuit, as stated for the final (DC steady) state of a capacitor in Remark 3.4(2). Third, the capacitor current obtained by substituting Equation (3.35) into Equation (3.4a) agrees with Equation (3.30), which was obtained from the mesh analysis:

$$i_C(t) \stackrel{(3.4a)}{=} C \frac{dv_C(t)}{dt} \stackrel{(3.35),(F.27)}{=} -C \frac{V_0 - V_i}{RC} e^{-t/(RC)} = \frac{V_i - V_0}{R} e^{-t/(RC)} u_s(t) \quad (3.38)$$

[Remark 3.6] Natural/Forced Responses and Transient/Steady State Responses

1. The response of a circuit can be decomposed into two components, the *natural* (zero-input) response and the *forced* (zero-state) response, depending on its cause, the initial condition, or

the input source. For instance, the mesh current of the RL circuit (Figure 3.7(a)) and the node voltage of the RC circuit (Figure 3.8(a)) can be decomposed as

$$\begin{aligned} & \text{forced response} + \text{natural response} \\ i(t) & \stackrel{(3.19)}{=} \frac{V_i}{R} \left(1 - e^{-Rt/L} \right) + I_0 e^{-Rt/L} \\ v(t) & \stackrel{(3.35)}{=} V_i \left(1 - e^{-t/(RC)} \right) + V_0 e^{-t/(RC)} \end{aligned}$$

2. From another point of view, it can be decomposed into two components, the transient response and the steady state response, depending on its resulting steadiness. The component that dies out (decaying exponentially) as time goes by is called the *transient response* and the component that survives time like a constant or a sinusoid is called the *steady state response*. For instance, the mesh current of the RL circuit and the node voltage of the RC circuit can be decomposed as

$$\begin{aligned} & \text{transient response} + \text{steady state response} \\ i(t) & \stackrel{(3.19)}{=} \left(I_0 - \frac{V_i}{R} \right) e^{-Rt/L} + \frac{V_i}{R} \\ v(t) & \stackrel{(3.35)}{=} (V_0 - V_i) e^{-t/(RC)} + V_i \end{aligned}$$

3.4.3 Time-Constant and Natural Responses of First-Order Circuits

As a measure of how fast the natural response of a circuit system is, the (natural) *time constant* is defined as the time required for the natural response to decay by a factor of $e^{-1} = 0.368$ (36.8 %). In view of the fact that the natural response is described by an exponential function $x(0)e^{-at}$, the time constant is $T = 1/a$, i.e. the reciprocal of the absolute multiplicative coefficient (which is multiplied by t) in the exponent. It can be described graphically by the intersection of the tangent at $t = 0$ of the natural response with the time axis since the slope of the tangent line is the time derivative $-a x(0)e^{-at}|_{t=0} = -a x(0) = -x(0)/T$.

Figure 3.9 shows the natural/forced responses of the RL circuit and the RC circuit, where the values of R , L , and C are assumed to be 1Ω , $1H$, and $1F$, respectively, the initial inductor current is $I_0 = 1A$, and the initial capacitor voltage is $V_0 = 1V$. As can be seen from Equations (3.24) and (3.27), which have the exponential term $e^{-Rt/L}$, the time constant of the RL circuit is $L/R = 1s$. As can be seen from Equations (3.35) and (3.38), which have the exponential term $e^{-t/(RC)}$, the time constant of the RC circuit is $RC = 1s$. In two and five time constants, the natural response decays to about 13.5 % (e^{-2}) and 0.7 % (e^{-5}) of its initial value, implying that the response proceeds up to 86.5 % and 99.3 % toward the final stage, i.e. the steady state.

[Remark 3.7] Time Constant and Natural Response

1. As illustrated in Section 3.4.2, all circuit variables involved in the natural response, voltages or currents, have an exponential behavior characterized by the same exponential term. Therefore, the *time constant*, which is supposed to be the reciprocal of the absolute multiplicative coefficient in the exponent, is the characteristic of the whole circuit system rather than individual circuit variables. In this context, the reciprocal of the time constant is referred to as the *natural frequency* of the circuit system, borrowing the concept of frequency that describes how fast a signal is oscillating.
2. The time constant of an RL circuit is L/R and that of an RC circuit is RC . What are their units? It can be seen from the voltage–current relationships (1.6a)/(3.1a)/(3.4a) of resistors/inductors/capacitors that the units of resistance R , inductance L , and capacitance C are $\Omega = V/A$, $H = V(A/s)$ and

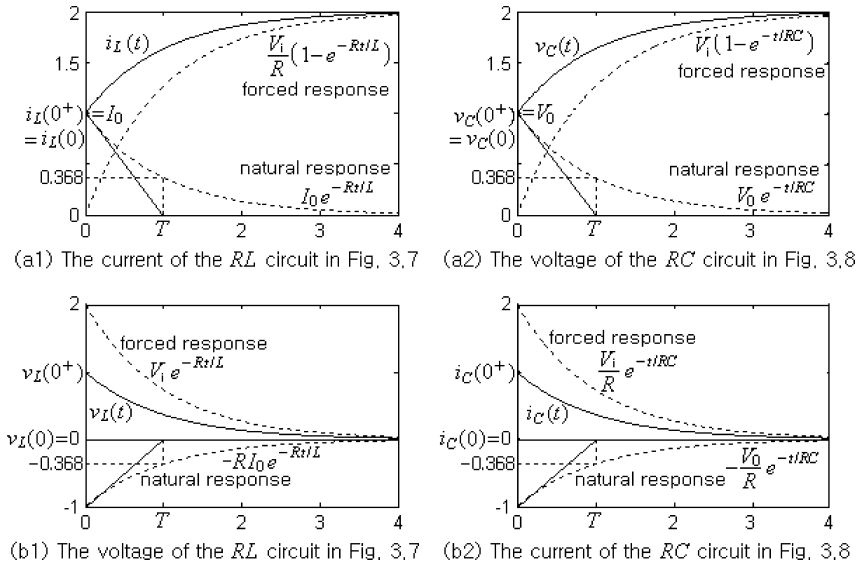


Figure 3.9 The forced/natural responses and the time constants of first-order circuits

$F = A/(V/s)$, respectively. Therefore the units of L/R and RC commonly turn out to be seconds (s), i.e. the unit of time, meeting the expectation:

$$\text{The unit of } \frac{L}{R}: \frac{H}{\Omega} = \frac{V/(A/s)}{V/A} = s, \quad \text{The unit of } RC: \Omega \cdot F = (V/A) \cdot A/(V/s) = s$$

How can we find the time constant of a circuit having multiple inductors/capacitors that can be combined in series and/or parallel to make a single equivalent inductor/capacitor? The following remark answers this question.

[Remark 3.8] Time Constant of a Virtual First-Order Circuit

Even if some circuits have multiple inductors or capacitors together with multiple resistors, they are virtually first-order circuits as long as the inductors/capacitors can be combined in series and/or parallel to make a single equivalent inductor/capacitor. The procedure for obtaining the time constant of such circuits is as follows:

1. Remove all the independent sources by short-circuiting/open-circuiting the voltage/current sources.
2. Combine the inductors/capacitors in series and/or parallel to get a single equivalent inductance(L_{eq})/capacitance(C_{eq}).
3. Find the Thevenin equivalent resistance R_{eq} seen from the terminals of the equivalent inductor/capacitor with the inductor/capacitor opened.
4. Then the resulting time constant is $L_{eq}/R_{eq}[s]$ for RL circuits and $R_{eq}C_{eq}[s]$ for RC circuits.

From Equations (3.19), (3.24), (3.30), and (3.35), a formula can be deduced that can be used as a short-cut to find the responses of first-order circuits excited by DC sources. This formula is of practical use for DC solutions of first-order circuits because it yields directly the time-domain solution without involving

anything like differential equations or the Laplace transform when only the time constant of the circuit and the initial/final values of the unknown variable are known. The following remark introduces the formula.

[Remark 3.9] A Formula for Responses of DC-Excited First-Order Circuits

1. The complete or total time response of a first-order circuit excited by DC sources can be found by using the following formula:

$$x(t) = [x(t_0^+) - x(\infty)]e^{-(t-t_0)/T} + x(\infty) \quad \text{with } T = \text{time constant} \quad (3.39)$$

The validity of this formula can be verified by seeing that it holds for both $t = t_0^+$ and $t = \infty$. Here, t_0^+ does not have to be a particular instant, but it often means the time just after switching. While it can be identified with t_0 for inductor currents and capacitor voltages subject to the continuity rule, it should not be equated with t_0 for inductor voltages and capacitor currents not obeying the continuity rule.

2. The initial/final values required for using the above formula can be obtained as follows:
 - (a) In order to find the *initial value* $x(t_0^+)$, all the inductors/capacitors should be replaced by the current/voltage sources having the value of their initial currents/voltages; especially, the inductors/capacitors with no initial condition should be opened/shorted.
 - (b) In order to find the (DC) *final* or *state-state value* $x(\infty)$, all the inductors/capacitors should be shorted/opened as their final states.

As far as DC-excited first-order circuits are concerned, Equation (3.39) will give the complete solution with the least effort. The next two examples will illustrate the use of the formula (3.39).

(Example 3.1) Time Constant and Solution of a DC-Excited First-Order Circuit

- (a) Let us use the formula (3.39) to find the inductor current/voltage for the RL circuit of Figure 3.7(a). Note that the time constant is $T = L/R$ and as for the inductor current,

- the initial value is $i_L(0^+) \stackrel{\text{continuity}}{=} i_L(0) = I_0$ and
- the final value is obtained by short-circuiting the inductor (Remark 3.9(2)) as

$$i_L(\infty) = \frac{V_i}{R}$$

The formula (3.39) can be used to obtain the inductor current as

$$i_L(t) \stackrel{(3.39)}{=} \left(I_0 - \frac{V_i}{R}\right)e^{-Rt/L} + \frac{V_i}{R} \quad (\text{E3.1.1})$$

which agrees with Equation (3.19). Also, noting that, as for the inductor voltage,

- the initial value is $v_L(0^+) = V_i - R i_L(0^+) = V_i - R I_0$ and
- the final value is obtained by short-circuiting the inductor (Remark 3.9(2)) as $v_L(\infty) = 0$,

the formula (3.39) can be used to obtain the inductor voltage as

$$v_L(t) \stackrel{(3.39)}{=} (V_i - R I_0)e^{-Rt/L} \quad (\text{E3.1.2})$$

which agrees with Equation (3.24). Here care should be taken not to substitute $v_L(0) = 0$ even if the inductor acts like a short-circuit at $t = 0^-$ in the DC steady state (Remark 3.2(2)), because the inductor voltage is not subject to the continuity rule.

- (b) Let us use the formula (3.39) to find the capacitor voltage/current for the RC circuit of Figure 3.8(a). Note that the time constant is $T = RC$ and as for the capacitor voltage,

- the initial value is $v_C(0^+) \stackrel{\text{continuity}}{=} v_C(0) = V_0$ and
- the final value is obtained by open-circuiting the capacitor (Remark 3.9(2)) as

$$v_C(\infty) = V_i - Ri(\infty) = V_i$$

The formula (3.39) can be used to obtain the capacitor voltage as

$$v_C(t) \stackrel{(3.39)}{=} (V_0 - V_i)e^{-t/(RC)} + V_i \quad (\text{E3.1.3})$$

which agrees with Equation (3.35). Also, noting that, as for the capacitor current,

- the initial value is

$$i_C(0^+) = \frac{V_i - v_C(0^+)}{R} = \frac{V_i - V_0}{R}$$

- and the final value is obtained by open-circuiting the capacitor (Remark 3.9(2)) as $i_C(\infty) = 0$.

the formula (3.39) can be used to obtain the capacitor voltage as

$$i_C(t) = \frac{V_i - V_0}{R} e^{-t/(RC)} \quad (\text{E3.1.4})$$

which agrees with Equation (3.30). Care should be taken not to substitute $i_C(0) = 0$ even if the capacitor acts like an open circuit at $t = 0^-$ in the DC steady state (Remark 3.4(2)), because the capacitor current is not subject to the continuity rule.

(Example 3.2) Time Constant and DC Solution of First-Order Circuits Having Multiple Resistors

- (a) Consider the circuit of Figure 3.10.1 in which the switch has been closed for a long time till it is to be opened at $t = 0$. Note that this circuit has two different time constants depending on whether the switch is closed or open. Following the steps suggested in Remark 3.8, we remove the (independent) current source of 6A by opening it and find the equivalent resistance (seen from nodes 2 and 3 with the inductor L open) and the corresponding time constants as

$$R_{eq1} = R_1 + R_2 = \frac{1}{2} + \frac{1}{4} = \frac{3}{4} \Omega \rightarrow T_1 = L/R_{eq1} = 4/3 \text{ s with the switch closed} \quad (\text{E3.2.1})$$

$$R_{eq2} = R_1 + R_2 + R_3 = \frac{1}{2} + \frac{1}{4} + \frac{1}{4} = 1 \Omega \rightarrow T_2 = L/R_{eq2} = 1 \text{ s with the switch open} \quad (\text{E3.2.2})$$

Subject to the continuity rule, the initial current through the inductor L at $t = 0^+$ is equal to that at $t = 0$ corresponding to the DC steady state (with the switch closed) where the inductor acts like a short-circuit. Therefore it is found to be the current through R_2 , which is connected in parallel with R_1 to make a current divider for the current source of $I_i = 6 \text{ A}$:

$$i_L(0^+) = i_L(0)|_{L \text{ shorted}} = \frac{R_1}{R_1 + R_2} I_i = \frac{1/2}{1/2 + 1/4} 6 = 4 \text{ A} \quad (\text{E3.2.3})$$

The final current of the inductor L is $i_L(t)$ at $t = \infty$ corresponding to the DC steady state (with the switch open) where the inductor acts like a short-circuit. Therefore it is found to be the current through R_2 - R_3 , which is connected in parallel with R_1 to make a current divider for the current source of $I_1 = 6$ A:

$$i_L(\infty)|_{L \text{ shorted}} = \frac{R_1}{R_1 + R_2 + R_3} I_1 = \frac{1/2}{1/2 + 1/4 + 1/4} 6 = 3 \text{ A} \quad (\text{E3.2.4})$$

Now the formula (3.39) can be used to find the inductor current as

$$i_L(t) \stackrel{(3.39)}{=} [i_L(0^+) - i_L(\infty)]e^{-t/T_2} + i_L(\infty) = (4 - 3)e^{-t} + 3 = 3 + e^{-t} \text{ [A]} \quad \text{for } t \geq 0 \quad (\text{E3.2.5})$$

As a rather formal approach, the $I_1 = 6$ A source with the resistor $R_1 = (1/2)\Omega$ in parallel is transformed into the $I_1 R_1 = 6 \text{ A} \times (1/2)\Omega = 3$ V source with the resistor $R_1 = (1/2)\Omega$ in series, the 3 V source and the initial inductor current $I_0 = 4$ A are replaced by their s -domain equivalent voltage sources of $3/s$ and $LI_0 = 4$ V (Figure 3.6(b1)), respectively, as shown in Figure 3.10.1(b), and then the mesh equation is set up and solved:

$$(R_1 + R_2 + R_3 + sL) I_L(s) = \frac{3}{s} + 4; \quad (s + 1)I_L(s) = \frac{3}{s} + 4 \quad (\text{E3.2.6})$$

$$I_L(s) = \frac{3}{s(s + 1)} + \frac{4}{s + 1} = \frac{3}{s} + \frac{1}{s + 1} \quad (\text{E3.2.7})$$

$$i_L(t) \stackrel{\text{Table A.1(3),(5)}}{=} (3 + e^{-t})u_s(t) \text{ [A]} \quad (\text{E3.2.8})$$

- (b) Consider the circuit of Figure 3.10.2 in which the switch has been opened for a long time until it is closed at $t = 0$. Note that this circuit has two different time constants depending on whether the switch is open or closed. Taking the steps suggested in Remark 3.8 we remove the (independent) 6 V voltage source by short-circuiting it and find the equivalent resistance (seen from the terminals of the capacitor C with the capacitor opened) and the corresponding time constants are formed as

$$R_{\text{eq1}} = R_1 \parallel R_2 = \frac{1}{1/2 + 1/4} = \frac{4}{3} \Omega \rightarrow T_1 = R_{\text{eq1}} C = 4/3 \text{ s} \quad \text{with the switch open} \quad (\text{E3.2.9a})$$

$$R_{\text{eq2}} = R_1 \parallel R_2 \parallel R_3 = \frac{1}{1/2 + 1/4 + 1/4} = 1 \Omega \rightarrow T_2 = R_{\text{eq2}} C = 1 \text{ s} \quad \text{with the switch closed} \quad (\text{E3.2.9b})$$

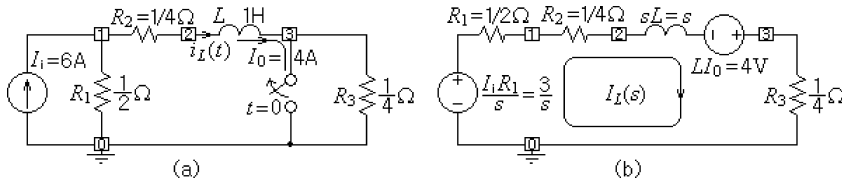


Figure 3.10.1 A first-order RL circuit with multiple resistors and its s -domain equivalent

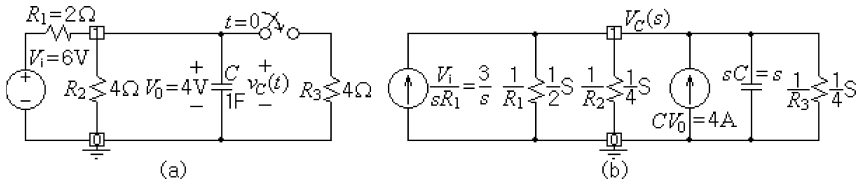


Figure 3.10.2 A first-order RC circuit with multiple resistors and its s -domain equivalent

Subject to the continuity rule, the initial voltage across the capacitor C at $t = 0^+$ is equal to that at $t = 0$ corresponding to the DC steady state (with the switch opened) where the capacitor acts like an open circuit and so the capacitor voltage is 4 V. This has been found as the voltage across R_2 , which is connected in series with R_1 to make a voltage divider for the voltage source of $V_i = 6$ V:

$$v_C(0^+) = v_C(\infty)|_{C \text{ opened}} = \frac{R_2}{R_1 + R_2} V_i = \frac{4}{2 + 4} 6 = 4V \quad (\text{E3.2.10})$$

The final voltage across the capacitor C is $v_C(t)$ at $t = \infty$ corresponding to the DC steady state (with the switch closed) where the capacitor acts like an open circuit and so the capacitor voltage is 3 V. This has been found as the voltage through $R_2 \parallel R_3$, which is connected in series with R_1 to make a voltage divider for the voltage source of $V_i = 6$ V:

$$v_C(\infty)|_{C \text{ opened}} = \frac{R_2 \parallel R_3}{R_1 + (R_2 \parallel R_3)} V_i = \frac{2}{2 + 2} 6 = 3V \quad (\text{E3.2.11})$$

Now the formula (3.39) is used to find the capacitor voltage as

$$\begin{aligned} v_C(t) &\stackrel{(3.39)}{=} [v_C(0^+) - v_C(\infty)]e^{-t/T_2} + v_C(\infty) \\ &= (4 - 3)e^{-t} + 3 = 3 + e^{-t} \text{ [V]} \quad \text{for } t \geq 0 \end{aligned} \quad (\text{E3.2.12})$$

As a rather formal approach, the $V_i = 6$ V voltage source with the resistor $R_1 = 2 \Omega$ in series is transformed into the $V_i/R_1 = 6 \text{ V}/2 \Omega = 3 \text{ A}$ current source with the resistor $R_1 = 2 \Omega$ in parallel, the 3 A current source and the initial capacitor voltage $V_0 = 4$ V are replaced by their s -domain equivalent current sources of $3/s$ and $CV_0 = 4$ A (Figure 3.6(c2)), respectively, as shown in Figure 3.10.2(b), and then the node equation is set up and solved:

$$\left(\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + sC \right) V_C(s) = \frac{3}{s} + 4; \quad (s + 1)V_C(s) = \frac{3}{s} + 4 \quad (\text{E3.2.13})$$

$$V_C(s) = \frac{3}{s(s + 1)} + \frac{4}{s + 1} = \frac{3}{s} + \frac{1}{s + 1} \quad (\text{E3.2.14})$$

$$v_C(t) \stackrel{\text{Table A.1(3),(5)}}{=} (3 + e^{-t}) u_s(t) \text{ [V]} \quad (\text{E3.2.15})$$

(Example 3.3) DC Solution of an RL Circuit Containing a Dependent (Controlled) Source

Consider the circuit of Figure 3.11(a). Note that the $v_{23}/2$ current source can be duplicated to make two copies in series, which are associated with the inductor L and the 4Ω resistor to be transformed into the voltage sources, as depicted in Figures 3.11(b) and (c2). In this case, in order to find the inductor current, the associated current source $v_{23}/2$ should be subtracted from the mesh current i_1 . Note also that the situation of the switch being open/closed in the circuit of Figure 3.11(a) corresponds to the case where the transfer switch is connected to position a/b in the circuits of Figures 3.11(c1) and (c2) and their equivalents in Figures 3.11(d1) and (d2).

- (a) Suppose the switch has been closed for a long time until it is to be opened at $t = 0$. The initial inductor current can be found from Figure 3.11(c1) with the transfer switch connected at position b , where the inductor is shorted in the DC steady state:

$$(4 + 1 + 6) i_1 = 2v_{23} + 36 = 2(1 \times i_1) + 36; \quad i_1 = 4A \quad (\text{E3.3.1})$$

$$i_L(0) = i_1 - \frac{1}{2} v_{23} = i_1 - \frac{1}{2} i_1 = \frac{1}{2} i_1 = 2A \quad \text{with the SW at position } b \quad (\text{E3.3.2})$$

The final inductor current can be found from Figure 3.11(c1) with the transfer switch connected at position *a*, where the inductor is shorted in the steady state:

$$(4 + 1 + 15)i_1 = 2v_{23} + 90 = 2(1 \times i_1) + 90; \quad i_1 = 5 \text{ A} \quad (\text{E3.3.3})$$

$$i_L(\infty) = i_1 - \frac{1}{2}v_{23} = i_1 - \frac{1}{2}i_1 = \frac{1}{2}i_1 = 2.5 \text{ A} \quad \text{with the SW at position } a \quad (\text{E3.3.4})$$

If we have neither the equivalent resistance nor the time constant, we might need to construct the transformed circuit with the initial inductor current represented by a voltage source as in Figure 3.11(c2) and apply the mesh analysis to proceed as follows:

$$(4 + sL + 1 + 15)I_1(s) = \frac{1}{2}sLV_{23}(s) + Li_L(0) + 2V_{23}(s) + \frac{90}{s} \quad \text{with } L = 1\text{H},$$

$$i_L(0) = 2 \text{ A}, \quad V_{23}(s) = I_1(s) \quad (\text{E3.3.5})$$

$$\left(\frac{1}{2}s + 18\right)I_1(s) = 2 + \frac{90}{s}; \quad I_1(s) = \frac{4}{s + 36} + \frac{180}{s(s + 36)} = \frac{5}{s} - \frac{1}{s + 36}$$

$$i_1(t) = 5 - e^{-36t} [\text{A}]; \quad i_L(t) = i_1(t) - \frac{1}{2}v_{23}(t) = \frac{1}{2}i_1(t) = \frac{1}{2}(5 - e^{-36t}) [\text{A}] \quad (\text{E3.3.6})$$

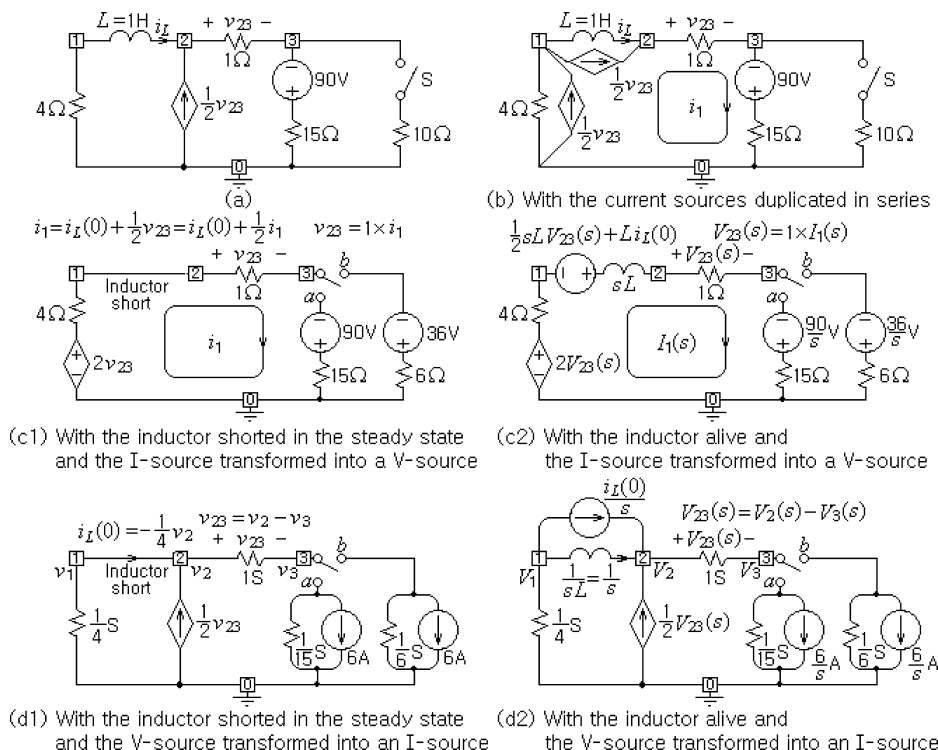


Figure 3.11 An *RL* circuit containing a dependent (controlled) source for Example 3.3

As an alternative, we can construct the transformed circuit with the initial current represented by a current source as in Figure 3.11(d2) and apply the node analysis to proceed as follows:

$$\begin{bmatrix} 1/4 + 1/s & -1/s & 0 \\ -1/s & 1/s + 1 & -1 \\ 0 & -1 & 1 + 1/15 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \\ V_3(s) \end{bmatrix} = \begin{bmatrix} -i_L(0)/s \\ i_L(0)/s + V_{23}(s)/2 \\ -6/s \end{bmatrix} \quad \text{with } V_{23}(s) = V_2(s) - V_3(s)$$

$$\begin{bmatrix} 1/4 + 1/s & -1/s & 0 \\ -1/s & 1/s + 1/2 & -1/2 \\ 0 & -1 & 1 + 1/15 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \\ V_3(s) \end{bmatrix} = \begin{bmatrix} -i_L(0)/s \\ i_L(0)/s \\ -6/s \end{bmatrix} \quad \text{with the SW at position } a \quad (\text{E3.3.7})$$

```
>>syms s
>>L=1; sL=s*L; iL0=2; Is=[ -iL0/s; iL0/s; -6/s] ;
>>Ys=[ 1/4+1/sL -1/sL 0; -1/s 1/s+1/2 -1/2; 0 -1 1+1/15] ;
>>Vs=Ys\Is; ILs=(Vs(1)-Vs(2))/sL + iL0/s; iL1t=ilaplace(ILs)
iL1t = 5/2-1/2*exp(-36*t) % exp(-18*t)*sinh(18*t)+2 in MATLAB 7.x
```

This agrees with Equation (E3.3.6). If the equivalent resistance 36Ω , seen from L (with L open and the switch open) is given (as obtained in Example 2.20), the time constant of the circuit could be found to be $T_1 = L/R_{\text{eq1}} = (1/36)\text{ s}$ and the formula (3.39) could be used to obtain the DC solution as

$$i_L(t) = [i_L(0) - i_L(\infty)]e^{-t/T_1} + i_L(\infty) \stackrel{(\text{E3.3.2}), (\text{E3.3.4})}{=} (2 - 2.5)e^{-36t} + 2.5 = 2.5 - 0.5e^{-36t} \text{ [A]} \quad (\text{E3.3.8})$$

Note that the initial/final currents can also be obtained from the circuit of Figure 3.11(d1):

$$\begin{bmatrix} 1 + 1/4 & -1 \\ -1 & 1 + 1/6 \end{bmatrix} \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} (1/2)v_{23} \\ -6 \end{bmatrix} = \begin{bmatrix} (1/2)(v_2 - v_3) \\ -6 \end{bmatrix} \quad \text{with the SW at position } b$$

$$\begin{bmatrix} 1 - 1/4 & -1/2 \\ -1 & 1 + 1/6 \end{bmatrix} \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 0 \\ -6 \end{bmatrix}; \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \frac{1}{7/8 - 1/2} \begin{bmatrix} -3 \\ -4.5 \end{bmatrix} = \begin{bmatrix} -8 \\ -12 \end{bmatrix}; i_L(0) = -\frac{1}{4}v_2 = 2 \text{ A} \quad (\text{E3.3.9})$$

$$\begin{bmatrix} 1 + 1/4 & -1 \\ -1 & 1 + 1/15 \end{bmatrix} \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} (1/2)v_{23} \\ -6 \end{bmatrix} = \begin{bmatrix} (1/2)(v_2 - v_3) \\ -6 \end{bmatrix} \quad \text{with the SW at position } a$$

$$\begin{bmatrix} 1 - 1/4 & -1/2 \\ -1 & 1 + 1/15 \end{bmatrix} \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 0 \\ -6 \end{bmatrix}; \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \frac{1}{4/5 - 1/2} \begin{bmatrix} -3 \\ -4.5 \end{bmatrix} = \begin{bmatrix} -10 \\ -15 \end{bmatrix}; i_L(\infty) = -\frac{1}{4}v_2 = 2.5 \text{ A} \quad (\text{E3.3.10})$$

- (b) Suppose the switch has been opened for a long time until it is to be closed at $t = 0$. The initial/final values of the inductor currents are the same as the final/initial values for the case of (a):

$$i_L(0) = 2.5 \text{ A} \quad \text{and} \quad i_L(\infty) = 2 \text{ A} \quad (\text{E3.3.11})$$

If we have is neither the equivalent resistance nor the time constant, we might be a need to apply the mesh analysis to the transformed circuit in Figure 3.11(c2) and proceed as follows:

$$(4 + sL + 1 + 6)I_1(s) = \frac{1}{2}sL V_{23}(s) + Li_L(0) + 2V_{23}(s) + \frac{36}{s}$$

with $L = 1\text{H}$, $i_L(0) = 2.5\text{A}$, $V_{23}(s) = I_1(s)$ (E3.3.12)

$$\left(\frac{1}{2}s + 9\right)I_1(s) = 2.5 + \frac{36}{s}, \quad I_1(s) = \frac{5}{s + 18} + \frac{72}{s(s + 18)} = \frac{4}{s} + \frac{1}{s + 18}$$

$$i_1(t) = 4 + e^{-18t}[\text{A}], \quad i_L(t) = i_1(t) - \frac{1}{2}v_{23}(t) = \frac{1}{2}i_1(t) = 2 + \frac{1}{2}e^{-18t}[\text{A}] \quad (\text{E3.3.13})$$

As an alternative, the node analysis might be applied to the transformed circuit in Figure 3.11(d2) and proceed as follows:

$$\begin{bmatrix} 1/4 + 1/s & -1/s & 0 \\ -1/s & 1/s + 1/2 & -1/2 \\ 0 & -1 & 1 + 1/6 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \\ V_3(s) \end{bmatrix} = \begin{bmatrix} -i_L(0)/s \\ i_L(0)/s \\ -6/s \end{bmatrix} \quad \text{with the SW at position } b \quad (\text{E3.3.14})$$

```
>>iL0=2.5; Is=[ -iL0/s; iL0/s; -6/s] ;
>>Ys=[ 1/4+1/sL -1/sL 0; -1/s 1/s+1/2-1/2; 0 -1 1+1/6] ;
>>Vs=Ys\Is; ILs=(Vs(1)-Vs(2))/sL + iL0/s; iL2t=ilaplace(ILs)
iL2t = 2 + 1/2*exp(-18*t) % -exp(-9*t)*sinh(9*t)+5/2 in MATLAB 7.x
```

This agrees with Equation (E3.3.13). If the equivalent resistance 18Ω , seen from L (with L open and the switch closed) is given (as obtained in Example 2.20), the time constant of the circuit could be found to be $T_2 = L/R_{\text{eq2}} = (1/18)\text{s}$ and the formula (3.39) could be used to obtain the DC solution as

$$i_L(t) = [i_L(0) - i_L(\infty)]e^{-t/T_2} + i_L(\infty) \stackrel{(\text{E3.3.11})}{=} (2.5 - 2)e^{-18t} + 2 = 2 + 0.5e^{-18t}[\text{A}] \quad (\text{E3.3.15})$$

3.4.4 Sequential Switching

This section deals with the case where a switching action is performed more than one time, i.e. a single switch is switched on/off or back/forth, or multiple switches are opened/closed at different instants of time, which is called *sequential switching*. In order to solve a circuit problem involved with sequential switching, there is a need to determine the inductor currents and the capacitor voltages at every switching instant and to use them to find the initial conditions for the circuit that is to be changed by the switching, because they are subject to the continuity rule.

Consider the RL circuit of Figure 3.12(a1) in which the (make-before-break) transfer switch has been connected at position a for a long time till it is to be flipped to position b at $t = 0\text{s}$. Then it is to be flipped back to position a at $t = t_1[\text{s}]$. First, in order to find the inductor current after the first switching at $t = 0\text{s}$, its initial value $i_L(0^+)$ is determined, which is equal to the final value in the DC steady state with the switch connected to position a :

$$i_L(0^+) = i_L(0) = i_L(0^-) = V_{i1}/R_1 \quad (\text{with the inductor shorted})$$

The time constant and the final value of the inductor current for the RL circuit with the switch connected to position b are

$$T_2 = L/R_2 \quad \text{and} \quad i_L(\infty) = V_{i2}/R_2$$

Thus the formula (3.39) can be used to find the inductor current for $0 \leq t < t_1$ [s] (before the next switching instant t_1) as

$$i_L(t) \stackrel{(3.39)}{=} [i_L(0^+) - i_L(\infty)]e^{-R_2 t/L} + i_L(\infty) = \left(\frac{V_{i1}}{R_1} - \frac{V_{i2}}{R_2}\right)e^{-R_2 t/L} + \frac{V_{i2}}{R_2} \quad \text{for } 0 \leq t < t_1 \quad (3.40)$$

Now for the circuit formed by the second switching to position *a*, the time constant and the initial/final values of the inductor current are

$$\begin{aligned} T_1 &= L/R_1 \\ i_L(t_1^+) &= i_L(t_1) \stackrel{(3.40)}{=} \left(\frac{V_{i1}}{R_1} - \frac{V_{i2}}{R_2}\right)e^{-R_2 t_1/L} + \frac{V_{i2}}{R_2} \quad \text{for } t = t_1 \\ i_L(\infty) &= V_{i1}/R_1 \end{aligned}$$

Thus, the formula (3.39) can be used to obtain the inductor current for $t \geq t_1$ [s] (after the second switching instant t_1) as

$$\begin{aligned} i_L(t) &\stackrel{(3.39)}{=} [i_L(t_1^+) - i_L(\infty)]e^{-R_1(t-t_1)/L} + i_L(\infty) \quad \text{for } t \geq t_1 \\ &= \left(\frac{V_{i2}}{R_2} - \frac{V_{i1}}{R_1}\right)(1 - e^{-R_1(t-t_1)/L}) + \frac{V_{i1}}{R_1} \end{aligned} \quad (3.41)$$

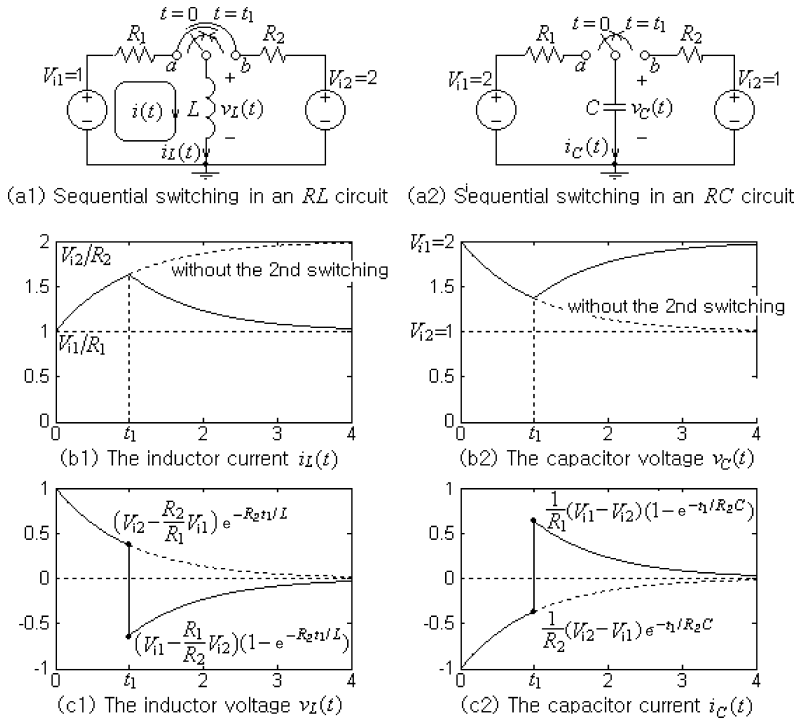


Figure 3.12 Sequential switchings in an RL circuit and an RC circuit

These two results (3.40) and (3.41) can be combined to write

$$i_L(t) = \begin{cases} \left(\frac{V_{i1}}{R_1} - \frac{V_{i2}}{R_2} \right) e^{-R_2 t/L} + \frac{V_{i2}}{R_2} & \text{for } 0 \leq t < t_1 \\ \left(\frac{V_{i2}}{R_2} - \frac{V_{i1}}{R_1} \right) (1 - e^{-R_2 t_1/L}) e^{-R_1(t-t_1)/L} + \frac{V_{i1}}{R_1} & \text{for } t > t_1 \end{cases} \quad (3.42)$$

which, with the assumption that $V_{i1}/R_1 < V_{i2}/R_2$, is depicted in Figure 3.12(b1). This inductor current can also be substituted into the voltage–current relationship (3.1a) to find the inductor voltage as

$$v_L(t) \stackrel{(3.1a)}{=} L \frac{di_L(t)}{dt} \stackrel{(3.42)}{=} \begin{cases} \left(V_{i2} - \frac{R_2}{R_1} V_{i1} \right) e^{-R_2 t/L} & \text{for } 0 \leq t < t_1 \\ \left(V_{i1} - \frac{R_1}{R_2} V_{i2} \right) (1 - e^{-R_2 t_1/L}) e^{-R_1(t-t_1)/L} & \text{for } t > t_1 \end{cases} \quad (3.43)$$

which is depicted in Figure 3.12(c1).

This time the RC circuit of Figure 3.12(a2) is considered, in which the switch has been connected at position a for a long time till it is to be flipped to position b at $t = 0$ s. Then it is to be flipped back to position a at $t = t_1$ [s]. First, in order to find the capacitor voltage after the first switching at $t = 0$ s, its initial value $v_C(0^+)$ is determined, which is equal to the final value in the DC steady state with the switch connected to position a :

$$v_C(0^+) = v_C(0) = v_C(0^-) = V_{i1} \quad (\text{with the capacitor opened})$$

The time constant and the final value of the capacitor voltage current for the RC circuit with the switch connected to position b are

$$T_2 = R_2 C \quad \text{and} \quad v_C(\infty) = V_{i2}$$

Thus the formula (3.39) can be used to find the capacitor voltage for $0 \leq t < t_1$ [s] (before the second switching instant t_1) as

$$\begin{aligned} v_C(t) &\stackrel{(3.39)}{=} [v_C(0^+) - v_C(\infty)] e^{-t/(R_2 C)} + v_C(\infty) \\ &= (V_{i1} - V_{i2}) e^{-t/(R_2 C)} + V_{i2} \quad \text{for } 0 \leq t < t_1 \end{aligned} \quad (3.44)$$

Now for the circuit formed by the second switching to position a , the time constant and the initial/final values of the capacitor voltage are

$$\begin{aligned} T_1 &= R_1 C \\ v_C(t_1^+) &\stackrel{(3.5)}{=} v_C(t_1) \stackrel{(3.44)}{=} (V_{i1} - V_{i2}) e^{-t_1/(R_2 C)} + V_{i2} \quad \text{for } t = t_1 \\ v_C(\infty) &= V_{i1} \end{aligned}$$

Thus, the formula (3.39) can be used to obtain the capacitor voltage for $t \geq t_1$ [s] (after the second switching instant t_1) as

$$\begin{aligned} v_C(t) &\stackrel{(3.39)}{=} [v_C(t_1^+) - v_C(\infty)] e^{-(t-t_1)/(R_1 C)} + v_C(\infty) \quad \text{for } t \geq t_1 \\ &= (V_{i2} - V_{i1}) (1 - e^{-t_1/R_2 C}) e^{-(t-t_1)/(R_1 C)} + V_{i1} \end{aligned} \quad (3.45)$$

These two results (3.44) and (3.45) can be combined to write

$$v_C(t) = \begin{cases} (V_{i1} - V_{i2})e^{-t/(R_2C)} + V_{i2} & \text{for } 0 \leq t < t_1 \\ (V_{i2} - V_{i1})(1 - e^{-t_1/(R_2C)})e^{-(t-t_1)/(R_1C)} + V_{i1} & \text{for } t > t_1 \end{cases} \quad (3.46)$$

which, with the assumption that $V_{i1} > V_{i2}$, is depicted in Figure 3.12(b2). This capacitor voltage can also be substituted into the voltage–current relationship (3.4a) to find the capacitor current as

$$i_C(t) = C \frac{dv_C(t)}{dt} = \begin{cases} \frac{1}{R_2} (V_{i2} - V_{i1})e^{-t/(R_2C)} & \text{for } 0 \leq t < t_1 \\ \frac{1}{R_1} (V_{i1} - V_{i2})(1 - e^{-t_1/(R_2C)})e^{-(t-t_1)/(R_1C)} & \text{for } t > t_1 \end{cases} \quad (3.47)$$

which is depicted in Figure 3.12(c2).

3.4.5 AC-Excited First-Order Circuits

Consider the RL circuit of Figure 3.7(a) where the initial inductor current is $i(0) = 0$ and the voltage source is of the AC type as

$$v_i(t) = V_{im} \cos \omega t \, u_s(t) \quad (3.48)$$

Noting that from Table A.1(8) in Appendix A, the Laplace transform of this cosine function is

$$V_i(s) \stackrel{\text{Table A.1(8)}}{=} \frac{V_{im}s}{s^2 + \omega^2}$$

the s -domain mesh current can be obtained as

$$I(s) \stackrel{(3.18a)}{=} \frac{V_i(s)}{R + sL} = \frac{V_{im}/L}{s + R/L} \frac{s}{s^2 + \omega^2} \quad (3.49)$$

To take the inverse Laplace transform, this can be written as

$$I(s) = \frac{K}{s + R/L} + \frac{As + B\omega}{s^2 + \omega^2} = \frac{(K + A)s^2 + (B\omega + RA/L)s + (BR/L + K\omega)\omega}{(s + R/L)(s^2 + \omega^2)} \quad (3.50)$$

and the numerators of Equations (3.49) and (3.50) are equated to find the coefficients K , A , and B as follows:

$$K \stackrel{(A.28a)}{=} \left(s + \frac{R}{L} \right) I(s) \Big|_{s=-R/L} \stackrel{(3.49)}{=} \frac{V_{im}}{L} \frac{s}{s^2 + \omega^2} \Big|_{s=-R/L} = \frac{-RV_{im}}{R^2 + (\omega L)^2}$$

The coefficients of the terms of degree 2: $K + A = 0, \quad A = -K = \frac{RV_{im}}{R^2 + (\omega L)^2}$

The coefficients of the terms of degree 0: $\frac{BR}{L} + K\omega = 0, \quad B = -\frac{\omega L}{R} K = \frac{\omega L V_{im}}{R^2 + (\omega L)^2}$

Finally, the inverse Laplace transform of the s -domain solution (3.50) is taken to obtain

$$\begin{aligned}
 i(t) &= \mathcal{L}^{-1}\{I(s)\} = (Ke^{-Rt/L} + A \cos \omega t + B \sin \omega t) u_s(t) \\
 &= \left[Ke^{-Rt/L} + \sqrt{A^2 + B^2} \cos\left(\omega t - \tan^{-1} \frac{B}{A}\right) \right] u_s(t) \\
 &= \left[\frac{-RV_{im}}{R^2 + (\omega L)^2} e^{-Rt/L} + \frac{V_{im}}{\sqrt{R^2 + (\omega L)^2}} \cos\left(\omega t - \tan^{-1} \frac{\omega L}{R}\right) \right] u_s(t) \quad (3.51)
 \end{aligned}$$

which is depicted in Figure 3.13(a).

Now consider the RC circuit of Figure 3.8(a) where the initial capacitor voltage is $v_C(0) = 0$ and the voltage source is of the AC type to generate

$$v_i(t) = V_{im} \sin \omega t u_s(t) \quad (3.52)$$

Noting that from Table A.1(7) in Appendix A, the Laplace transform of this sine function is

$$V_i(s) \stackrel{\text{Table A.1(7)}}{=} \frac{V_{im}\omega}{s^2 + \omega^2}$$

the s -domain capacitor voltage can be obtained as

$$V_C(s) \stackrel{(3.34a)}{=} \frac{V_i(s)/R}{sC + 1/R} = \frac{V_{im}/(RC)}{s + 1/(RC)} \frac{\omega}{s^2 + \omega^2} \quad (3.53)$$

To take the inverse Laplace transform, this can be written as

$$V_C(s) = \frac{K}{s + 1/(RC)} + \frac{As + B\omega}{s^2 + \omega^2} = \frac{(K + A)s^2 + [B\omega + A/(RC)]s + [B/(RC) + K\omega]\omega}{[s + 1/(RC)](s^2 + \omega^2)} \quad (3.54)$$

and the numerators of Equations (3.53) and (3.54) are equated to find the coefficients K , A , and B as follows:

$$K \stackrel{(A.28a)}{=} \left(s + \frac{1}{RC} \right) V_C(s) \Big|_{s=-1/(RC)} \stackrel{(3.53)}{=} \frac{V_{im}}{RC} \frac{\omega}{s^2 + \omega^2} \Big|_{s=-1/(RC)} = \frac{R/(\omega C)}{R^2 + [1/(\omega C)]^2} V_{im}$$

$$\text{The coefficients of the terms of degree 2: } K + A = 0, \quad A = -K = -\frac{R/(\omega C)}{R^2 + [1/(\omega C)]^2} V_{im}$$

$$\text{The coefficients of the terms of degree 1: } B\omega + A/RC = 0, \quad B = -\frac{A}{\omega RC} = \frac{[1/(\omega C)]^2}{R^2 + [1/(\omega C)]^2} V_{im}$$

Finally, the inverse Laplace transform of the s -domain solution (3.54) is taken to obtain

$$\begin{aligned}
 v_C(t) &= \mathcal{L}^{-1}\{V_C(s)\} = (Ke^{-t/(RC)} + A \cos \omega t + B \sin \omega t) u_s(t) \\
 &= \left[Ke^{-t/(RC)} + \sqrt{A^2 + B^2} \sin\left(\omega t + \tan^{-1} \frac{A}{B}\right) \right] u_s(t) \\
 &= \left[\frac{(R/\omega C)V_{im}}{R^2 + [1/(\omega C)]^2} e^{-t/(RC)} + \frac{V_{im}/\omega C}{\sqrt{R^2 + [1/(\omega C)]^2}} \sin(\omega t - \tan^{-1} \omega RC) \right] u_s(t) \quad (3.55)
 \end{aligned}$$

which is depicted in Figure 3.13(b) (see Section 3.7.2).

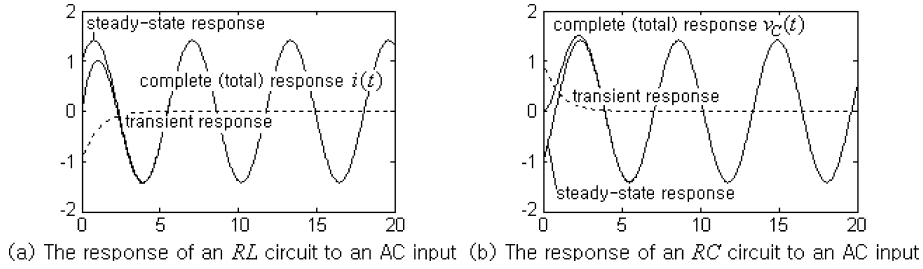


Figure 3.13 The responses of RL/RC circuits to an AC input

Note. It can be seen from Equations (3.51) and (3.55) that the AC-excited first-order circuits containing such energy storage elements as inductors/capacitors have a transient response (decaying exponentially to zero with a certain time constant) as well as a steady state response that is another sinusoid of the same frequency, but with different amplitude and phase from those of the AC input source.

3.5 Analysis of First-Order OP Amp Circuits

In this section some first-order circuits will be studied that contain one or more OP Amps, which are of great use.

3.5.1 First-Order OP Amp Circuits with Negative Feedback

Let us first consider the RC circuit containing an OP Amp shown in Figure 3.14(a1). Since the OP Amp has a negative feedback path connecting its output terminal to its negative input terminal without having any positive feedback path (connecting its output terminal to its positive input terminal), the virtual short principle (Remark 1.2(2)) can be used to set the node voltage at the negative input terminal (node 2) of the OP Amp (almost) equal to that at the positive terminal that is grounded:

$$v_2(t) \simeq 0 \quad (\text{virtual ground})$$

Noting also that, by the virtual open principle (Remark 1.2(1)), no current can flow into or out of the input terminals of the (ideal) OP Amp, KCL can be applied to node 2 to write

$$\begin{aligned} \frac{v_2(t) - v_i(t)}{R} + C \frac{d[v_2(t) - v_o(t)]}{dt} &\stackrel{\text{KCL}}{=} 0 \xrightarrow{v_2=0} \frac{v_i(t)}{R} + C \frac{dv_o(t)}{dt} = 0 \\ \frac{dv_o(t)}{dt} &= -\frac{v_i(t)}{RC}; \quad v_o(t) = -\frac{1}{RC} \int_{-\infty}^t v_i(\tau) d\tau = -\frac{1}{C} \int_{-\infty}^0 i(\tau) d\tau - \frac{1}{RC} \int_0^t v_i(\tau) d\tau \end{aligned}$$

and finally, the input–output relationship is obtained as

$$\begin{aligned} v_o(t) &= v_o(0) - \frac{1}{RC} \int_0^t v_i(\tau) d\tau = -v_C(0) - \frac{1}{RC} \int_0^t v_i(\tau) d\tau \\ \text{with } v_C(0) &= \frac{1}{C} \int_{-\infty}^0 i(\tau) d\tau = \frac{1}{C} \int_{-\infty}^0 \frac{v_i(\tau)}{R} d\tau \end{aligned} \quad (3.56)$$

Since this implies that the output is qualitatively the integral of the input, the RC OP Amp circuit is called an *inverting RC integrator* or a *Miller integrator* with the integration time constant RC . As an alternative,

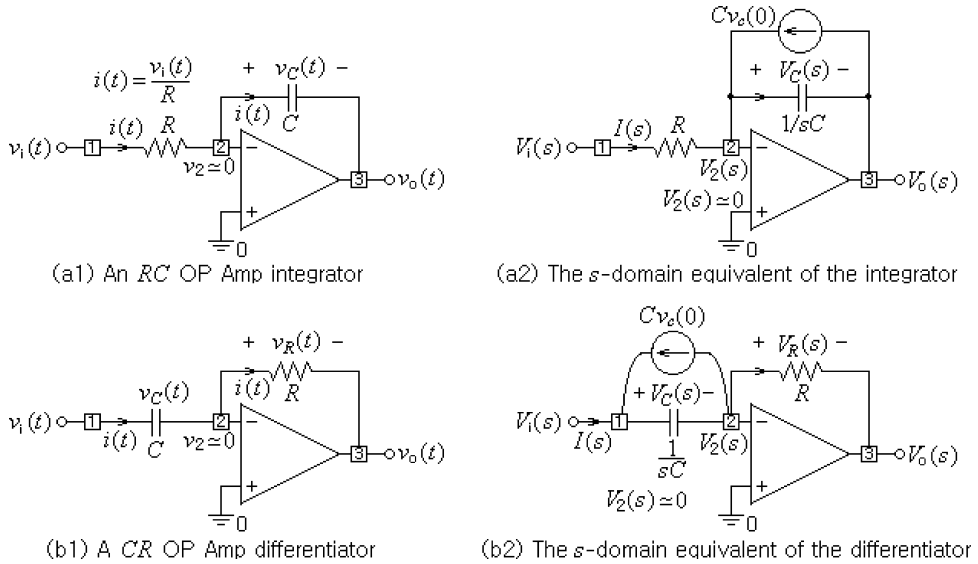


Figure 3.14 First-order circuits containing an OP Amp

this circuit can be transformed into its s -domain equivalent as depicted in Figure 3.14(a2) and KCL applied to node 2 to obtain the s -domain input–output relationship as

$$\frac{0 - V_i(s)}{R} + \frac{0 - V_o(s)}{1/(sC)} = C v_c(0); \quad V_o(s) = -\frac{1}{s} v_c(0) - \frac{1}{sRC} V_i(s) \quad (3.57)$$

which is the Laplace transform of the above t -domain input–output relationship.

Note. If the initial condition term is omitted, this s -domain input–output relationship is just like what can be obtained by substituting R and $1/(sC)$ for R_1 and R_f in Equation (2.22), which is the input–output relationship of an inverting amplifier consisting of an OP Amp together with two resistors R_1 and R_f .

Now let us consider the CR circuit containing an OP Amp shown in Figure 3.14(b1). Since this circuit has the same structure as the RC OP Amp circuit in Figure 3.14(a1), KCL and the virtual open principle with $v_2(t) \simeq 0$ (virtual ground) are applied to node 2 to obtain the input–output relationship as

$$C \frac{d[v_2(t) - v_i(t)]}{dt} + \frac{v_2(t) - v_o(t)}{R} \stackrel{\text{KCL}}{=} 0 \xrightarrow{v_2=0} \frac{v_o(t)}{R} + C \frac{dv_i(t)}{dt} = 0; \\ v_o(t) = -RC \frac{dv_i(t)}{dt} \quad (3.58)$$

Since this implies that the output is qualitatively the derivative of the input, the CR OP Amp circuit is called a CR differentiator. As an alternative, this circuit can be transformed into its s -domain equivalent as depicted in Figure 3.14(b2) and KCL can be applied to node 2 to obtain the s -domain input–output relationship as

$$\frac{0 - V_i(s)}{1/(sC)} + \frac{0 - V_o(s)}{R} = -C v_c(0); \quad V_o(s) = RC v_c(0) - sRC V_i(s) \quad (3.59)$$

which is the Laplace transform of the above t -domain input–output relationship.

Note. If the initial condition term is omitted, this s -domain input–output relationship is just like what can be obtained by substituting $1/(sC)$ and R for R_1 and R_f in Equation (2.22).

3.5.2 First-Order OP Amp Circuits with Positive Feedback

As stated in Remark 2.3(1) (Section 2.9.5.2), a positive feedback path (from the output to the positive input terminal) of an OP Amp destabilizes the output so that the output voltage will have the alternative of $+V_{om}$ (positive saturation voltage $+V_{sat}$) or $-V_{om}$ (negative saturation voltage $-V_{sat}$). As will be illustrated in this section, positive/negative feedback can be combined to generate a periodic waveform such as a rectangular or triangular wave, which seems to be a wonderful harmony of two antagonistic entities each causing instability/stability.

3.5.2.1 Square-Wave Generator

Consider the OP Amp circuit with positive/negative feedback in Figure 3.15(a). Noting that:

- (a) its output voltage will be either $v_o = +V_{om}$ or $-V_{om}$ depending on which one of the two input terminals is of higher voltage, and
- (b) the voltage of the positive input terminal is $v_+ = bv_o = \pm bV_{om}$ with $b = R_1/(R_1 + R_2)$,

suppose that the output voltage at some instant is $v_o = +V_{om}$. Then the positive input terminal of the OP Amp has the node voltage of

$$v_+ = +bV_{om} \quad \text{with} \quad b = \frac{R_1}{R_1 + R_2}$$

(which is possible only when $v_C = v_- < v_+$) and the voltage $v_C = v_-$ (at the $-$ input terminal) of the capacitor (connected via R_3 to the output terminal with $v_o = +V_{om}$) rises exponentially toward $+V_{om}$ till it catches up with $v_+ = +bV_{om}$. As soon as $v_C = v_-$ goes above $v_+ = +bV_{om}$ (so that $v_- > v_+$), the voltages at the output and $+$ input terminal will go down to

$$v_o = -V_{om} \quad \text{and} \quad v_+ = -bV_{om}$$

respectively. Then $v_C = v_-$ falls exponentially toward $-V_{om}$ till it touches down at $v_+ = -bV_{om}$. As soon as $v_C = v_-$ goes below $v_+ = -bV_{om}$ (so that $v_- < v_+$), the voltages at the output and $+$ input terminal will go up to

$$v_o = +V_{om} \quad \text{and} \quad v_+ = +bV_{om}$$

respectively. This process repeats itself periodically, generating a rectangular wave at the output, as depicted in Figure 3.15(b).

Now, in order to find the period of the rectangular wave, we start from observing that the voltage $v_C = v_-$ will go up and down repetitively in the range limited by the following two threshold values:

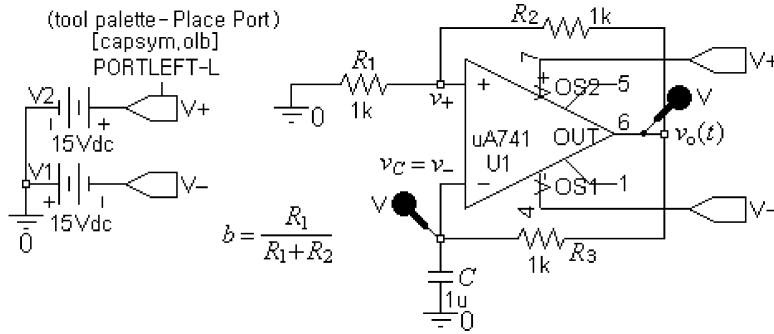
$$V_H = +bV_{om} \quad \text{and} \quad V_L = -bV_{om} \quad \text{with} \quad b = \frac{R_1}{R_1 + R_2} \quad (3.60)$$

and its waveform can be described by the formula (3.39) as

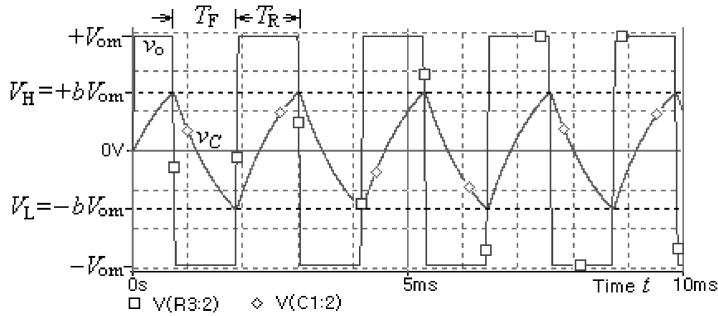
$$v_-(t) \stackrel{(3.39)}{=} [v_-(t_0^+) - v_-(\infty)]e^{-(t-t_0)/T} + v_-(\infty), \quad T = R_3C \quad (3.61)$$

Complying with this formula, the voltage $v_C = v_-$ during the rising interval is described by

$$v_-(t) \stackrel{(3.61)}{=} (V_L - V_{om})e^{-(t-t_0)/T} + V_{om}$$



(a) The PSpice schematic for a square-wave generator using an OP Amp



(b) The Probe output waveform in the PSpice A/D (Probe) window

Figure 3.15 An RC OP Amp circuit with positive/negative feedback as a square-wave generator

and the time T_R taken for $v_C = v_-$ to rise from V_L to V_H can be obtained as

$$V_H = (V_L - V_{om})e^{-T_R/T} + V_{om}; \quad e^{-T_R/T} = \frac{V_{om} - V_H}{V_{om} - V_L} \stackrel{(3.60)}{=} \frac{1-b}{1+b}$$

$$T_R = -T \ln \frac{1-b}{1+b} = R_3 C \ln \frac{1+b}{1-b} \quad (3.62)$$

In the same manner, the time T_F taken for $v_C = v_-$ to fall from V_H to V_L can be obtained as

$$V_L = (V_H + V_{om})e^{-T_F/T} - V_{om}; \quad e^{-T_F/T} = \frac{V_{om} + V_L}{V_{om} + V_H} \stackrel{(3.60)}{=} \frac{1-b}{1+b}$$

$$T_F = -T \ln \frac{1-b}{1+b} = R_3 C \ln \frac{1+b}{1-b} \quad (3.63)$$

Consequently, the period of the rectangular/sawtooth waves turns out to be

$$P = T_R + T_F = 2T \ln \frac{1+b}{1-b} = 2R_3 C \ln \frac{2R_1 + R_2}{R_2} \quad (3.64)$$

Note. This oscillatory phenomenon may be interpreted as the result of negative feedback, having the node voltage at the negative input terminal try to catch up with that at the positive input terminal, which runs away to the other side every time it is about to be caught.

Note. The formula (3.39) seems to be just right for this problem. Why are we not happy to have it at the right place at the right time?

Note. In fact, Figure 3.15(a) is the PSpice schematic, which can be created in the Schematic Editor window, where $R_1 = R_2 = R_3 = 1 \text{ k}\Omega$ and $C = 1 \text{ }\mu\text{F}$. This has been run with the Analysis type of ‘Time Response (Transient)’ to yield the waveforms depicted in Figure 3.15(b). To your mind, is the period of the voltage waveforms close to what you get from Equation (3.64)? Who could restrain himself/herself from shouting in admiration of the PSpice software developers at this time?

3.5.2.2 Rectangular/Triangular-Wave Generator

Consider the circuit of Figure 3.16(a) in which the left OP Amp U1 with negative feedback makes an inverting integrator (Section 3.5.1) and the right OP Amp U2 with positive feedback forms a noninverting trigger (Section 2.9.5.2). Noting that:

- (a) the output (v_{o1}) of the inverting integrator is applied to the input of the noninverting trigger,
- (b) the output (v_{o2}) of the noninverting trigger is fed back into the input of the inverting integrator, and
- (c) the two threshold values at which the noninverting trigger changes its output voltage v_{o2} are

$$V_H = +bV_{om}, \quad V_L = -bV_{om} \quad \text{with} \quad b = R_2/R_3 \quad (\text{Equation (2.40) or Figure 2.36}) \quad (3.65)$$

suppose the output voltage v_{o2} of the OP Amp U2 at some instant is

$$v_{o2} = +V_{om}$$

Then this constant positive voltage is fed back into the input (v_{i1}) of the inverting integrator to make its output (v_{o1}) decrease linearly till $v_{o1} < -bV_{om}$. When $v_{o1} < -bV_{om}$, the voltage v_{+2} (at the positive input terminal of U2) goes below $v_{-2} = 0$ (at the negative input terminal of U2) as

$$v_{+2} = v_{o1} - R_2 \frac{v_{o1} - V_{om}}{R_2 + R_3} = \frac{R_3}{R_2 + R_3} (v_{o1} + bV_{om}) < 0 = v_{-2} \rightarrow v_{+2} - v_{-2} < 0$$

so that the output voltage v_{o2} of U2 changes from $+V_{om}$ to $-V_{om}$:

$$v_{o2} = -V_{om}$$

This constant negative voltage is fed back into the input (v_{i1}) of the inverting integrator to make its output (v_{o1}) increase linearly till $v_{o1} > bV_{om}$. When $v_{o1} > bV_{om}$, the voltage v_{+2} goes above $v_{-2} = 0$:

$$v_{+2} = v_{o1} - R_2 \frac{v_{o1} - (-V_{om})}{R_2 + R_3} = \frac{R_3}{R_2 + R_3} (v_{o1} - bV_{om}) > 0 = v_{-2} \rightarrow v_{+2} - v_{-2} > 0$$

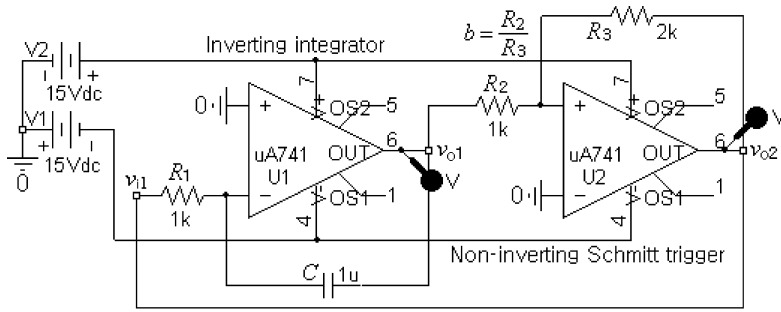
so that the output voltage v_{o2} of U2 changes from $-V_{om}$ back to $+V_{om}$:

$$v_{o2} = +V_{om}$$

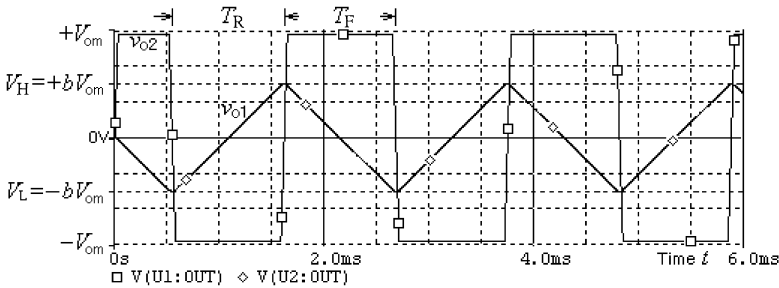
This process repeats itself periodically, generating a triangular wave at the output (v_{o1}) of U1 and a rectangular wave at the output (v_{o2}) of U2, as depicted in Figure 3.16(b).

Now, in order to find the period of the triangular/rectangular wave, we start from observing that the output voltage v_{o1} of the inverting integrator is described by Equation (3.56) as

$$v_{o1}(t) \stackrel{(3.56)}{=} v_{o1}(t_0) - \frac{1}{R_1 C} \int_{t_0}^t v_{o2}(\tau) d\tau \quad (3.66)$$



(a) The PSpice schematic for a rectangular/triangular generator using two OP Amps



(b) The output waveforms in the PSpice A/D (Probe) window

Figure 3.16 A combination of an inverting integrator and a noninverting trigger

Using this equation, the time taken for v_{o1} to rise from V_L to V_H and the time taken for v_{o1} to fall from V_H to V_L can be obtained as

$$V_H = V_L - \frac{1}{R_1 C} (-V_{om}) T_R; \quad T_R = \frac{R_1 C (V_H - V_L)}{V_{om}} \quad (3.67)$$

$$V_L = V_H - \frac{1}{R_1 C} (+V_{om}) T_F; \quad T_F = \frac{R_1 C (V_H - V_L)}{V_{om}} \quad (3.68)$$

Consequently, the period of the triangular/rectangular waves turns out to be

$$P = T_R + T_F = 2R_1 C \frac{V_H - V_L}{V_{om}} \stackrel{(3.65)}{=} 4bR_1 C = 4\frac{R_2}{R_3} R_1 C \quad (3.69)$$

Note. Although the circuits in Figures 3.15(a) and 3.16(a) are called *astable* (unstable) circuits, they have two ‘quasi-stable’ states in each of which they stay for a time interval determined by the time constant.

Note. A circuit like those of Figures 3.15(a) and 3.16(a) that repeatedly alternates between two states with a period depending on the charging of a capacitor is called a *relaxation oscillator*.

Note. Figure 3.16(a) is the PSpice schematic of a triangular/rectangular wave generator consisting of an inverting integrator (realized by an RC OP Amp circuit with negative feedback) and a noninverting trigger (realized by a positive feedback OP Amp), where $R_1 = R_2 = 1 \text{ k}\Omega$, $R_3 = 2 \text{ k}\Omega$, and $C = 1 \mu\text{F}$. This has been run with the Analysis type of ‘Time Response (Transient)’ to yield the waveforms depicted in Figure 3.16(b). To your eyes, is the period of the voltage waveforms close to what you get from Equation (3.69)?

3.6 LRL Circuits and CRC Circuits

In this section an attempt is made to gain a better understanding of singular/degenerate circuits in which a switching operation may cause the continuity rules on inductor currents or capacitor voltages to be violated. With this purpose in mind, a start is made with an *LRL* circuit and a *CRC* circuit, where a resistor plays the role of arbitrator between two inductors or capacitors in conflict. These circuits may be regarded as practical models (with the large resistance of an open switch or some nonzero resistance of a wire taken into consideration) and at least are expected to present a clue to singular cases with no resistance. The law of flux linkage conservation and the law of charge conservation will also be derived; the former may help to deal with the case where the continuity rule of inductor current is violated and the latter may take care of the case where the continuity rule of capacitor voltage is violated. In some cases, consideration should be given on how to avoid such singular circuits, which may damage the system involved with them.

3.6.1 An LRL Circuit

Consider the *LRL* circuit of Figure 3.17(a) in which the current I_0 (supplied by the current source) has been flowing through only the inductor L_1 before $t = 0$ and the transfer switch is to be flipped to the left position so that an *LRL* circuit will be formed for $t \geq 0$. To analyze this circuit by using the node analysis, it is transformed by replacing the inductor L_1 (having the initial current I_0) with its s -domain equivalent (see Figure 3.6(b2)) having a current source as depicted in Figure 3.17(b). Then the node equation is set up and solved as follows:

$$V(s) = \frac{I_0/s}{1/R + 1/(sL_1) + 1/(sL_2)} = \frac{I_0/s}{1/R + 1/(sL_e)} = \frac{RI_0}{s + R/L_e} \quad (3.70a)$$

$$\left(\frac{1}{L_e} = \frac{1}{L_1} + \frac{1}{L_2}; L_e = \frac{L_1 L_2}{L_1 + L_2} \quad (\text{parallel combination of inductors}) \right)$$

$$v(t) = \mathcal{L}^{-1}\{V(s)\} \stackrel{\text{Table A.1(5)}}{=} RI_0 e^{-Rt/L_e} u_s(t) \quad (3.70b)$$

The currents through L_1 and L_2 can also be found as

$$I_{L_1}(s) = \frac{1}{sL_1} V(s) + \frac{i_{L_1}(0)}{s} \stackrel{(3.70a)}{=} \frac{1}{sL_1} \frac{RI_0}{s + R/L_e} - \frac{I_0}{s}$$

$$= \frac{I_0 L_e}{L_1} \frac{R/L_e}{s(s + R/L_e)} - \frac{I_0}{s} = \frac{I_0 L_2}{L_1 + L_2} \left(\frac{1}{s} - \frac{1}{s + R/L_e} \right) - \frac{I_0}{s} \quad (3.71a)$$

$$i_{L_1}(t) \stackrel{\text{Table A.1(3),(5)}}{=} I_0 \left[\frac{L_2}{L_1 + L_2} (1 - e^{-Rt/L_e}) - 1 \right] u_s(t) \xrightarrow[R \rightarrow \infty]{t \rightarrow \infty} I_{L_1} = -\frac{L_1 I_0}{L_1 + L_2} \quad (3.71b)$$

$$I_{L_2}(s) \stackrel{(3.15b)}{=} \frac{1}{sL_2} V(s) \stackrel{(3.70a)}{=} \frac{1}{sL_2} \frac{RI_0}{s + R/L_e} = \frac{I_0 L_e}{L_2} \frac{R/L_e}{s(s + R/L_e)} = \frac{I_0 L_1}{L_1 + L_2} \left(\frac{1}{s} - \frac{1}{s + R/L_e} \right) \quad (3.72a)$$

$$i_{L_2}(t) \stackrel{\text{Table A.1(3),(5)}}{=} \frac{I_0 L_1}{L_1 + L_2} (1 - e^{-Rt/L_e}) u_s(t) \xrightarrow[R \rightarrow \infty]{t \rightarrow \infty} I_{L_2} = \frac{L_1 I_0}{L_1 + L_2} \quad (3.72b)$$

As a consequence, the two inductors will have the flux linkages as

$$\lambda_1 \stackrel{(1.11)}{=} L_1 |I_{L_1}| \stackrel{(3.71b)}{=} L_1 \frac{L_1 I_0}{L_1 + L_2} = \frac{L_1}{L_1 + L_2} \lambda_0 \quad \text{with } \lambda_0 = L_1 I_0 \quad (3.73a)$$

$$\lambda_2 \stackrel{(1.11)}{=} L_2 |I_{L_2}| \stackrel{(3.72b)}{=} L_2 \frac{L_1 I_0}{L_1 + L_2} = \frac{L_2}{L_1 + L_2} \lambda_0 \quad (3.73b)$$

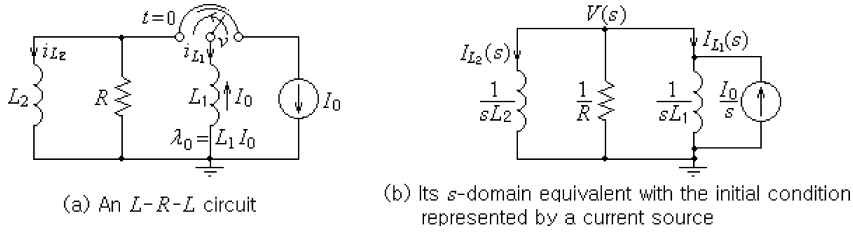


Figure 3.17 An LRL circuit and s -domain equivalent

The sum of the magnetic field energies stored in the two inductors is

$$W_{L_1} \stackrel{(1.16)}{=} \frac{1}{2} L_1 I_{L_1}^2 \stackrel{(3.71b)}{=} \frac{1}{2} L_1 \left(\frac{L_1 I_0}{L_1 + L_2} \right)^2, \quad W_{L_2} \stackrel{(1.16)}{=} \frac{1}{2} L_2 I_{L_2}^2 \stackrel{(3.72b)}{=} \frac{1}{2} L_2 \left(\frac{L_1 I_0}{L_1 + L_2} \right)^2$$

$$W_L = W_{L_1} + W_{L_2} = \frac{1}{2} \frac{L_1^2 I_0^2}{L_1 + L_2} \quad (3.74)$$

and the energy dissipated in the resistor is

$$W_R \stackrel{(1.9)}{=} \int_0^\infty \frac{v^2(t)}{R} dt \stackrel{(3.70b)}{=} R \int_0^\infty I_0^2 e^{-2Rt/L_e} dt$$

$$\stackrel{(F.33)}{=} \frac{R I_0^2}{-2R/L_e} e^{-2Rt/L_e} \Big|_0^\infty = \frac{1}{2} L_e I_0^2 = \frac{1}{2} \frac{L_1 L_2}{L_1 + L_2} I_0^2 \quad (3.75)$$

There are several observations to be made about what has been discussed above:

1. It is implied by Equations (3.70b), (3.71b), and (3.72b) that, eventually (if R is finite) or instantly (if $R = \infty$), the node voltage will be zero and the current flows not through R but only through the loop consisting of L_1 and L_2 . It is supported by the DC steady state condition that every inductor behaves like a short-circuit (Remark 3.2(2)).
2. The value of R does not matter in the above results and everything is only a matter of time.
3. Regardless of how fast the steady state is reached, the initial flux linkage $\lambda_0 = L_1 I_0$ is conserved in the steady state and then distributed between the two inductors in proportion to their inductances (see Equations (3.73a) and (3.73b)). This is based on the law of flux linkage conservation stated in Remark 3.1(2).
4. The sum of the magnetic field energies distributed in the two inductors and the energy dissipated in the resistor is equal to the magnetic energy that has been stored in the inductor L_1 before $t = 0$:

$$W_{L_1} + W_{L_2} + W_R \stackrel{(3.74),(3.75)}{=} \frac{1}{2} \frac{L_1^2 I_0^2}{L_1 + L_2} + \frac{1}{2} \frac{L_1 L_2 I_0^2}{L_1 + L_2} = \frac{1}{2} L_1 I_0^2 = \frac{1}{2L_1} \lambda_0^2$$

5. What if $R \rightarrow \infty$ (corresponding to the virtual nonexistence of R)? Since the inductor voltage $v(t) = R I_0 e^{-Rt/L_e} u_s(t)$ described by Equation (3.70b) has the height of $R I_0$ and the time constant of L_e/R , it will be like an impulse with an infinitely large magnitude and instantly short duration and can be modeled as $L_e I_0 \delta(t)$:

$$\int_{-\infty}^\infty v(t) dt = \int_0^\infty R I_0 e^{-Rt/L_e} dt = L_e I_0, \quad L_e I_0 \delta(t) \xrightarrow[\text{Table A.1(1)}]{\text{Laplace transform}} L_e I_0 = s L_e \frac{I_0}{s} \quad (3.76)$$

Note. Recall that the integration of the unit impulse function $\delta(t)$ from $-\infty$ to $+\infty$ is 1.

Note. Recall that the value of the s -domain equivalent voltage source representing the initial current I_0 of an inductor L_e is $L_e I_0$, as described in Figure 3.6(b1).

Note. The resistor placed between the two inductors seems to pose as an arbitrator who likes to resolve the conflict between one inductor L_2 desiring to maintain zero current and the other inductor L_1 trying to keep its initial current flow even through L_2 at the switching instant so that the continuity rule on the inductor currents will be observed.

3.6.2 A CRC Circuit

Consider the CRC circuit of Figure 3.18(a) in which only the capacitor C_1 has been charged with the initial voltage $-V_0$ (by the voltage source) before $t = 0$ and the transfer switch is to be flipped to the left position so that a CRC circuit will be formed for $t \geq 0$. To analyze this circuit using the mesh analysis, it is transformed by replacing the capacitor C_1 (having the initial voltage $-V_0$) with its s -domain equivalent (see Figure 3.6(c1)) having a voltage source as depicted in Figure 3.18(b). Then the mesh equation is set up and solved as follows:

$$I(s) = \frac{V_0/s}{R + 1/(sC_1) + 1/(sC_2)} = \frac{V_0/s}{R + 1/(sC_e)} = \frac{V_0/R}{s + 1/(RC_e)} \quad (3.77a)$$

$$\left(\frac{1}{C_e} = \frac{1}{C_1} + \frac{1}{C_2}; \quad C_e = \frac{C_1 C_2}{C_1 + C_2} \text{ (series combination of capacitors)} \right)$$

$$i(t) = \mathcal{L}^{-1}\{I(s)\} \stackrel{\text{Table A.1(5)}}{=} \frac{V_0}{R} e^{-t/(RC_e)} u_s(t) \quad (3.77b)$$

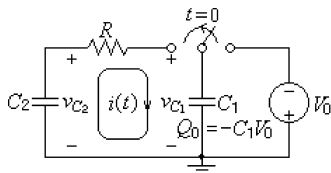
The voltages can also be found across C_1 and C_2 as

$$\begin{aligned} V_{C_1}(s) &\stackrel{(3.16b)}{=} \frac{1}{sC_1} I(s) + \frac{v_{C_1}(0)}{s} \stackrel{(3.77a)}{=} \frac{1}{sC_1} \frac{V_0/R}{s + 1/(RC_e)} - \frac{V_0}{s} \\ &= \frac{V_0 C_e}{C_1} \frac{1/(RC_e)}{s(s + 1/(RC_e))} - \frac{V_0}{s} = \frac{V_0 C_2}{C_1 + C_2} \left[\frac{1}{s} - \frac{1}{s + 1/(RC_e)} \right] - \frac{V_0}{s} \end{aligned} \quad (3.78a)$$

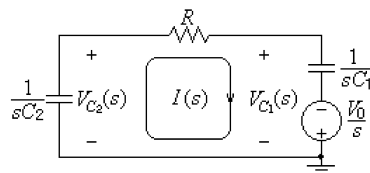
$$v_{C_1}(t) \stackrel{\text{Table A.1(3),(5)}}{=} V_0 \left[\frac{C_2}{C_1 + C_2} (1 - e^{-t/(RC_e)}) - 1 \right] u_s(t) \xrightarrow[R \rightarrow 0]{t \rightarrow \infty} V_{C_1} = -\frac{C_1 V_0}{C_1 + C_2} \quad (3.78b)$$

$$V_{C_2}(s) \stackrel{(3.16b)}{=} \frac{1}{sC_2} [-I(s)] \stackrel{(3.77a)}{=} -\frac{1}{sC_2} \frac{V_0/R}{s + 1/(RC_e)} = -\frac{V_0 C_e}{C_2} \left[\frac{1}{s} - \frac{1}{s + 1/(RC_e)} \right] \quad (3.79a)$$

$$v_{C_2}(t) \stackrel{\text{Table A.1(3),(5)}}{=} -\frac{V_0 C_1}{C_1 + C_2} (1 - e^{-t/(RC_e)}) u_s(t) \xrightarrow[R \rightarrow 0]{t \rightarrow \infty} V_{C_2} = -\frac{C_1 V_0}{C_1 + C_2} \quad (3.79b)$$



(a) A CRC circuit



(b) Its s -domain equivalent with the initial condition represented by a voltage source

Figure 3.18 A CRC circuit and its s -domain equivalent

As a consequence, the capacitors will have the charges

$$Q_1 \stackrel{(1.17)}{=} C_1 |V_{C_1}| \stackrel{(3.78b)}{=} C_1 \frac{C_1 V_0}{C_1 + C_2} = \frac{C_1}{C_1 + C_2} Q_0 \text{ with } Q_0 = C_1 V_0 \quad (3.80a)$$

$$Q_2 \stackrel{(1.17)}{=} C_2 |V_{C_2}| \stackrel{(3.79b)}{=} C_2 \frac{C_1 V_0}{C_1 + C_2} = \frac{C_2}{C_1 + C_2} Q_0 \quad (3.80b)$$

The sum of the electric field energies stored in the two capacitors is

$$W_{C_1} \stackrel{(1.21)}{=} \frac{1}{2} C_1 V_{C_1}^2 \stackrel{(3.78b)}{=} \frac{1}{2} C_1 \left(\frac{C_1 V_0}{C_1 + C_2} \right)^2, \quad W_{C_2} \stackrel{(1.21)}{=} \frac{1}{2} C_2 V_{C_2}^2 \stackrel{(3.79b)}{=} \frac{1}{2} C_2 \left(\frac{C_1 V_0}{C_1 + C_2} \right)^2$$

$$W_C = W_{C_1} + W_{C_2} = \frac{1}{2} \frac{C_1^2 V_0^2}{C_1 + C_2} \quad (3.81)$$

and the energy dissipated in the resistor is

$$W_R = \int_0^\infty R i^2(t) dt \stackrel{(3.77b)}{=} R \int_0^\infty \left(\frac{V_0}{R} \right)^2 e^{-2t/(RC_e)} dt$$

$$\stackrel{(F.33)}{=} \frac{V_0^2}{R} \frac{1}{-2/(RC_e)} e^{-2t/(RC_e)} \Big|_0^\infty = \frac{1}{2} C_e V_0^2 = \frac{1}{2} \frac{C_1 C_2}{C_1 + C_2} V_0^2 \quad (3.82)$$

There are several observations:

1. It is implied by Equations (3.77b), (3.78b), and (3.79b) that, eventually (if R is nonzero) or instantly (if $R = 0$), the current around the circuit will be cut off and nonzero voltages appear only across C_1 and C_2 . It is supported by the DC steady state condition that every capacitor behaves like an open circuit (Remark 3.4(2)).
2. The value of R does not matter in the above results and everything is only a matter of time.
3. Regardless of how fast the steady state is reached, the initial charge $Q_0 = C_1 V_0$ is conserved in the steady state and is distributed between the two capacitors in proportion to their capacitances (see Equations (3.80a) and (3.80b)). This may be better explained by the law of charge conservation stated in Remark 3.3(2).
4. The sum of the electric field energies distributed in the two capacitors and the energy dissipated in the resistor is equal to the electric field energy that has been stored in the capacitor C_1 before $t = 0$:

$$W_{C_1} + W_{C_2} + W_R \stackrel{(3.81),(3.82)}{=} \frac{1}{2} \frac{C_1^2 V_0^2}{C_1 + C_2} + \frac{1}{2} \frac{C_1 C_2 V_0^2}{C_1 + C_2} = \frac{1}{2} C_1 V_0^2 = \frac{1}{2 C_1} Q_0^2$$

5. What if $R \rightarrow 0$ (corresponding to the virtual nonexistence of R)? Since the capacitor current $i(t) = (V_0/R)e^{-t/(RC_e)}u_s(t)$ described by Equation (3.77b) has the height of V_0/R and the time constant of RC_e , it will be like an impulse with an infinitely large magnitude and instantly short duration and can be modeled as $C_e V_0 \delta(t)$:

$$\int_{-\infty}^\infty i(t) dt \stackrel{(3.77b)}{=} \int_0^\infty \frac{V_0}{R} e^{-t/(RC_e)} dt = C_e V_0; \quad C_e V_0 \delta(t) \xrightarrow[\text{Table A.1(1)}]{\text{Laplace transform}} C_e V_0 = \frac{V_0/s}{1/(sC_e)} \quad (3.83)$$

Note. Recall that the value of the s -domain equivalent current source representing the initial voltage V_0 of a capacitor C_e is $C_e V_0$, as described in Figure 3.6(c2).

Note. The resistor placed between the two capacitors seems to pose as an arbitrator that likes to resolve the conflict between one capacitor C_2 remaining at the voltage of zero before switching and the other capacitor C_1 trying to maintain its initial voltage at the switching instant so that the continuity rule on the capacitor voltages will not be violated.

3.6.3 Conservation of Flux Linkage and Charge

If $R = \infty$ in the *LRL* circuit of Figure 3.17(a) or if $R = 0$ in the *CRC* circuit of Figure 3.18(a), singular/degenerate circuits are formed. To take a broad view of the behavior of those circuits, the law of flux linkage conservation and the law of charge conservation need to be known. That is why they are now going to be derived.

The law of flux linkage conservation described by Equation (3.3) can be derived by applying KVL to the *LL* circuit (with the resistor opened for removal) to write

$$L_1 \frac{di_1(t)}{dt} + L_2 \frac{di_2(t)}{dt} = 0$$

Integrating both sides from $t = 0^-$ to $t = 0^+$ gives

$$\begin{aligned} \int_{0^-}^{0^+} L_1 \frac{di_1}{dt} dt + \int_{0^-}^{0^+} L_2 \frac{di_2}{dt} dt &= 0; \quad L_1[i_1(0^+) - i_1(0^-)] + L_2[i_2(0^+) - i_2(0^-)] = 0 \\ \lambda(0^+) &= L_1 i_1(0^+) + L_2 i_2(0^+) = L_1 i_1(0^-) + L_2 i_2(0^-) = \lambda(0^-) \end{aligned} \quad (3.84)$$

This can be extended into Equation (3.3) for the case of multiple inductors.

The law of charge conservation described by Equation (3.6) can be derived by applying KCL to the top node of the *CC* circuit (with the resistor shorted for removal) to write

$$C_1 \frac{dv_1(t)}{dt} + C_2 \frac{dv_2(t)}{dt} = 0$$

Integrating both sides from $t = 0^-$ to $t = 0^+$ gives

$$\begin{aligned} \int_{0^-}^{0^+} C_1 \frac{dv_1}{dt} dt + \int_{0^-}^{0^+} C_2 \frac{dv_2}{dt} dt &= 0; \quad C_1[v_1(0^+) - v_1(0^-)] + C_2[v_2(0^+) - v_2(0^-)] = 0 \\ Q(0^+) &= C_1 v_1(0^+) + C_2 v_2(0^+) = C_1 v_1(0^-) + C_2 v_2(0^-) = Q(0^-) \end{aligned} \quad (3.85)$$

This can be extended into Equation (3.6) for the case of multiple capacitors.

3.6.4 A Measure against Violation of the Continuity Rule on the Inductor Current

Consider the simple motor driving circuit in Figure 3.19 where a large current flows through the V_{CC} - R_C - Q (CE)- L loop when the switch is closed to turn on the transistor Q . Suppose the switch is opened at $t = 0$ to block off the inductor current. If the diode is not connected in parallel with the inductor, the switching-off operation will shortly induce a negative high voltage $v_L = L di_L/dt = -\infty(?)$ across the inductor, which is intensively applied to the terminals C and E of the transistor being turned off and eventually may damage the transistor Q . The diode connected in parallel with the inductor will provide a bypass for the inductor current, which wants to keep flowing continuously, guaranteeing its continuity and thus preventing a high voltage from being induced by switching off the inductor. The diode used for this purpose is called a *free-wheeling* diode.

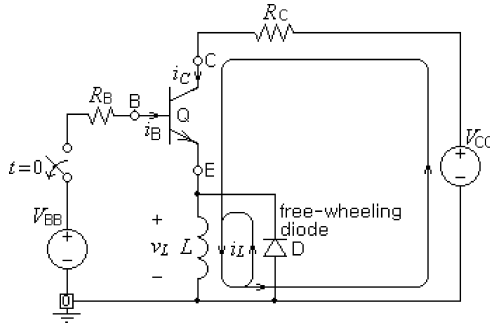


Figure 3.19 A circuit using a free-wheeling diode to allow the inductor current to flow continuously

3.7 Simulation Using PSpice and MATLAB

In this section PSpice is used to simulate an RC circuit excited by a DC sources and MATLAB as an auxiliary means for analyzing an RL circuit excited by an AC source.

3.7.1 An RC Circuit with Sequential Switching

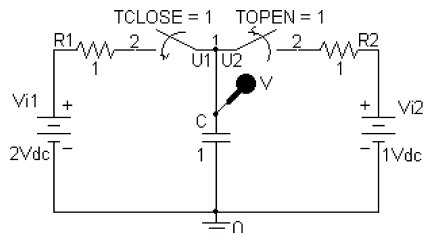
In order to simulate the RC circuit of Figure 3.12(a2), the parameters are set as

$$R_1 = 1\ \Omega, R_2 = 1\ \Omega, C = 1\ \text{F}, V_{i1} = 2\ \text{V}, V_{i2} = 1\ \text{V}, \text{ and } t_1 \text{ (second switching instant)} = 1\ \text{s}$$

Since PSpice, as version 10.0, does not have any part functioning as a transfer switch, we realize the transfer switch using a combination of two switches, 'Sw_tClose' and 'Sw_tOpen'. Also, since the switches presented by PSpice do not have a multiple switching function, the initial condition (designated as the parameter IC) of the capacitor is set to its initial voltage $v_C(0) = V_{i1} = 2\ \text{V}$ (possibly with the negative sign in consideration of the reference polarity of the capacitor) and the switching instants of the two switches commonly to $t_1 = 1\ \text{s}$. Then in the Edit Simulation Settings dialog box, the Analysis type and Run_to_time are set to 'Time Domain (Transient)' and 4 s, respectively. Figures 3.20(a), (b), and (c) show the PSpice schematic for the RC circuit, the Property Editor spreadsheets for setting the parameters of C , 'Sw_tClose', and 'Sw_tOpen', and the waveforms of the capacitor voltage/current obtained from the PSpice simulation, respectively. Do the simulation results agree with Figures 3.12(b2) and (c2) obtained by running the following MATLAB program `cir03_07_1.m`?

```
%cir03_07_1.m
% plots Fig. 3.12 (the capacitor voltage/current of an RC circuit)
clear, clf
Vi1=2; Vi2=1; R1=1; R2=1; L=1; C=1;
T= 0.01; % Sampling Interval
tf= 4; t_1= 1; % the final time and the 2nd switching instant
N= round(tf/T)+1; N1=round(t_1/T);
t1=[ 0:N1-1]*T; t2=[ N1:N-1]*T; % the intervals before/after t_1
% the capacitor voltage by Eq. (3.46)
vC1= (Vi1-Vi2)*exp(-t1/R2/C) +Vi2; vC1_2= (Vi1-Vi2)*exp(-t2/R2/C) +Vi2;
vC2= (Vi2-Vi1)*(1-exp(-t_1/R2/C))*exp(-(t2-t_1)/R1/C) + Vi1;
t=[ t1 t2]; vC=[ vC1 vC2];
subplot(222), plot(t,vC,'b', t2,vC1_2,'b:'), title('vC(t)')
```

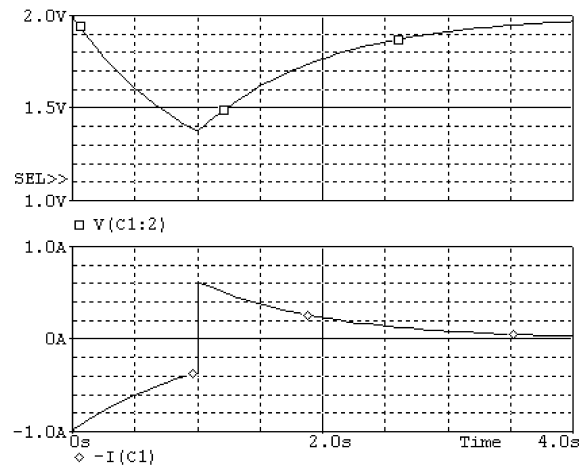
```
% the capacitor current by Eq. (3.47)
iC1= (Vi2-Vi1)/R2*exp(-t1/R2/C); iC1_2= (Vi2-Vi1)/R2*exp(-t2/R2/C);
iC2= (Vi1-Vi2)/R1*(1-exp(-t1/R2/C))*exp(-(t2-t1)/R1/C);
iC=[ iC1 iC2];
subplot(224), plot(t,iC,'b', t2,iC1_2,'b:'), title('iC(t)')
```



(a) The PSpice schematic for the RC circuit with sequential switching

Property Editor									
New Column... Apply Display... Delete Property Filter < Current properties > Help									
1	SCHEMATIC1:PAGE1: C	Color	Graphic	IC	Part Reference	Source	Package	Source Part	Value
		Default	C,Normal	-2	C		C	C,Normal	1
1	SCHEMATIC1:PAGE1: U1	Color	Part Reference	ROPEN	RCOLSE	Source Part	TCLOSE	TTRAN	Value
		Default	U1	1Meg	0,01	Sw_tClose,Normal	1	1u	Sw_tClose
1	SCHEMATIC1:PAGE1: U2	Color	Part Reference	ROPEN	RCOLSE	Source Part	TOPEN	TTRAN	Value
		Default	U2	1Meg	0,01	Sw_tOpen,Normal	1	1u	Sw_tOpen

(b) The Property Editor spreadsheet for setting the parameter values of C , Sw_tClose , and Sw_tOpen



(c) The waveforms of the capacitor voltage/current obtained from PSpice simulation

Figure 3.20 A PSpice simulation

3.7.2 An AC-Excited RL Circuit

In order to simulate the RL circuit that was analyzed in Section 3.4.5, the PSpice schematic is drawn as depicted in Figure 3.21(a) and the parameters are set as

$$R_1 = 1\ \Omega, \quad L = 1\ \text{H}, \quad \text{and} \quad t_f \text{ (run to time)} = 20\ \text{s}$$

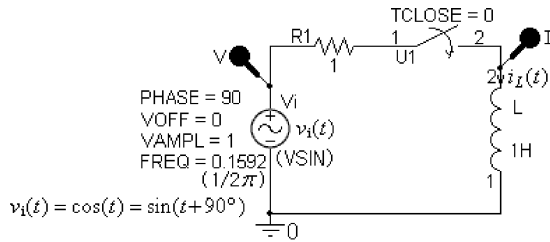
Note that PSpice has two AC voltage sources, ‘VSIN’ and ‘VAC’, which are used to find a time response for the time-domain analysis and a frequency response for the AC sweep analysis, respectively. Since the objective is to get the inductor current to a unity-amplitude sinusoidal voltage source $v_1(t) = \cos(t) = \sin(t + 90^\circ)$ for $0 \leq t \leq 20\ \text{s}$, ‘VSIN’ is used with the parameters set as

$$\text{VAMPL} = 1, \quad \text{FREQ}(f) = \omega/2\pi = 1/2\pi = 0.1592, \quad \text{PHASE} = 90, \quad \text{VOFF} = 0$$

Also in the Edit Simulation Settings dialog box, the Analysis type and Run_to_time are set to ‘Time Domain (Transient)’ and 20 s, respectively. Figures 3.21(a), (b), and (c1) show the PSpice schematic for the RL circuit, the Property Editor spreadsheet for setting the parameters of the sinusoidal voltage source ‘VSIN’, and the waveforms of the input voltage and the inductor current obtained from the PSpice simulation, respectively. Does the simulation result agree with Figure 3.13(a) or 3.21(c2) obtained by running the following MATLAB program `cir03_07_2.m`?

```
%cir03_07_2.m

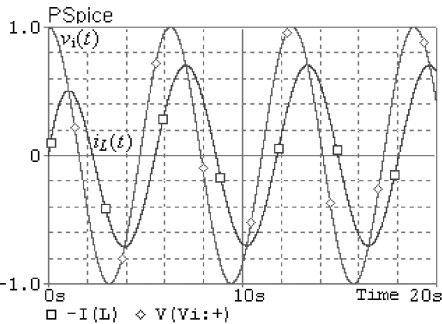
% plots Fig.3.13(a) (inductor current of an RL circuit with AC input)
clear, clf
R=1; L=1; % The resistance and inductance in the circuit
% the amplitude and angular frequency of the sinusoidal voltage source
Vim=1; w=1;
t0=0; tf=20; N=1000; % the initial/final time
t=t0+(tf-t0)/N*[ 0:N] ; % the whole time interval
syms s
Is = Vim/L/(s+R/L)*s/(s^2+w^2); % Eq. (3.49)
it= ilaplace(Is); pretty(it) % inverse Laplace transform
itt= eval(it); % the mesh/inductor current (complete/total response)
plot(t,itt), hold on
plot(t,Vim*cos(w*t), 'k') % the input voltage source waveform
%transient/steady-state responses from MATLAB
P=2*pi/w; % hopefully greater than 5 times the time constant L/R
t0 = t; t = t0+P;
itt_ss= eval(it); % steady-state responses surviving after one period
itt_tr= itt-itt_ss; % transient responses
t=t0; plot(t,itt_tr, 'r:', t,itt_ss, 'r'), hold on, pause
% With the analytical forms of transient/steady-state responses (3.51)
transient_response= -R*Vim/(R^2+(w*L)^2)*exp(-R*t/L);
steady_state_response=Vim/sqrt(R^2+(w*L)^2)*cos(w*t-atan2(w*L,R));
plot(t,transient_response, 'b:', t,steady_state_response, 'b')
```



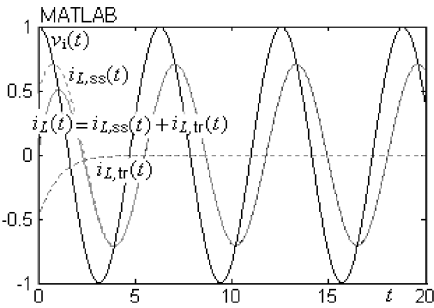
(a) The PSpice schematic for an AC-excited RL circuit

Property Editor											
New Column...		Apply	Display...	Delete Property	Filter	< Current properties >		Help			
1	SCHEMATIC1:PAGE1:V1	AC	DC	DF	FREQ	ID	TD	PHASE	VAMPL	VOFF	Part Reference Value
				0	0.1592		0	90	1	0	v1 VSIN

(b) The Property Editor spreadsheet for setting the parameter values of VSIN (refer to Fig. H.3(a1))



(c1) The PSpice simulation result



(c2) The MATLAB analysis result

Figure 3.21 PSpice simulation and MATLAB analysis for an RL circuit excited by a sinusoidal voltage source

3.8 Application and Design of First-Order Circuits

The following examples will help you to build up the circuit application and design point of view.

(Example 3.4) Applications of a 555 Timer with OP Amps Operating in the Nonlinear Region

Figures 3.22(a1) and (a2) show the two circuits that realize astable/monostable circuits, respectively, by using a 555 timer. Note the following about the 555 timer circuit:

1. Each of the two OP Amps having no feedback functions as a *comparator*, which exhibits a high/low output for the voltage at its + input terminal higher/lower than that at its – input terminal, respectively.
2. The three resistors of equal resistance R make a three-level voltage divider. Thus the voltage at the – input terminal of the comparator U1 is $2V_{CC}/3$ and the voltage at the + input terminal of the comparator U2 is $V_{CC}/3$, as long as the supply voltage V_{CC} is applied to the terminal +Vcc.
3. The flip-flop (FF) is set/reset to make its output high ($Q = \text{high}(\text{logic-1}), \bar{Q} = \text{low}(\text{logic-0})$) or low ($Q = \text{low}, \bar{Q} = \text{high}$) by raising the S or R input to high, while its output remains unchanged if the two R and S inputs are both low. It can also be reset by lowering the RESET input (active-low).

4. High/low $\bar{Q}$ applies to the base terminal of the NPN-transistor Q_1 to turn it on/off.
5. A high output of U1 resets the FF to make $Q = \text{low}$ and $\bar{Q} = \text{high}$, while a high output of U2 sets the FF to make $Q = \text{high}$ and $\bar{Q} = \text{low}$.

- (a) Figure 3.22(a1) shows an application of the 555 timer for an *astable* (free-running) circuit, whose output voltage is not stabilized but repeats periodic transitions between two opposite (high/low) states, appearing as a rectangular-wave generator. Find the period of the rectangular wave.

Suppose the supply voltage V_{CC} is applied to the terminal $+V_{CC}$ when the capacitor is uncharged, i.e. $v_C(0) = 0$. Then U2 with $0 = v_C = v_- < v_+ = V_{CC}/3$ sets the FF to make the output high ($Q = \text{high}$, $\bar{Q} = \text{low}$) and the low $\bar{Q}$ turns off the transistor Q_1 so that the capacitor will be charged (toward V_{CC}) through R_A - R_B from V_{CC} . This rising (charging) mode will continue until v_C goes above $2V_{CC}/3$ (the voltage at the $-$ input terminal of U1) to trigger U1 so that the high output of U1 resets the FF to make the output low ($Q = \text{low}$, $\bar{Q} = \text{high}$). Then the high $\bar{Q}$ turns on the transistor Q_1 so that the capacitor will be discharged (towards 0 V) through R_B - Q_1 to the ground (GND). This falling (discharging) mode will continue until v_C goes below $V_{CC}/3$ (the voltage at the $+$ input terminal of U2) to trigger U2 so that the high output of U2 sets the FF to make the output high again and another rising mode starts. Accordingly, the capacitor voltage v_C will repeat the cycle of going up to $2V_{CC}/3$ and down to $V_{CC}/3$. Noting that the charging/discharging time constants are $(R_A + R_B)C$ and $R_B C$, respectively, the formula (3.39) is used to obtain the period as

$$\text{Rising period: } \left(\frac{V_{CC}}{3} - V_{CC} \right) e^{-T_R / (R_A + R_B)C} + V_{CC} = \frac{2V_{CC}}{3}; \quad T_R = (R_A + R_B)C \ln 2 \quad (\text{E3.4.1})$$

$$\text{Falling period: } \frac{2V_{CC}}{3} e^{-T_F / R_B C} = \frac{V_{CC}}{3}; \quad T_F = R_B C \ln 2 \quad (\text{E3.4.2})$$

$$\text{Whole oscillation period: } P = T_R + T_F = (R_A + 2R_B)C \ln 2 \quad (\text{E3.4.3})$$

Then the duty cycle of the output rectangular wave turns out to be

$$\text{Duty cycle} = \frac{\text{high(on) time}}{\text{period}} = \frac{T_R}{T_R + T_F} = \frac{R_A + R_B}{R_A + 2R_B} \quad (\text{E3.4.4})$$

Figures 3.22(b1) and (d1) show the PSpice schematic and the simulation result for this circuit, respectively.

- (b) Figure 3.22(a2) shows another application of the 555 timer for a *monostable* (one-shot) circuit, whose output is normally stabilized at the low level but generates a single positive rectangular pulse of fixed duration when a short negative (triggering) pulse is applied to the TRIG terminal. Find the duration of the rectangular pulse.

Suppose the TRIG input is normally high and the FF is reset to make the output low ($Q = \text{low}$, $\bar{Q} = \text{high}$). Then the transistor Q_1 is turned on and the capacitor C is discharged through Q_1 to the ground (GND) so that $v_C(0) = 0$. This state is stably maintained until U1 is triggered by a low signal at its $-$ input terminal. Let a negative triggering pulse be applied to the terminal TRIG. Then U2 sets the FF to make the output high ($Q = \text{high}$, $\bar{Q} = \text{low}$) and the low $\bar{Q}$ turns off Q_1 so that C will be charged (towards V_{CC}) through R_A from V_{CC} . This rising (charging) mode will continue until v_C goes above $2V_{CC}/3$ (the voltage of the $-$ input terminal of U1) to trigger U1 so that the high output of U1 resets the FF to make the output low ($Q = \text{low}$, $\bar{Q} = \text{high}$) again and, subsequently, the high $\bar{Q}$ turning on Q_1 . Then C will be discharged and the low output level is maintained until another negative triggering pulse is applied to the TRIG terminal. Therefore the

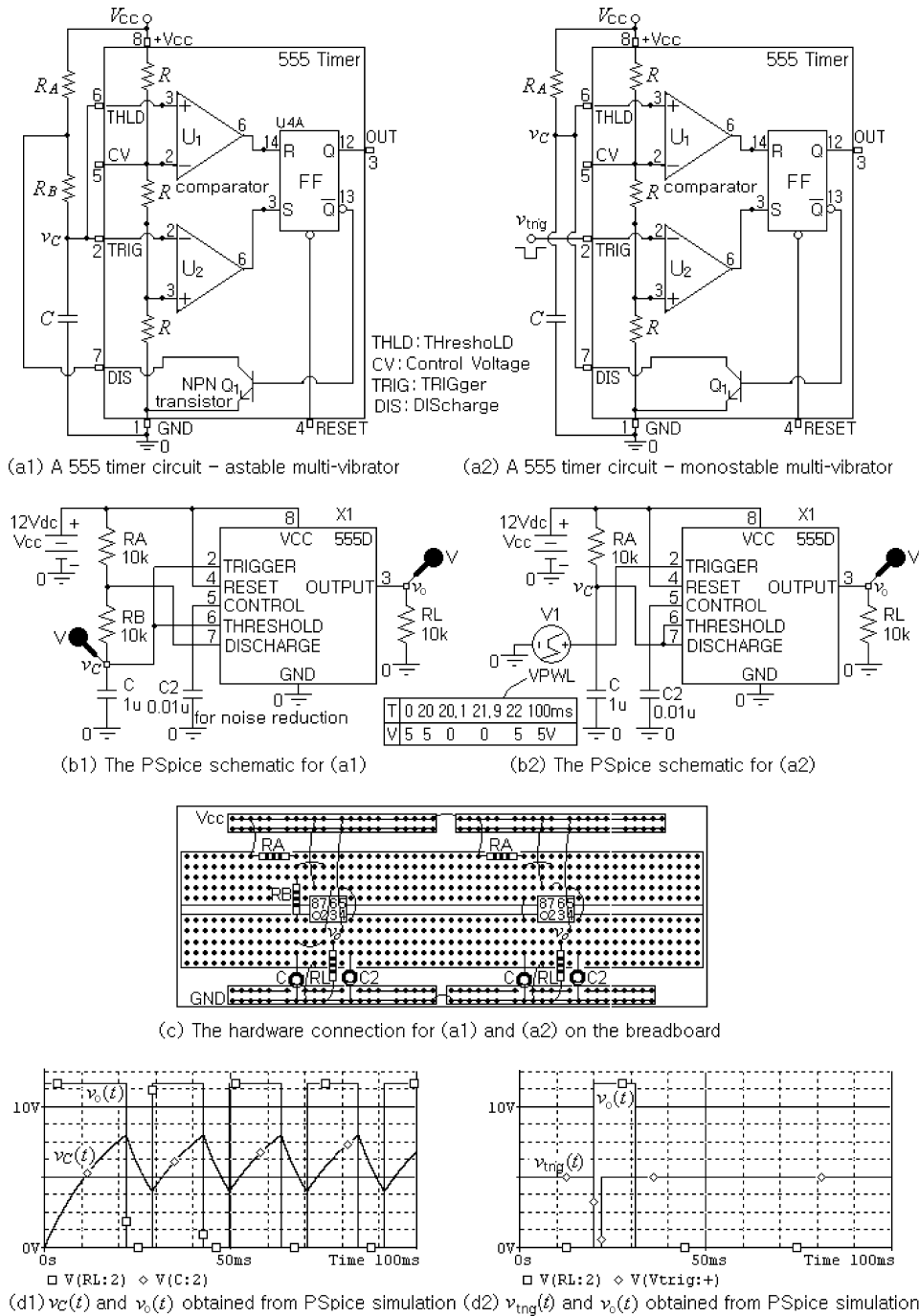


Figure 3.22 Applications of the 555 timer

output pulse duration is equal to the time taken for v_C to rise from 0 V to $2V_{CC}/3$, which is obtained by using the formula (3.39) as follows:

$$(0 - V_{CC})e^{-T_R/(R_A C)} + V_{CC} = \frac{2V_{CC}}{3}; \quad T_R = R_A C \ln 3 \text{ [s]} \quad (\text{E3.4.5})$$

Figures 3.22(b2) and (d2) show the PSpice schematic and the simulation results for this circuit, respectively.

Note. Visit the following websites to see more applications of the 555 timer:

<<http://www.uoguelph.ca/~antoon/gadgets/555/555.html>>
 <http://www.williamson-labs.com/480_555.htm>
 <<http://home.cogeco.ca/~rpaisley4/LM555.html>>

(Example 3.5) Design of an RL Circuit

Consider the RL circuit of Figure 3.23(a), which consists of a DC voltage source of $V_i = 10$ V, two resistors R_1 and R_2 , an inductor L of 10 mH, and a transfer switch. Choose the appropriate values of R_1 and R_2 among the standard resistance values with 1 % tolerance listed in Table G.2 (Appendix G).

- (a) Determine the value of R_1 such that, after the switch is flipped from b to position a at $t = 0$, the inductor current i_L will rise from the initial value of zero to more than 0.08 A within $t_1 = 0.1$ ms, with the time constant as short as possible.

Equation (3.39) can be used with $i_L(0) = 0$, $i_L(\infty) = V_i/R_1$, and $T = L/R_1$ to write the condition on the inductor current as

$$i_L(t_1) \stackrel{(3.39)}{=} \frac{V_i}{R_1} (1 - e^{-R_1 t_1/L}) \geq 0.08 \text{ A} \quad \text{for } t_1 = 0.0001 \text{ s} \quad (\text{E3.5.1})$$

In order to find the boundary value of R_1 narrowly satisfying this inequality, the nonlinear equation solver `fsolve()` of MATLAB can be used as follows:

```
>>Vi=10; L=0.01; t1=0.0001; ILlmin=0.08;
>>fR=inline('V*(1-exp(-R*t/L))./R-ILl','R','V','L','t','ILl');
>>R1_0=1; R1_d=fsolve(fR,R1_0,[],Vi,L,t1,ILlmin)
R1_d = 45.1740
```

We type these statements into the MATLAB command window to obtain the desired value of R_1 as 45.2 Ω and choose 44.2 Ω from Table G.2, which is just smaller than the desired value. Why not use a greater one rather than a smaller one? Because the value of the function on the left side of the inequality (E3.5.1) increases as R_1 gets smaller, as can be seen from its plot for $R_1 = 0 \sim 100 \Omega$ (Figure 3.23(b)) obtained by typing the following statements into the command window:

```
>>RR=[0:200]/2; iL=fR(RR,Vi,L,t1,0); %iL=Vi*(1-exp(-RR*t1/L))./RR;
>>plot(RR,iL,[RR(1)RR(end)],ILlmin*[1 1],':')
```

- (b) With R_1 fixed as 44.2 Ω , determine the value of R_2 such that, after the switch is flipped from position a to position b at $t = 0$ when the circuit with the switch connected to position a has reached a steady state, the inductor current i_L will not be smaller than 0.1 A until $t_2 = 0.1$ ms, and decrease with the time constant as short as possible.

With $i_L(0) = V_i/R_1$, $i_L(\infty) = 0$, and $T = L/(R_1 + R_2)$, Equation (3.39) is used to write the condition on the inductor current as

$$i_L(t) \stackrel{(3.39)}{=} \frac{V_i}{R_1} e^{-(R_1+R_2)t/L} \geq 0.1 \text{ A} \quad \text{for } t_2 = 0.0001 \text{ s} \quad (\text{E3.5.2})$$

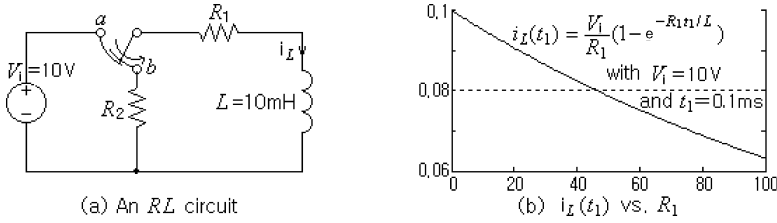


Figure 3.23 A circuit design of Example 3.5

Thus MATLAB can be used to obtain

$$R_1 + R_2 \leq \frac{L}{t_2} \ln \frac{V_1/R_1}{0.1} = \frac{0.01}{0.0001} \ln \frac{10/44.2}{0.1} = 81.6445; \quad R_2 \leq 81.6445 - R_1 = 37.4445 \, \Omega \quad (\text{E3.5.3})$$

```
>>R1=44.2; t2=0.0001; IL2max=0.1; R2_d= log(V1/R1/IL2max)/t2*L - R1
R2_d = 37.4445
```

Finally, $37.4 \, \Omega$ is chosen from Table G.2, which is just smaller than the desired value.

(Example 3.6) MATLAB-Based Design and Simulation of an RC Circuit for a Flashing Lamp

Consider the RC circuit of Figure 3.24(a), which consists of a DC voltage source of $V_1 = 50 \text{ V}$, a resistor R , a capacitor C , and a lamp. Once the lamp starts to conduct at the voltage of 40 V or higher, it continues to be on until its voltage becomes lower than 10 V . The lamp can be modeled as a resistance of $R_L = 1 \text{ k}\Omega$ when it is on and as open when it is off.

- (a) Choose the value of C among the standard capacitance values listed in Table G.3 in Appendix G, such that the energy of 0.0075 J can be stored in the capacitor while its voltage rises from 10 V to 40 V :

$$E_C \stackrel{(1.21)}{=} \frac{1}{2} C v_2^2 - \frac{1}{2} C v_1^2 = \frac{1}{2} C (40^2 - 10^2) = 750 C = 0.0075 \text{ J}; \quad C = 10 \, \mu\text{F} \quad (\text{E3.6.1})$$

Since $10 \, \mu\text{F}$ is one of the standard capacitance values, we choose $C = 10 \, \mu\text{F}$.

- (b) With C fixed as $10 \, \mu\text{F}$, choose the appropriate values of R among the standard resistance values with 5% tolerance listed in Table G.2 (Appendix G) such that the lamp flashes about 7 times for one second.

For the rising period during which the capacitor–lamp voltage $v_C(t)$ rises from $V_1 = 10 \text{ V}$ to $V_2 = 40 \text{ V}$ (Figure 3.24(b)), the formula (3.39) can be used to write the capacitor voltage as

$$v_C(t) \stackrel{(3.39)}{=} (V_1 - V_\infty) e^{-t/(RC)} + V_\infty \quad \text{with} \quad V_\infty = V_i = 50 \text{ V} \quad (\text{E3.6.2})$$

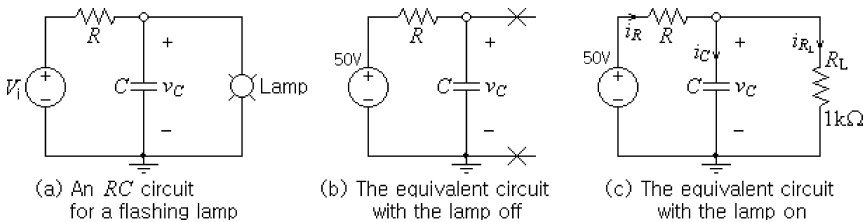


Figure 3.24 RC circuit of a flashing lamp

so that the length of the rising period can be obtained as

$$(V_1 - V_\infty)e^{-T_R/(RC)} + V_\infty = V_2; \quad T_R = RC \ln \frac{V_i - V_1}{V_i - V_2} = RC \ln 4 [s] \quad (\text{E3.6.3})$$

Also, for the falling period during which the capacitor–lamp voltage $v_C(t)$ falls from 40 V to 10 V (Figure 3.24(c)), the formula (3.39) can be used to write the capacitor voltage as

$$v_C(t) \stackrel{(3.39)}{=} (V_2 - V_\infty)e^{-t/(R_e C)} + V_\infty \quad \text{with} \quad V_\infty = \frac{R_L}{R + R_L} V_i \quad \text{and} \quad R_e = \frac{RR_L}{R + R_L} \quad (\text{E3.6.4})$$

so that the length of the falling period can be obtained as

$$(V_2 - V_\infty)e^{-T_F/(R_e C)} + V_\infty = V_1; \quad T_F = R_e C \ln \frac{V_2 - V_\infty}{V_1 - V_\infty} \quad (\text{E3.6.5})$$

Thus the condition on the whole period can be written as

$$P = T_R + T_F = RC \ln \frac{V_i - V_1}{V_i - V_2} + R_e C \ln \frac{V_2 - V_\infty}{V_1 - V_\infty} = \frac{1}{7} \text{ s} \quad (\text{E3.6.6})$$

In order to find the value of R satisfying this condition, the following MATLAB program `cir03e06.m` is composed and run to get the desired value as

```
>>cir03e06
    R_d = 9.0435e+003
```

Thus $8200 \, \Omega$ is chosen from Table G.2, which is just smaller than the desired value.

Note. The following points about the program `cir03e06.m` should be noted:

1. The nonlinear equation solver `newtons()` listed in Appendix D seems to work as well as the MATLAB built-in function `fsolve()`.
2. The nonlinear equation (E3.6.6) is defined in an M-file `fR_cir03e06.m`.

```
%cir03e06.m
clear, clf
Vi=50; R=9000; RL=1000; C=1e-5; V1=10; V2=40;
P_d=1/7 %Desired period
R_0=1; %Initial guess on R
TolX=0.001; MaxIter=100; % Error tolerance, Maximum # of iterations
R_d=newtons('fR_cir03e06', R_0, TolX, MaxIter, RL, C, Vi, V1, V2, P_d)
%Does fsolve() work with better initial guess?
R_0=100; option=optimset('TolFun', 1e-9);
R_d1=fsolve('fR_cir03e06', R_0, option, RL, C, Vi, V1, V2, P_d)

function y=fR_cir03e06(R, RL, C, Vi, V1, V2, P_d)
Re=R*RL./(R+RL); Vf=RL./(R+RL)*Vi;
y= R*C*log((Vi-V1)/(Vi-V2))+Re*C*log((V2-Vf)/(V1-Vf))-P_d; % (E3.6.6)
```

- (c) The following simulation program `cir03e06c.m` is composed to simulate the circuit with $R = 8200\ \Omega$ and $C = 10\ \mu\text{F}$ and run to obtain Figure 3.25, which shows the capacitor voltage and the lamp current for 1 s.

Note the following points about the program:

1. It defines the conductance of the lamp as

$$G_L = 1/R_L = \begin{cases} 1/1000 & \text{if } v > 40 \text{ or if } i > 0(\text{on}) \text{ and } v \geq 10 \\ 0 & \text{otherwise} \end{cases} \quad (\text{E3.6.7})$$

2. It uses the discretized version of the circuit equation with the sampling period T as

$$v_C(t+T) = v_C(t) + \frac{1}{C} \int_t^{t+T} i_C(\tau) d\tau = v_C(t) + \frac{1}{C} \int_t^{t+T} [i_R(\tau) - i_{R_L}(\tau)] d\tau \quad (\text{E3.6.8})$$

$$v_C(t+T) = v_C(t) + \frac{1}{C} \left[\frac{V_i - v_C(t)}{R} - G_L v_{R_L}(t) \right] T \quad \text{with } v_{R_L}(t) = v_C(t) \quad (\text{E3.6.9})$$

```
%cir03e06c.m
% Simulation of the flash lamp driver with R=8200 (with 5% tolerance)
R=standard_value(R_d, 'R', '1', 5) % Appendix G
% R=8200 chosen as a 5%-tolerance standard resistance less than R_d
T=0.001; % Sampling Interval
N=1000; tt=[0:N]*T; % Time duration for simulation
GL= inline('(v>40|(i>eps&v>=10))*(1/1000)', 'v', 'i'); % (E3.6.7)
iRL(1)=0; vC(1)=0;
for n=1:N
    iRL(n+1)=vC(n)*GL(vC(n), iRL(n)); % Lamp current from Ohm's law
    iC=(Vi-vC(n))/R-iRL(n+1); % Capacitor current from KCL at top node
    vC(n+1)=vC(n)+iC*T/C; % Capacitor voltage Eq. (3.4b) & (E3.6.9)
end
subplot(211), plot(tt, vC), title('Capacitor voltage vC(t)')
subplot(212), plot(tt, iRL, 'r'), title('Lamp current iRL(t)')
```

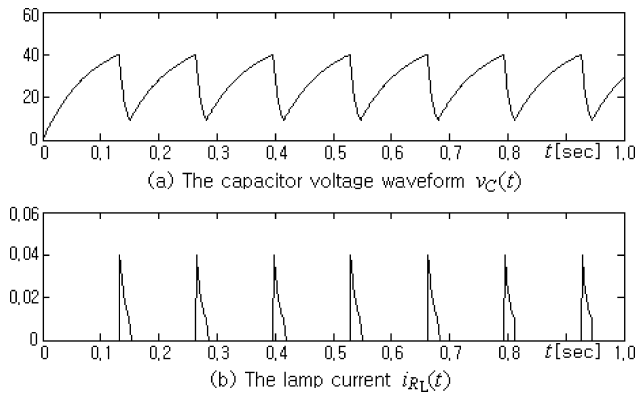


Figure 3.25 The simulation results for the flashing lamp circuit depicted in Figure 3.24

Problems

3.1 Switching of an RL Circuit for Relay Control

Consider the relay control circuit of Figure P3.1 in which the relay contact is connected or disconnected depending on the inductor current $i_L(t)$; i.e. it is connected for $i_L(t) \geq 4$ A, is disconnected for $i_L(t) < 1$ A, and keeps the current state for $1 \text{ A} \leq i_L(t) < 4$ A. The resistance of the diode is $R_{D,ON} = 1 \Omega$ when it is on and is $R_{D,OFF} = \infty$ when off.

- Write the expression for the inductor current $i_L(t)$ for $t \geq 0$ s when the switch S is closed at $t = 0$ s after it has been open for a long time and find the time taken for the relay contact to be closed.
- Write the expression for $i_L(t)$ for $t \geq 0$ s when the switch S is opened at $t = 0$ s after it has been closed for a long time and find the time taken for the relay contact to be opened.
- Explain the role of the diode on the continuity of the inductor current.

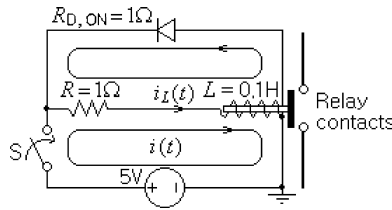


Figure P3.1 An RL circuit for relay control

3.2 Periodic Switching of an RL Circuit

Consider the circuit of Figure P3.2 where the switch is on for T_{ON} and off for T_{OFF} every $P = T_{ON} + T_{OFF}$ seconds. Assume that the circuit has reached the steady state in which the inductor current i keeps going up to I_2 and down back to I_1 with the period of $P = T_{ON} + T_{OFF}$.

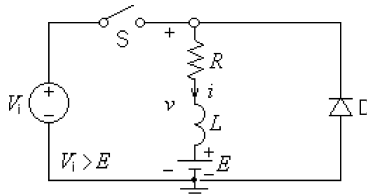


Figure P3.2 A chopper for an RL load

- Show that the average current I_a and the peak-to-peak ripple current $\Delta I = I_2 - I_1$ are as follows:

$$I_a = \frac{V_i - E}{2R} \frac{(1 - e^{-T_{ON}/T})(1 + e^{-T_{OFF}/T})}{1 - e^{-(T_{ON} + T_{OFF})/T}} \stackrel{T=L/R}{=} \frac{V_i - E}{2R} \frac{(1 - e^{-T_{ON}R/L})(1 + e^{-T_{OFF}R/L})}{1 - e^{-(T_{ON} + T_{OFF})R/L}} \quad (\text{P3.2.1})$$

$$\Delta I = \frac{V_i - E}{R} \frac{(1 - e^{-T_{ON}/T})(1 - e^{-T_{OFF}/T})}{1 - e^{-(T_{ON} + T_{OFF})/T}} \stackrel{T=L/R}{=} \frac{V_i - E}{R} \frac{(1 - e^{-T_{ON}R/L})(1 - e^{-T_{OFF}R/L})}{1 - e^{-(T_{ON} + T_{OFF})R/L}} \quad (\text{P3.2.2})$$

- (b) Show that the ripple expressed by Equation (P3.2.2) is maximized by the following duty cycle:

$$d = \frac{T_{\text{ON}}}{P} = \frac{T_{\text{ON}}}{T_{\text{ON}} + T_{\text{OFF}}} = \frac{1}{2} \quad (\text{P3.2.3})$$

3.3 The Effect of Switch Capacitance

Figure P3.3 shows a model of a nonideal switch, which represents the effect of its switch capacitance C , on-resistance R_{ON} , and off-resistance R_{OFF} on switching on/off a load resistance R_L .

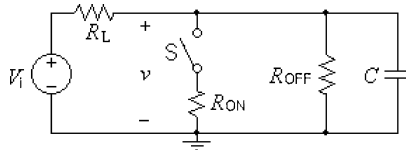


Figure P3.3 A model of a nonideal switch for representing the effect of its switch capacitance on switching

- Write the expression for the capacitor voltage $v(t)$ for $t \geq 0$ s when the switch S is closed at $t = 0$ s after it has been open for a long time. Assume $R_{\text{OFF}} \gg R_{\text{ON}}$ so that $R_{\text{ON}} \parallel R_{\text{OFF}} \cong R_{\text{ON}}$.
- Write the expression for the capacitor voltage $v(t)$ for $t \geq 0$ s when the switch S is opened at $t = 0$ s after it has been closed for a long time.

3.4 An RC Circuit

Consider the RC circuit of Figure P3.4.

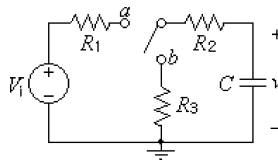


Figure P3.4

- Write the expression for the capacitor voltage $v(t)$ for $t \geq 0$ s when the switch S is flipped to position a at $t = 0$ s after it has been connected to position b for a long time.
- Write the expression for the capacitor voltage $v(t)$ for $t \geq 0$ s when the switch S is flipped to position b at $t = 0$ s after it has been connected to position a for a long time.

3.5 The Effect of Resistance/Capacitance of a Cable on Maximum Interconnect Length

Figure P3.5(a) shows a PC (personal computer) connected with a printer via a cable. Since a unit length of the cable consisting of two twisted conducting wires has a (distributed) capacitance of $c = 80$ pF/m as well as a (distributed) resistance of $r = 0.5$ Ω /m, the cable between the PC and the printer is modeled for analysis as depicted in Figure P3.5(b), even though it has neither a physical (lumped) resistor nor a capacitor.

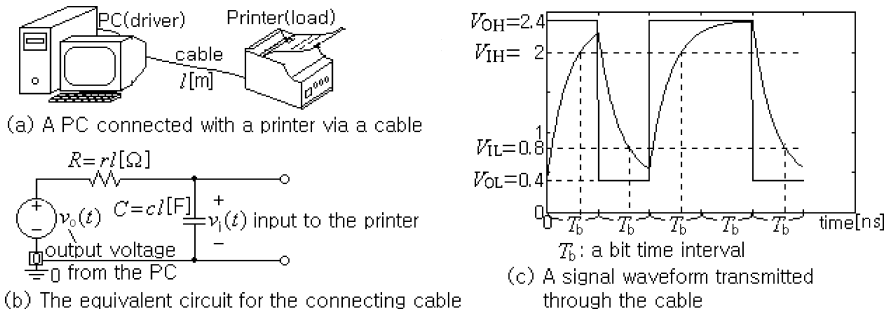


Figure P3.5

According to the TTL logic used for data communication between the PC and the printer, the transmitter outputs $V_{OH}/V_{OL} = 2.4\text{ V}/0.4\text{ V}$ to send a logical signal 1(high)/0(low), respectively, and the receiver recognizes the input voltage $v_i = V_{IH}$ (higher than 2.0 V)/ V_{IL} (lower than 0.8 V) as 1(high)/0(low), respectively.

- Assuming that a rectangular wave going up to V_{OH} and down to V_{OL} is sent periodically to the printer via the cable, find the expressions of the typical signal waveform arriving at the printer input terminal for the rising/falling intervals in terms of V_{OH} , V_{OL} , R , and C .
- Find the time T_H taken for $v_i(t)$ to rise from V_{OL} to V_{IH} and the time T_L taken for $v_i(t)$ to fall from V_{OH} to V_{IL} in terms of R and C .
- The data rate is required to be higher than $8\text{ megabits/s} = 8 \times 10^6\text{ bits/s}$, meaning the transmission time less than $(1/8) \times 10^{-6}\text{ s} = 125\text{ ns}$ per bit. Find the maximum length of the cable such that the longer one of T_H and T_L is not greater than 100 ns .

3.6 A Half-Wave Rectifier Using a Diode and a Capacitor

Figure P3.6(a) shows a half-wave rectifier using a capacitor and a diode that conducts in the forward direction for $v_D \geq V_{TD} = 0.65\text{ V}$. Figures P3.6(b) and (c) show the equivalent circuits of the rectifier for $v_s \geq v_o + V_{TD}$ and $v_s < v_o + V_{TD}$. It can be seen from the PSpice simulation result depicted in Figure P3.6(d) that the output voltage v_o follows the input voltage $v_s(t) = V_m \sin(\omega t) = 5 \sin(2\pi f t)$ ($f = 60\text{ Hz}$) promptly when rising up, but very lazily when falling down, which is helpful for making the rectifier output $v_o(t)$ smooth with a small ripple.

- Why are the behaviors of the circuit different for the two cases of the capacitor being charged and discharged?

Hint. Determine the time constants of the equivalent circuits in Figures P3.6(b) and (c).

- In order to get the upper/lower limit V_H/V_L of the output voltage $v_o(t)$ and the rising/falling period T_R/T_F , it seems that the following equations need to be set up:

$$(V_L - V_m)e^{-T_R/(R_f C)} + V_m - V_H = 0 \quad (\text{P3.6.1})$$

$$V_H e^{-T_F/(RC)} - V_L = 0 \quad (\text{P3.6.2})$$

$$V_H - V_L - V_m(1 - \cos \omega T_R) = 0 \quad (\text{P3.6.3})$$

$$\omega(T_R + T_F) = 2\pi; \quad T_R + T_F - 1/f = 0 \quad (\text{P3.6.4})$$

However, one of these equations is absurd. Excluding it and noting that $V_H = V_m - V_{TD} = 5 - 0.65 = 4.35$, solve the three remaining equations to find V_L , T_R , and T_F . You can use the following MATLAB program `cir03p06.m` after saving these equations in an M-file named, say, `f_cir03p06.m`.

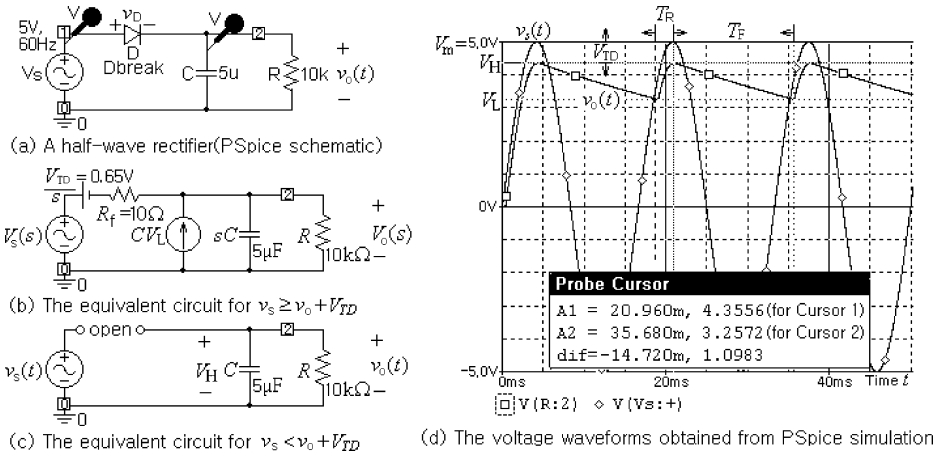


Figure P3.6 The equivalent circuits of a half-wave rectifier and its input/output voltage waveforms

```
%cir03p06.m
global Vm f VTD
Vm=5; f=60; VTD=0.65; VH=Vm-VTD; R=1e4; Rf=10; C=5e-6;
x_0=[0 0 0]; x=fsolve('f_cir03p06',x_0,optimset('fsolve'),C,R)

function y=f_cir03p06(x,C,R)
global Vm f VTD
VH=Vm-VTD; w=2*pi*f; T=1/f;
VL=x(1); TR=x(2); TF=x(3);
y=[VH*exp(-TF/R/C)-VL; VH-VL-Vm*(1-cos(w*TR)); TR+TF-T]; % (P3.6.2-4)
```

- (c) Use PSpice to simulate the rectifier and find the upper/lower limit V_H/V_L of the output voltage $v_o(t)$ and the rising/falling period T_R/T_F . After getting the waveforms in the Probe window, click the Toggle Cursor button on the toolbar to activate the two cross-type cursors on the graph. Then use the left/right mouse button and/or arrow/shift-arrow key or click the appropriate toolbar button to move them to the maximum, minimum, peak, or trough and read their coordinates from the Probe Cursor box. If there are two or more waveforms on the Probe window, choose one that you want to take a close look at by clicking the name of the corresponding variable under the graph. Are they similar to those obtained in (b)?
- (d) Only those who are eager to obtain the expression for the output voltage waveform $v_o(t)$ are welcome to set up the node equation for the equivalent circuit in Figure P3.6(b) as

$$\left(sC + \frac{1}{R_f} + \frac{1}{R}\right)V_o(s) = \frac{1}{R_f}\left[V_s(s) - \frac{V_{TD}}{s}\right] + CV_L \quad (\text{P3.6.5})$$

with $V_s(s) = \mathcal{L}\{\cos[\omega(t - T_R)]\} \stackrel{(\text{F.6})}{=} \mathcal{L}[\cos(\omega t) \cos(\omega T_R) + \sin(\omega t) \sin(\omega T_R)]$

$$\stackrel{\text{Table A.1(7),(8)}}{=} \frac{s \cos(\omega T_R) + \omega \sin(\omega T_R)}{s^2 + \omega^2} \quad (\text{P3.6.6})$$

where the voltage source of $v_s(t) = V_m \cos(\omega t)$ applied from $t = -T_R$ has been regarded as $v_s(t) = V_m \cos[\omega(t - T_R)]$ applied from $t = 0$ and the initial capacitor voltage of V_L at the start time of the rising period is transformed into its equivalent current source of CV_L . Solve this equation to get $V_o(s)$ and take its inverse Laplace transform to get $v_o(t)$ for the rising period $0 \leq t \leq T_R$. You may use the following MATLAB program `cir03p06d.m`. In fact, since the time constant $(R_f \parallel R)C \simeq R_f C = 5 \times 10^{-5}$ s is very short, the transient response will disappear like a flash and so $v_o(t)$ will instantly follow the input source waveform only with a gap of $V_{TD} = 0.65$ V.

```
%cir03p06d.m
%You should have the rising period TR (in (b)) to use this program.
w=2*pi*f; wTR=w*TR;
syms s
% Laplace transform of vs(t)=Vm*cos(wt-wTR)
Vss=Vm*(cos(wTR)*s+sin(wTR)*w)/(s^2+w^2); % (P3.6.6)
Vrs=((Vss-VTD/s)/Rf+C*VL)/(C*s+1/Rf+1/R) % (P3.6.5)
vrt=ilaplace(Vrs); % vo(t) during rising time interval
pretty(vrt)

%cir03p06e.m
% Plot the rectified voltage waveform for one period T=1/f
T=1/f; TF=T-TR; Ts=T/400; t1=[0:Ts:TF]; t2=[TF:Ts:T]; tt=[t1 t2];
vot1=VH*exp(-t1/R/C); % vo(t) during the falling period
t=t2-t2(1); vot2=eval(vrt); % vo(t) for the rising period
vot=[vot1 vot2]; vs=Vm*cos(w*tt); % the sinusoidal voltage source
plot(tt,vs,tt,vot,'r',t2,Vm*cos(w*(t-TR))-VTD,'m')
```

- (e) Referring to the equivalent circuit in Figure P3.6(c), find the output voltage waveform $v_o(t)$ for the falling period and plot it together with that (for the rising period) obtained in (d) for the whole period $T = 1/f$. The above MATLAB program `cir03p06e.m` can be used. Is it similar to one period of the waveform obtained from the PSpice simulation (Figure P3.6(d))?

Note. This kind of circuit can be used not only for rectifying an AC voltage into a DC voltage, but also for demodulating a conventional AM (amplitude modulated) signal to get the message signal.

3.7 An RC Debouncer

Bouncing is defined as the tendency of any two metal contacts in an electronic device to generate multiple signals as the contacts close or open and *debouncing* is any kind of hardware or software measure ensuring that only a single (clean) signal will be acted upon for a single opening or closing of a contact (Reference [W-6]). Figure P3.7(a) shows a switching circuit, which is used to reset a microprocessor, where the bouncing (chattering) effect of the switch contact is supposed to be alleviated by the RC circuit before the resulting ripple signal is taken care of by the Schmitt trigger (see Example 2.32). Figure 3.7(b) shows the typical waveform of the input voltage $v_C(t)$ to the Schmitt trigger when the switch is on and off. Find the time constants of this circuit for the cases where the switch is on/off and explain why $v_C(t)$ falls down towards V_L more rapidly than it rises up towards V_H .

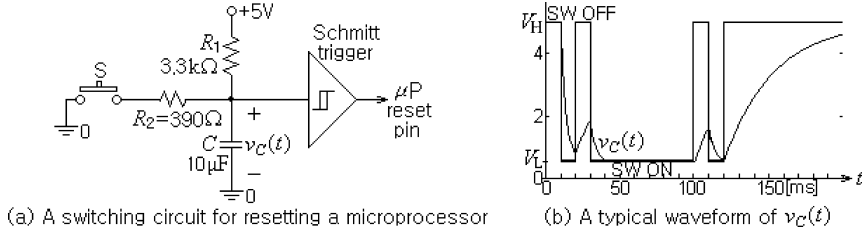


Figure P3.7

3.8 The Response of an RC Circuit to a Square Wave

Consider the RC circuit of Figure P3.8(a) where the voltage level generated by the square-wave voltage source is $v_s(t) = V_2/V_1[V]$ for $T_H/T_L[s]$ every $P = T_H + T_L[s]$, as depicted in Figure P3.8(b).

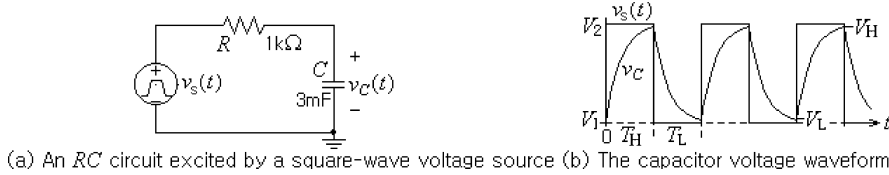


Figure P3.8

- (a) Express the steady state capacitor voltage $v_C(t)$ for the rising/falling periods in terms of V_1 , V_2 , T_H , T_L , R , and C . Find the expressions of its upper/lower limits V_H and V_L as

$$\begin{bmatrix} V_H \\ V_L \end{bmatrix} = \frac{1}{1 - e^{-(T_H+T_L)/T}} \begin{bmatrix} (1 - e^{-T_H/T})V_2 + e^{-T_H/T}(1 - e^{-T_L/T})V_1 \\ (1 - e^{-T_L/T})V_1 + e^{-T_L/T}(1 - e^{-T_H/T})V_2 \end{bmatrix} \quad \text{with } T = RC \quad (\text{P3.8.1})$$

- (b) Find the expressions for the high time T_H and low time T_L of the square-wave input such that the upper/lower limits of the steady state capacitor voltage $v_C(t)$ become V_H and V_L , respectively.

3.9 ‘Virtually Parallel’ Capacitors and Equivalent Circuits for Two Interconnected CMOS Gates

- (a) Consider the RC circuit of Figure P3.9(a1), which is excited by a square-wave source and a constant (DC) one. Figure P3.9(a2) shows its s -domain (transformed) equivalent with the two voltage sources replaced by their equivalent current sources. Find the value of the equivalent current source for the DC voltage source. Show that the voltage $v_{C1}(t)$ across the capacitor C_1 due to that current source is

$$v_{C1}(t)|_{\text{due to } V_s} = \frac{C_2}{C_1 + C_2} e^{-t/[R_1(C_1+C_2)]} V_s u_s(t) \quad (\text{P3.9.1})$$

This implies that, as time goes by, the effect of the DC voltage source fades away so that the two capacitors can be regarded as connected virtually in parallel. To verify this implication, perform

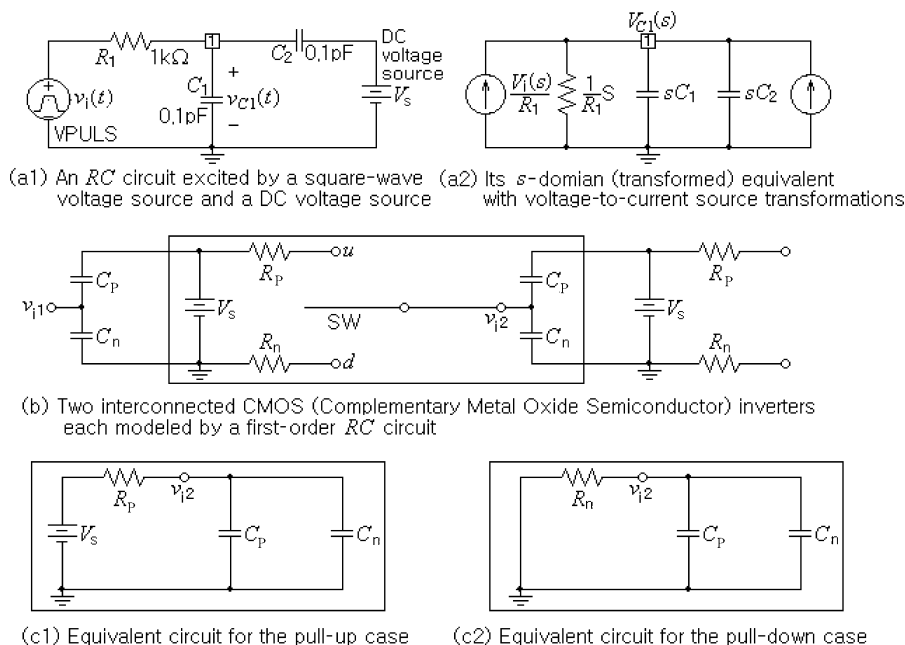


Figure P3.9

the PSpice simulation for 5 ns (with a maximum step of, say, 1 ps) twice, once with $V_s = 5$ V and once with $V_s = 0$ V, and compare the waveforms of $v_{C1}(t)$, where the square-wave voltage source is represented by the PSpice part 'VPULS' with the following parameter values (see Figure H.3(b1) in Appendix H):

$$\begin{aligned} V1 = 0, \quad V2 = 5, \quad TD(\text{Delay Time}) = 0, \quad TR(\text{Rise Time}) = 0, \\ TF(\text{Falling Time}) = 0, \quad PW(\text{Pulse Width}) = 0.5 \text{ n}, \quad PER(\text{Period}) = 1 \text{ n} \end{aligned} \quad (\text{P3.9.2})$$

Are the upper/lower limits of the steady state voltage $v_{C1}(t)$ obtained from the PSpice simulation similar to those estimated from the result of Problem 3.8(a)?

- (b) Figure P3.9(b) shows an RC circuit model of two interconnected CMOS (complementary metal oxide semiconductor) inverters, where the switch moves up to position u or down to position d depending on whether the input voltage v_{i1} falls below the low threshold V_L or rises above the high threshold V_H . Note from Remark 1.3 that the left part of the left V_s and the right part of the right V_s can be removed without making any difference to the analysis of the interconnection part since each of them is connected in parallel with a voltage source. Also based on the observation made in (a), the right V_s will be neglected so that the interconnection part can be modeled as Figures P3.9(c1) and (c2) for the pull-up phase and pull-down phase, respectively. Find the pull-up transition time taken for v_{i2} to rise from V_L to V_H and the pull-down transition time taken for v_{i2} to fall from V_H to V_L in terms of V_H , V_L , R_p , R_n , C_p , and C_n .

3.10 An RC Circuit Containing a Dependent (Controlled) Source

Consider the RC circuit of Figure P3.10.

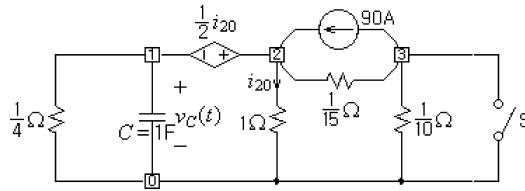


Figure P3.10

- Assuming that the switch S has been open for a long time until it is closed at $t = 0$, find the capacitor voltage $v_C(t)$ for $t \geq 0$.
- Assuming that the switch S has been closed for a long time until it is open at $t = 0$, find the capacitor voltage $v_C(t)$ for $t \geq 0$.

3.11 An RL Circuit Containing Two Dependent (Controlled) Sources

Consider the RL circuit of Figure P3.11.

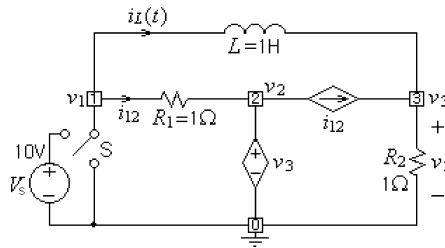


Figure P3.11

- Assuming that the switch S has been connected to the voltage source V_s for a long time until it is flipped to node 0 at $t = 0$, find the inductor current $i_L(t)$ for $t \geq 0$.
- Assuming that the switch S has been connected to node 0 for a long time until it is flipped to V_s at $t = 0$, find the inductor current $i_L(t)$ for $t \geq 0$.

3.12 An RC Circuit Containing Two Dependent (Controlled) Sources

Consider the RC circuit of Figure P3.12.

- Assuming that the switch S has been connected to node 1 for a long time until it is flipped to node 0 at $t = 0$, find the capacitor voltage $v_C(t)$ for $t \geq 0$.
- Assuming that the switch S has been connected to node 0 for a long time until it is flipped to node 1 at $t = 0$, find the capacitor voltage $v_C(t)$ for $t \geq 0$.

3.13 Sequential Switching

Consider the RL circuit of Figure P3.11.

- Assuming that the switch has been connected to the voltage source V_s for a long time until it is flipped to node 0 at $t = 0$, and then back to V_s at $t = 2$ s, find the inductor current $i_L(t)$ for $0 \leq t < 2$ and $t \geq 2$.

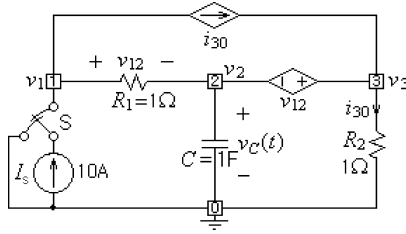


Figure P3.12

- (b) Assuming that the switch has been connected to node 0 for a long time until it is flipped to V_s at $t = 0$, and then back to node 0 at $t = 2$ s, find the inductor current $i_L(t)$ for $0 \leq t < 2$ and $t \geq 2$.

3.14 Sequential Switching

Consider the RC circuit of Figure P3.12.

- (a) Assuming that the switch has been connected to node 1 for a long time until it is flipped to node 0 at $t = 0$, and then back to node 1 at $t = 2$ s, find the capacitor voltage $v_C(t)$ for $0 \leq t < 2$ and $t \geq 2$.
- (b) Assuming that the switch has been connected to node 0 for a long time until it is flipped to node 1 at $t = 0$, and then back to node 0 at $t = 2$ s, find the capacitor voltage $v_C(t)$ for $0 \leq t < 2$ and $t \geq 2$.

3.15 The Input–Output Relationship of a First-Order Circuit Containing a Dependent Source

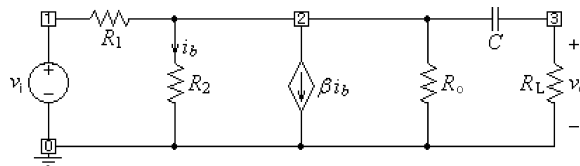


Figure P3.15

Consider the circuit of Figure P3.15. Assuming that the capacitor C has no initial voltage ($v_C(0) = 0$), verify that the (Laplace-transformed) output voltage can be written in terms of R_1 , R_2 , R_o , R_L , C , β , and $V_i(s) = \mathcal{L}\{v_i(t)\}$ as follows:

$$V_o(s) = V_3(s) = \frac{s C G_1}{[G_1 + (\beta + 1)G_2 + G_o + G_L]Cs + [G_1 + (\beta + 1)G_2 + G_o]G_L} V_i(s) \quad (\text{P3.15.1})$$

where $G_1 = 1/R_1$, $G_2 = 1/R_2$, $G_o = 1/R_o$, and $G_L = 1/R_L$.

3.16 A Rectangular/Triangular-Wave Generator

Consider the circuit of Figure P3.16(a), which generates a triangular wave at node 1 and two rectangular waves at nodes 2 and 3, as depicted in Figure P3.16(b). The OP Amp U3 (having no feedback path) outputs $v_3 = \pm V_{om}$ depending on which one of the two inputs v_1 and v_2 is higher, functioning as a comparator. The output v_3 of U3 is applied to the input terminal of the inverting RC

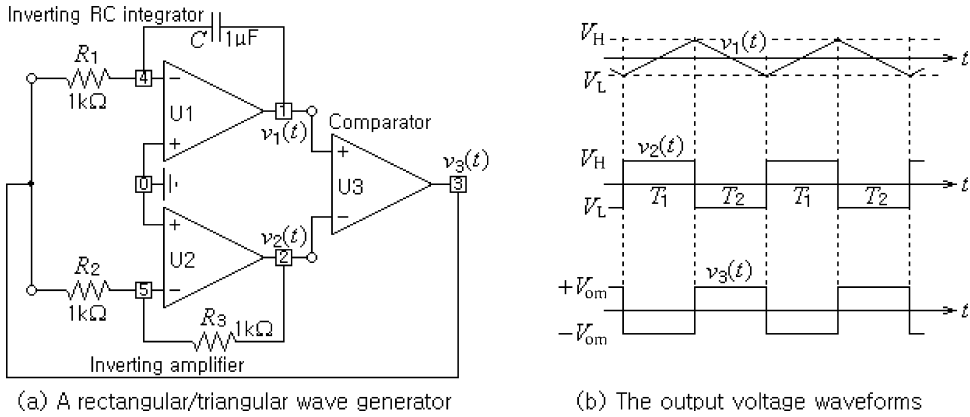


Figure P3.16

integrator, making its output

$$v_1(t) = -\frac{1}{R_1 C} \int_{t_0}^t v_3(\tau) d\tau + v_1(t_0) \quad (\text{P3.16.1})$$

and also to the input terminal of the inverting amplifier, making its output

$$v_2(t) = -\frac{R_3}{R_2} v_3(t) \quad (\text{P3.16.2})$$

- (a) Circle the appropriate one of the two examples in the following parentheses. Let $v_1 = 0$ and $v_3 = +V_{om}$ so that $v_2 = -(R_3/R_2)V_{om}, +(R_3/R_2)V_{om}$ at some time. Since the input of the inverting integrator is positive, its output voltage v_1 goes (down, up) until it reaches $V_L = -(R_3/R_2)V_{om}, +(R_3/R_2)V_{om}$. As soon as v_1 goes below V_L , the comparator output voltage v_3 becomes $-V_{om}$ so that $v_2 = -(R_3/R_2)V_{om}, +(R_3/R_2)V_{om}$ and, accordingly, the inverting integrator output v_1 goes (up, down) until it reaches $V_H = -(R_3/R_2)V_{om}, +(R_3/R_2)V_{om}$. This cycle repeats itself over and over again.
- (b) Find the lengths of the rising/falling periods T_1 and T_2 and the whole period $P = T_1 + T_2$ in terms of R_1, R_2, R_3 , and C . Will it be valid for $R_3/R_2 > 1$? If not, how should it be modified for that case?

3.17 Realization of a First-Order Pole/Zero

Figure P3.17 shows two connection diagrams of an OP Amp-RC chip. First, complete the corresponding circuit diagrams on the right-hand side and fill in the square boxes with the appropriate pin numbers. Second, find the transfer functions, i.e., the ratios of the transformed output voltage to the transformed input voltage.

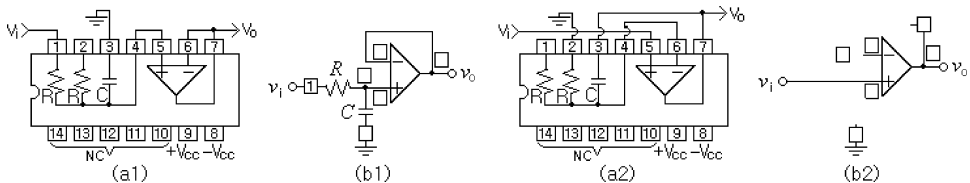


Figure P3.17

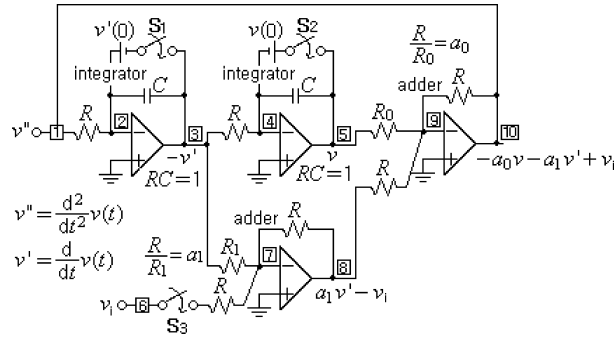


Figure P3.18 An analog computer for solving a differential equation

3.18 Analog Computer

Figure P3.18 shows the block diagram of an analog computer for solving the following differential equation, where $R/R_1 = a_1$, $R/R_0 = a_0$, and $RC = 1$:

$$\frac{d^2}{dt^2}v(t) + a_1 \frac{d}{dt}v(t) + a_0v(t) = v_1(t) \quad (\text{P3.18.1})$$

At which node can you get the solution of the differential equation?

3.19 A Small-Gain RC OP Amp Integrator

- Note that, by the short principle in connection with the negative feedback, the voltage potential at the negative input terminal (node 3) of the OP Amp U1 is $V_3 \simeq 0$, following that of the positive input terminal (node 0), which is grounded. Apply KCL to set up the node equation and solve it for $V_2(s)$.
- Assuming that the capacitor has no initial voltage, verify that the transformed output voltage of the circuit is

$$V_o(s) = -\frac{1}{sR_2C}V_2(s) = -\frac{1}{sC[R(1 + R_2/R_1) + R_2]}V_1(s) \quad (\text{P3.19.1})$$

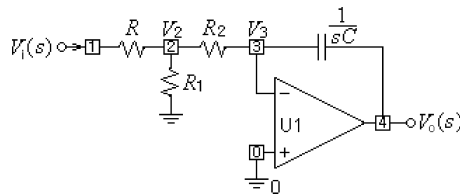


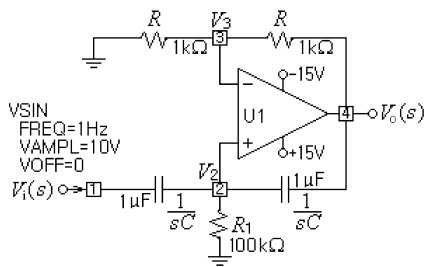
Figure P3.19 An RC OP Amp Integrator

Note. Theoretically, the magnitude of the gain of this integrator can be made as small as required by increasing R_2/R_1 , i.e. the ratio of the two resistances regardless of the value of each resistance.

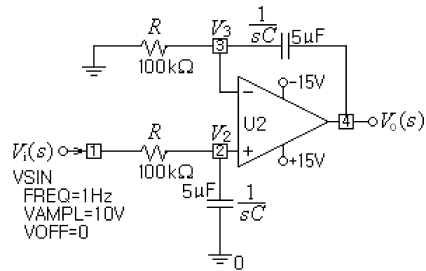
3.20 RC OP Amp Circuits – Differentiator and Integrator

- Verify that the transformed output voltage of the circuit in Figure P3.20(a) is

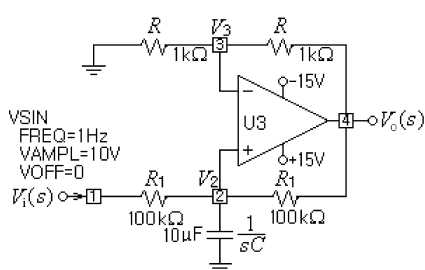
$$V_o(s) = 2sR_1CV_1(s) \quad (\text{P3.20.1})$$



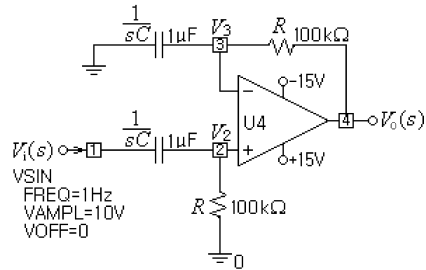
(a) A noninverting differentiator



(b) A noninverting integrator



(c) A noninverting integrator



(d) A noninverting differentiator

Figure P3.20

Hint. First of all, express $V_3(s)$ in terms of $V_o(s)$ by using the voltage divider rule (Section 2.2.1) and set $V_2(s) = V_3(s)$ by the short principle activated by negative feedback. Then apply KCL to node 2 to obtain the relationship between $V_o(s)$ and $V_i(s)$.

- Perform the PSpice simulation of circuit (a) with $R = 1\text{ k}\Omega$, $R_1 = 100\text{ k}\Omega$, and $C = 1\text{ }\mu\text{F}$ excited by a sinusoidal voltage source $v_i(t) = 10\sin(2\pi f t)$ ($f = 1\text{ Hz}$) for 5 s to see the amplitude of the output voltage waveform. Is it close to that anticipated from the analytical expression?
- Verify that the transformed output voltage of the circuit in Figure P3.20(b) is

$$V_o(s) = \frac{1}{sRC} V_i(s) \quad (\text{P3.20.2})$$

Hint. First of all, express $V_2(s)$ in terms of $V_i(s)$ using the voltage divider rule and set $V_3(s) = V_2(s)$ by the short principle activated by the negative feedback. Then apply KCL to node 3 to obtain the relationship between $V_o(s)$ and $V_i(s)$.

- Perform the PSpice simulation of circuit (b) with $R = 100\text{ k}\Omega$ and $C = 5\text{ }\mu\text{F}$ excited by a sinusoidal voltage source $v_i(t) = 10\sin(2\pi f t)$ ($f = 1\text{ Hz}$) for 5 s to see the amplitude of the output voltage waveform. Is it close to that anticipated from the analytical expression?
- Express the transformed output voltage $V_o(s)$ of the circuit in Figure P3.20(c) in terms of R_1 , C , and $V_i(s)$.
- Perform the PSpice simulation of circuit (c) with $R = 1\text{ k}\Omega$, $R_1 = 100\text{ k}\Omega$, and $C = 10\text{ }\mu\text{F}$ excited by a sinusoidal voltage source $v_i(t) = 10\sin(2\pi f t)$ ($f = 1\text{ Hz}$) for 5 s to see the amplitude of the output voltage waveform. Is it close to that anticipated from the analytical expression?
- Express the transformed output voltage $V_o(s)$ of the circuit in Figure P3.20(d) in terms of R , C , and $V_i(s)$.

- (h) Perform the PSpice simulation of circuit (d) with $R = 100\text{ k}\Omega$ and $C = 1\text{ }\mu\text{F}$ excited by a sinusoidal voltage source $v_i(t) = 10 \sin(2\pi f t)$ ($f = 1\text{ Hz}$) for 5 s to see the amplitude of the output voltage waveform. Is it close to that anticipated from the analytical expression?

Note. In the Simulation Settings dialog box, you can set Run_to_time and Maximum step to 5 s and 1 ms, respectively, and check the square box before ‘Skip the initial bias point calculation (SKIPBP)’.

3.21 Capacitance Multiplier

Consider the circuit of Figure P3.21.

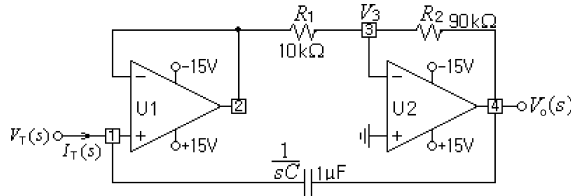


Figure P3.21

- (a) Verify that the transformed output voltage $V_o(s)$ can be expressed in terms of the transformed test input voltage $V_T(s)$ as follows:

$$V_o(s) = -\frac{R_2}{R_1} V_T(s) \quad (\text{P3.21.1})$$

- (b) Verify that the input impedance, i.e. the ratio of the transformed test input voltage $V_T(s)$ to the transformed test current $I_T(s)$, is obtained as follows:

$$Z_{in}(s) = \frac{V_T(s)}{I_T(s)} = \frac{1}{sC(R_2 + R_1)/R_1} \quad (\text{P3.21.2})$$

- (c) Perform the PSpice simulation of this circuit with $R_1 = 10\text{ k}\Omega$, $R_2 = 90\text{ k}\Omega$, and $C = 1\text{ }\mu\text{F}$ excited by a triangular-wave voltage source $v_T(t)$ (VPULSE with TD (Time Delay) = 0 s, TF (Falling Time) = 0.5 ms, PW (Pulse Width) = 0.01 ms, PER (PERiod) = 1 ms, $V_1 = -1\text{ V}$, TR (Rise Time) = 0.49 ms, $V_2 = 1\text{ V}$) for 5 ms to see the test current waveform $i_T(t)$. Is it close to that anticipated from the analytical expression? Note that the first derivative (slope) of the triangular voltage waveform is $dv_T(t)/dt = \pm 2\text{ V}/0.5\text{ ms} = \pm 4000\text{ V/s}$ for the rising/falling periods, respectively, and the voltage–current relationship of the circuit is

$$I_T(s) = sC \frac{R_1 + R_2}{R_1} V_T(s); \quad i_T(t) = C \frac{R_1 + R_2}{R_1} \frac{d}{dt} v_T(t) \quad (\text{P3.21.3})$$

Note. In the Simulation Settings dialog box, set Run_to_time and Maximum step to 5 ms and 1 μs, respectively, and do not check the square box before ‘Skip the initial bias point calculation (SKIPBP)’.

Note. This implies that the capacitance is magnified $(R_1 + R_2)/R_1$ times. However, this function of magnifying the capacitance becomes ineffective as the frequency of the input voltage signal and/or the capacitance increase(s) so that the output voltage is directly affected by the input signal rather than determined as Equation (P3.21.1) by the negative feedback mechanism of the OP Amp U2. Note that the

AC (alternating current) impedance of a capacitance C has the magnitude of $1/(\omega C)$, which is called the reactance (see Section 6.3.3).

3.22 Inductance Emulator Using a Capacitor

(a) For the circuit of Figure P3.22(a), verify the following:

$$V_4(s) = \frac{R_3 + R_4}{R_4} V_T(s) \quad (\text{P3.22.1})$$

$$V_2(s) = \left(1 + \frac{R_3}{sR_2R_4C}\right) V_T(s) \quad (\text{P3.22.2})$$

$$\text{The input impedance: } Z_{in}(s) = \frac{V_T(s)}{I_T(s)} = sC \frac{R_1R_2R_4}{R_3} \quad (\text{P3.22.3})$$

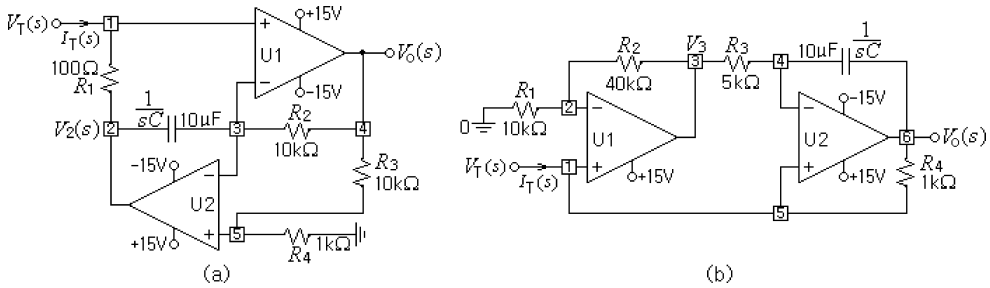


Figure P3.22

(b) Perform the PSpice simulation of this circuit with $R_1 = 100\ \Omega$, $R_2 = 10\ \text{k}\Omega$, $R_3 = 10\ \text{k}\Omega$, $R_4 = 1\ \text{k}\Omega$, and $C = 10\ \mu\text{F}$ excited by a sinusoidal voltage source $v_T(t) = 0.1 \sin(2\pi f t)$ ($f = 1\ \text{Hz}$) for 5 s to see the amplitude of the test current waveform $i_T(t)$. Is it close to that anticipated from the analytical expression?

(c) For the circuit of Figure P3.22(b), verify the following:

$$V_6(s) = \left(1 - \frac{R_2}{sR_1R_3C}\right) V_T(s) \quad (\text{P3.22.4})$$

$$\text{The input impedance: } Z_{in}(s) = \frac{V_T(s)}{I_T(s)} = sC \frac{R_1R_3R_4}{R_2} \quad (\text{P3.22.5})$$

(d) Perform the PSpice simulation of this circuit with $R_1 = 10\ \text{k}\Omega$, $R_2 = 40\ \text{k}\Omega$, $R_3 = 5\ \text{k}\Omega$, $R_4 = 1\ \text{k}\Omega$, and $C = 10\ \mu\text{F}$ excited by a sinusoidal voltage source $v_T(t) = 0.1 \sin(2\pi f t)$ ($f = 0.1\ \text{Hz}$) for 50 s to see the amplitude of the test current waveform $i_T(t)$. Is it close to that anticipated from the analytical expression?

Note. The input impedances (P3.22.3) and (P3.22.5) imply that both of the circuits of Figures P3.22(a) and (b) function as virtual inductors. However, their inductive functions become ineffective as the frequency of the input voltage signal and/or the capacitance decrease(s) so that the negative feedback effect through the capacitor is attenuated.

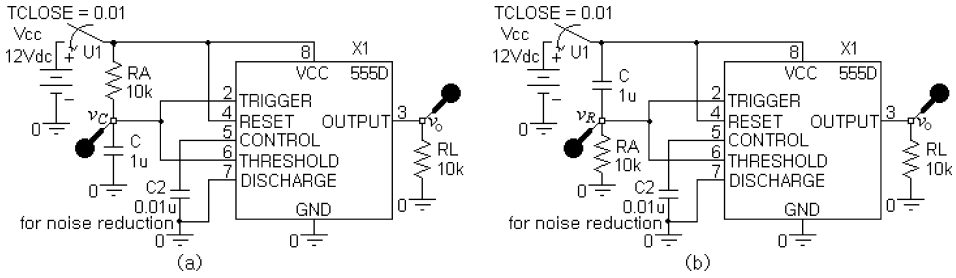


Figure P3.23 Two types of power-on delays realized by a 555 timer

3.23 Applications of the 555 Timer/Oscillator (Reference [W-4])

Refer to Example 3.4 for the internal structure/behavior of the 555 timer.

- Let the switch be closed at $t = 0.01$ s in the 555 timer circuit of Figure P3.23(a), where the switch has been open for a long time before $t = 0.01$ s. Find the duration (pulse width) of the rectangular pulse $v_o(t)$ appearing at the output terminal. Support your results by PSpice simulation.
- Let the switch be closed at $t = 0.01$ s in the 555 timer circuit of Figure P3.23(b), where the switch has been open for a long time before $t = 0.01$ s. Find the time delay of $v_o(t)$ measured from the switching-on time. Support your results by PSpice simulation.

3.24 Design of an RL Circuit with a Specified Ripple

Note from Problem 3.2 that the current ripple of the RL circuit (Figure P3.2) excited by a periodically switched voltage source is maximized for the duty cycle of $d = T_{ON}/(T_{ON} + T_{OFF}) = 1/2$ (50%) or, equivalently, for $T_{ON} = T_{OFF}$, and accordingly the maximum ripple current and the corresponding average current are as follows:

$$\Delta I_{\max} = \frac{V_i - E}{R} \frac{(1 - e^{-T_{ON}/T})^2}{1 - e^{-2T_{ON}/T}} = \frac{V_i - E}{R} \frac{1 - e^{-T_{ON}/T}}{1 + e^{-T_{ON}/T}} \stackrel{T=L/R}{=} \frac{V_i - E}{R} \frac{1 - e^{-T_{ON}R/L}}{1 + e^{-T_{ON}R/L}} \quad (\text{P3.24.1})$$

$$I_a = \frac{I_1 + I_2}{2} = \frac{V_i - E}{2R} \frac{1 - e^{-2T_{ON}/T}}{1 - e^{-2T_{ON}/T}} = \frac{V_i - E}{2R} \quad (\text{P3.24.2})$$

Verify that, in order to keep the relative ripple to the average current, $\Delta I_{\max}/I_a$, limited below r (a positive constant less than 1), the following condition is required for the time constant $T = L/R$:

$$T > \frac{T_{ON}}{\ln[(2+r)/(2-r)]} \quad (\text{P3.24.3})$$

3.25 Design of an RC OP Amp Circuit with a Variable Time Constant Depending on a Resistance

Consider the RC OP Amp circuit of Figure P3.25, where $R_1 = 10 \text{ k}\Omega$, $R_3 = 10 \text{ k}\Omega$, and $R_4 = 5 \text{ k}\Omega$.

- Determine the value of the capacitance C such that the time constant becomes 1 ms for $R_2 = 0 \Omega$.
- With the capacitance determined in (a) and $R_2 = 5 \text{ k}\Omega$, find the output voltage waveform $v_o(t)$ for $v_C(0) = 1 \text{ V}$. Support your design/analysis results by PSpice simulation for 5 s.

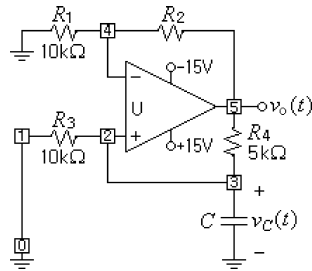


Figure P3.25

- (c) With the capacitance determined in (a) and $R_2 = 10\text{ k}\Omega$, find the output voltage waveform $v_o(t)$ for $v_C(0) = 1\text{ V}$. Support your design/analysis results by PSpice simulation for 5 s.

Hint. Refer to Problem 2.26 for the Thevenin equivalent resistance of this circuit (with the capacitor C open) seen from nodes 3 and 0:

$$R_o = \frac{R_4}{R_4/R_3 - R_2/R_1} \quad (\text{P3.25.1})$$

3.26 Design of a Square-Wave Generator with a Variable Period Depending on a Resistance

Consider the square-wave generator of Figure 3.15(a) in which the values of C and R_2 are $C = 100\text{ }\mu\text{F}$ and $R_2 = 0 \sim 10\text{ k}\Omega$ (variable), respectively.

- (a) Determine the values of R_1 and R_3 such that the period of the square wave becomes $P_{\max} = 4.5\text{ s}$ for $R_2 = 10\text{ k}\Omega$ and $P_{\min} = 0.5\text{ s}$ for $R_2 = 100\text{ k}\Omega$ (maximum).

Hint. Equation (3.64) can be used to write the design specifications on the period of the square wave as

$$P_{\max} = 2R_3C \ln \frac{2R_1 + R_2}{R_2} \bigg|_{R_2=R_{2,\min}=10\text{ k}\Omega} = 4.5\text{ s} \quad (\text{P3.26.1})$$

$$P_{\min} = 2R_3C \ln \frac{2R_1 + R_2}{R_2} \bigg|_{R_2=R_{2,\max}=100\text{ k}\Omega} = 0.5\text{ s} \quad (\text{P3.26.2})$$

which can be solved using the following MATLAB program.

```
%cir03p26.m
clear, clf
R2min=1e4; R2max=1e5; C=100e-6;
Pmax=4.5; Pmin=0.5; % desired minimum period of a wiper cycle.
x_0=[10 100]; % Initial guess for 2*R3C and R1
options=optimset('fsolve');
xo=fsolve('f_cir03p26', x_0, options, R2min, R2max, Pmax, Pmin)
R1=xo(2), R3C2=xo(1); R3=R3C2/C/2
R1=standard_value(R1, 'R', 'c', 1), R3=standard_value(R3, 'R', 'c', 1)
function y=f_cir03p26(x, R2min, R2max, Pmax, Pmin)
R3C2=x(1); R1=x(2); % R3C2=2*R3*C
y=[R3C2*log((2*R1+R2min)/R2min)-Pmax;
   R3C2*log((2*R1+R2max)/R2max)-Pmin]; % Eq. (P3.26.1&2)
```

- (b) Select the appropriate 1% tolerance standard resistance values of R_1 and R_3 from Table G.2 (Appendix G) and support your design results by two PSpice simulations, one with $R_2 = 10\text{ k}\Omega$ for 10 s and one with $R_2 = 100\text{ k}\Omega$ for 1 s. Set the initial voltage (IC, or initial condition) of the capacitor C to $v_C(0) = 0\text{ V}$ in the Property Editor spreadsheet, set the maximum step in the Transient analysis options to 1 ms, and check the square box before 'Skip the initial transient bias point calculation (SKIPBP)' in the Simulation Settings dialog box (Figure H.5(c1) in Appendix H) so that the initial transient bias point calculation will be skipped.

Note. If you come across a warning message such as 'Unable to find library file templates.lib; Subcircuit uA741 is undefined' for a device like an OP Amp or a transistor, you can click that device for selection and click Edit/PSpice Model on the menu bar of the Capture window to open the PSpice Model Editor window, press '^s' to save the library file for that device, and just click on x on the PSpice Model Editor window to close it.

3.27 A Nonlinear (First-Order) RL Circuit Driven by a Sinusoidal Source

Consider the circuit of Figure P3.27(a), which consists of a nonlinear resistor, a linear resistor $R = 2\ \Omega$, and an inductor $L = 14\text{ H}$, and is driven by a DC voltage source of $V_s = 12\text{ V}$ and an AC voltage source $v_\delta \sin t = 2.8 \sin t\text{ [V]}$. The v - i relationship of the nonlinear resistor is $v_2(i) = i^3$ and is described by the characteristic curve in Figure P3.27(b). KVL can be applied to obtain the following mesh equation:

$$L \frac{di(t)}{dt} + Ri(t) + i^3(t) = V_s + v_\delta \sin t; 14 \frac{di(t)}{dt} + 2i(t) + i^3(t) = 12 + 2.8 \sin t \quad (\text{P3.27.1})$$

- (a) Verify that the equation for the operating point Q in the DC steady state is obtained by removing the AC source and the time derivative term and can be solved as

$$2I_Q + I_Q^3 = 12; \quad I_Q = 2\text{ A}; \quad V_Q = v_2(I_Q) = I_Q^3 = 8\text{ V} \rightarrow Q = (I_Q, V_Q) = (2\text{ A}, 8\text{ V}) \quad (\text{P3.27.2})$$

Hint. This can be solved by typing the following statements into the MATLAB command window:

```
>>ftn=inline('2*i+i.^3-12','i');
>>I0=0; IQ=fsolve(ftn,I0)
```

- (b) Verify that in order to linearize the nonlinear differential equation (P3.27.1) around the operating point Q , we can substitute $i = I_Q + \delta i = 2 + \delta i$ and neglect the second or higher degree terms in δi as

$$14 \frac{d(2+\delta i)}{dt} + 2(2+\delta i) + (2+\delta i)^3 = 12 + 2.8 \sin t; \quad \frac{d}{dt} \delta i(t) = -\delta i(t) + 0.2 \sin t \quad (\text{P3.27.3})$$

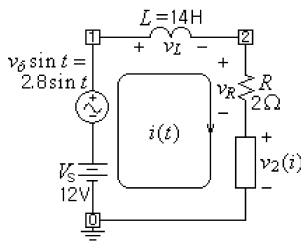
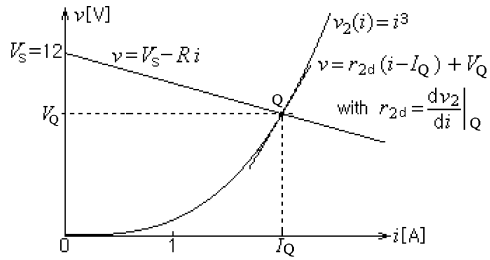
Note. This can be obtained by applying KVL to the circuit with the DC source V_s removed and the nonlinear resistor replaced by its dynamic resistance $r_{2d} = dv_2/di|_Q = 3I_Q^2 = 12\ \Omega$.

- (c) Solve the linear first-order differential equation (P3.27.3) with the zero initial condition $\delta i(0) = 0$ to get $\delta i(t)$ and use it to write the approximate solution for $i(t)$ as

$$i(t) = I_Q + \delta i(t) = 2 + 0.1(e^{-t} - \cos t + \sin t)\text{ [A]} \quad (\text{P3.27.4})$$

Hint. This can be solved by typing the following statements into the MATLAB command window:

```
>>syms s; dIs=0.2/(s^2+1)/(s+1); dit_linearized= ilaplace(dIs)
>>dit1_linearized=dsolve('Dx=-x+0.2*sin(t)','x(0)=0') % Alternatively
```

(a) A nonlinear RL circuit(b) The v - i characteristic curve and the load lines**Figure P3.27**

- (d) Use the MATLAB routine `ode45` () to solve the nonlinear first-order differential equation (P3.27.1). Plot the numerical solution $i(t)$ for the time interval $[0, 10]$ s to compare it with the approximate analytical solution (P3.27.4).

Hint. Referring to Appendix D, Equation (P3.27.1) can be cast into the following MATLAB function and saved as an M-file `cir03p27_f.m` in a directory that can be searched by MATLAB:

```
function di=cir03p27_f(t,i)
di= (12+2.8*sin(t)-2*i-i.^3)/14; % (P3.27.1)
```

Then type the following statements into the MATLAB command window:

```
>>di0= 0; i0=IQ+di0; tspan=[0 10] ;
>>[t,i_numerical]=ode45(@cir03p27_f, tspan, i0); % numerical sol
>>i_linearized=eval(IQ+diti_linearized); % analytical sol (P3.27.4)
>>plot(t,i_numerical,'k', t,i_linearized,'r')
```


4

Second-Order Circuits

In this chapter second-order circuits are studied whose behavior can be described by second-order (ordinary linear) differential equations. The order of a circuit equation equals the number of energy storage elements resulting from all the possible series/parallel combinations of inductors/capacitors. In fact, there is no reason why the scope should be limited to second-order circuits. However, only up to second-order circuits are discussed in detail because the responses of higher-order circuits can be approximated by linear combinations of the responses of first/second-order circuits. By applying the Laplace transform method together with the symbolic computation of MATLAB there is no difficulty in solving higher-order circuits, even in the case where they are driven by sinusoidal sources.

Especially in Section 4.5, the concepts of the transfer function and the impulse response are introduced and the input–output relationship of a linear time-invariant (LTI) system is derived in the form of convolution to expose readers to the system theory in order to give a broad view of circuit systems. In Section 4.6, for the purpose of making the readers ready to study the analysis of AC circuits, it is examined how the steady state response of a system to a sinusoidal input is expressed in terms of the frequency response. The frequency response of a system is obtained by substituting $s = j\omega$ into the transfer function, where ω is the angular frequency of the input source applied to the system.

4.1 The Laplace Transform for Second-Order Differential Equations

In the previous chapter use of the Laplace transform for solving the first-order circuits was discussed. Here we consider a second-order differential equation

$$\frac{d^2y(t)}{dt^2} + a_1 \frac{dy(t)}{dt} + a_0 y(t) = x(t) \quad \text{with the initial condition } y(0) = y_0, y'(0) = y_1 \quad (4.1)$$

which describes the time-domain relationship between the input $x(t)$ and the output $y(t)$ of a (circuit) system. Taking the Laplace transform of both sides and using the differentiation property (Table A.2(5) in Appendix A) of the Laplace transform yields

$$\begin{aligned} s^2 Y(s) - y'(0) - s y(0) + a_1 [s Y(s) - y(0)] + a_0 Y(s) &= X(s) \\ (s^2 + a_1 s + a_0) Y(s) &= X(s) + y'(0) + s y(0) + a_1 y(0) \end{aligned}$$

This algebraic equation is solved to obtain the s -domain solution

$$Y(s) = \frac{X(s) + y'(0) + s y(0) + a_1 y(0)}{s^2 + a_1 s + a_0} \quad (4.2)$$

This expression will be expanded into partial fractions and the inverse Laplace transform taken to find $y(t)$. Assuming zero initial conditions $y(0) = 0$ and $y'(0) = 0$ for simplicity gives the *transfer* or *system function*, which is defined to be the s -domain input–output relationship, i.e. the ratio of the transformed output to the transformed input (with zero initial conditions) as

$$G(s) = \frac{Y(s)}{X(s)} = \frac{1}{s^2 + a_1s + a_0} \quad (4.3)$$

Suppose that the input is of the unit step function $x(t) = u_s(t)$ with $X(s) = 1/s$. Then the transformed output becomes

$$Y(s) = G(s)X(s) = \frac{X(s)}{s^2 + a_1s + a_0} = \frac{1}{s(s^2 + 2\zeta\omega_r s + \omega_r^2)} \quad (4.4)$$

The process and result of taking the partial fraction expansion of Equation (4.4) depends on the *characteristic roots*, i.e. the roots of the *characteristic equation*, which is formed by setting the denominator of the transfer function (4.3) to zero:

$$s^2 + a_1s + a_0 = s^2 + 2\zeta\omega_r s + \omega_r^2 = 0 \quad \text{with } \omega_r = \sqrt{a_0}, \quad \zeta = a_1/(2\omega_r) = a_1/2\sqrt{a_0} \quad (4.5)$$

where the discriminant of this equation is

$$D = a_1^2 - 4a_0 = (2\zeta\omega_r)^2 - 4\omega_r^2 = 4(\zeta^2 - 1)\omega_r^2$$

Depending on the value of the discriminant D or the parameter ζ (zeta), there are three cases:

- (1) The overdamped case with two distinct real roots: $|\zeta| > 1$
- (2) The critically damped case with double real roots: $|\zeta| = 1$
- (3) The underdamped case with two distinct complex roots: $0 \leq |\zeta| < 1$

Before looking into these three cases in detail, let us think about the meaning of the *characteristic equation*, i.e. ‘Why do we call it the characteristic equation?’ It is so called because it characterizes the behavior of the system regardless of the input or the initial conditions in the sense that its roots (called the characteristic roots) tell about the transient response, i.e. the output of the system during the transient period; this will be further explored.

4.1.1 Overdamped Case with Two Distinct Real Characteristic Roots

With two distinct real roots

$$s_1, s_2 = \frac{1}{2}(-a_1 \pm \sqrt{a_1^2 - 4a_0}) = -\zeta\omega_r \pm \sqrt{\zeta^2 - 1}\omega_r \quad \text{with } |\zeta| > 1 \quad (4.6)$$

the transformed output equation (4.4) can be expanded into the partial fraction form as

$$Y(s) = \frac{1}{s(s - s_1)(s - s_2)} = \frac{K_0}{s} + \frac{K_1}{s - s_1} + \frac{K_2}{s - s_2} \quad (4.7)$$

for which the inverse Laplace transform is obtained as

$$y(t) = \mathcal{L}^{-1}\{Y(s)\} \stackrel{\text{Table A.1(3),(5)}}{=} (K_0 + K_1 e^{s_1 t} + K_2 e^{s_2 t}) u_s(t) \quad (4.8)$$

If $a_1 > 0$ so that $s_1 < 0$ and $s_2 < 0$ with $\zeta > 1$, this output converges to K_0 and the system is said to be *stable*. If $a_1 < 0$ so that $s_1 > 0$ and $s_2 > 0$ with $\zeta < -1$, this output diverges (to ∞ or $-\infty$) and the system is said to be *unstable* in the sense that the output is unbounded for a bounded input like $x(t) = u_s(t)$.

4.1.2 Critically Damped Case with Double Real Characteristic Roots

With double real roots

$$s_1, s_2 = -\frac{a_1}{2} = -\omega_r \quad \text{or} \quad \omega_r \quad \text{with} \quad |\zeta| = 1 \quad (4.9)$$

the transformed output equation (4.4) can be expanded into the partial fraction form as

$$Y(s) = \frac{1}{s(s-s_1)^2} = \frac{K_0}{s} + \frac{K_1}{(s-s_1)^2} + \frac{K_2}{s-s_1} \quad (4.10)$$

for which the inverse Laplace transform is obtained as

$$y(t) = \mathcal{L}^{-1}\{Y(s)\} \stackrel{\text{Table A.1(3),(5),(6)}}{=} (K_0 + K_1 t e^{s_1 t} + K_2 e^{s_2 t}) u_s(t) \quad (4.11)$$

If $a_1 > 0$ so that $s_1 = s_2 < 0$ with $\zeta = 1$, this output converges to K_0 and the system is said to be *stable*. If $a_1 < 0$ so that $s_1 = s_2 > 0$ with $\zeta = -1$, this output diverges and the system is said to be *unstable* in the sense that the output is unbounded even for a bounded input like $x(t) = u_s(t)$.

Note. You may wonder whether $t e^{-at} = t/e^{at}$ (with $a > 0$) converges. Apply L'Hospital's rule (refer to the website <http://tutorial.math.lamar.edu/AllBrowsers/2413/LHospitalsRule.asp>):

$$\lim_{t \rightarrow \infty} t e^{-at} = \lim_{t \rightarrow \infty} \frac{t}{e^{at}} = \lim_{t \rightarrow \infty} \frac{1}{a e^{at}} = 0$$

4.1.3 Underdamped Case with Two Distinct Complex Characteristic Roots

With two distinct complex roots

$$s_1, s_2 = \frac{1}{2}(-a_1 \pm j\sqrt{4a_0 - a_1^2}) = -\zeta\omega_r \pm j\sqrt{1 - \zeta^2}\omega_r = -\sigma \pm j\omega_d \quad \text{with} \quad 0 \leq |\zeta| < 1 \quad (4.12)$$

the transformed output equation (4.4) can be decomposed as

$$Y(s) = \frac{1}{s(s^2 + 2\zeta\omega_r s + \omega_r^2)} = \frac{1}{s((s + \sigma)^2 + \omega_d^2)} = \frac{K_0}{s} + \frac{K_1(s + \sigma)}{(s + \sigma)^2 + \omega_d^2} + \frac{K_2\omega_d}{(s + \sigma)^2 + \omega_d^2} \quad (4.13)$$

for which the inverse Laplace transform is obtained as

$$y(t) = \mathcal{L}^{-1}\{Y(s)\} \stackrel{\text{Table A.1(3),(9),(10)}}{=} (K_0 + K_1 e^{-\sigma t} \cos \omega_d t + K_2 e^{-\sigma t} \sin \omega_d t) u_s(t) \quad (4.14)$$

Note. There should be no concern about how to get the numerical values of the coefficients K_i 's, because they can be computed using the formula (A.28) in Appendix A or something similar and they do not affect the behavioral characteristic of the output.

Note. What is the notion of ‘damped’ contained in the terms ‘overdamped’, ‘critically damped’, and ‘underdamped’? It means that the amplitude decreases as time goes by.

The feature of the underdamped case that distinguishes it from the other cases is the oscillation (with the *damped frequency* of $\omega_d = \sqrt{1 - \zeta^2} \omega_r$) described by the cosine/sine terms in Equation (4.14), where ω_r is the *undamped resonant frequency* for the undamped case with $\zeta = 0$. If only the real part of the complex characteristic roots is negative, i.e. $-\zeta\omega_r = -\sigma < 0$, the amplitude of oscillation decreases (exponentially), where $\sigma = \zeta\omega_r$ is the *damping constant* describing how fast the amplitude decreases. The key parameter ζ , which affects the oscillation frequency as well as the damping constant, is called the *damping ratio*.

4.1.4 Stability of a System and Location of its Characteristic Roots

To obtain an overview of the relationship between the characteristic roots and the features of the system in terms of its natural response, it would be good to plot the locations of the characteristic roots on the complex plane called the s -plane, as depicted in Figure 4.1. From Figure 4.1, the following observations can be made:

1. The coefficients (a_0 , a_1 , ζ , and ω_r) of the characteristic equation (4.5) or the denominator of the transfer function (4.3) are solely determined by the system parameters and are not affected by the input ($x(t)$) or the initial conditions ($y(0)$ and $y'(0)$) of the system. This implies that the characteristic roots characterize the system itself rather than its outputs.
2. If only $a_1 > 0$ or, equivalently, $-\zeta\omega_r = -\sigma < 0$ so that the characteristic roots are located in the left-half plane (LHP), the system is *stable* in the sense that the output is bounded for any bounded input like $x(t) = u_s(t)$. In this case $T = 1/\sigma$ is the *time constant*, which is defined to be the time taken for the transient output to reach 63.2 % of its steady state value. On the contrary, if only $a_1 < 0$ or, equivalently, $-\zeta\omega_r = -\sigma > 0$ so that the characteristic roots are located in the right-half plane (RHP), the system is *unstable* in the sense that the output can be unbounded for a bounded input like $x(t) = u_s(t)$.
3. The closer the characteristic roots (with $\omega_d = \sqrt{1 - \zeta^2} \omega_r \gg \zeta\omega_r = \sigma$ or $\zeta \approx 0$) are to the $j\omega$ axis, the tougher the oscillation in the output stemming from the roots becomes, as depicted in Figure 4.1(3).

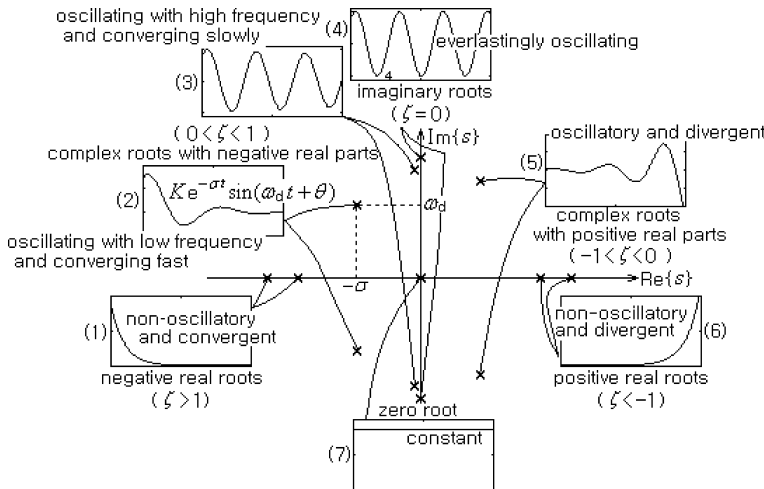


Figure 4.1 Locations of characteristics roots, the natural responses, and the system stability

4. If $a_1 = 0$ or, equivalently, $-\zeta\omega_r = -\sigma = 0$ so that the characteristic roots are located on the $j\omega$ axis (Figures 4.1(4) and (7)), the system is *marginally* or *neutrally stable* in the sense that the output is bounded for any bounded input that does not have the same mode as the characteristic roots. For example, suppose a sinusoidal input $x(t) = \cos t u_s(t)$ is applied to a neutrally stable system having the transfer function $G(s) = 2/(s^2 + 1)$, where the characteristic roots are obtained as $s = \pm j$ (lying on the $j\omega$ axis) by setting the denominator of $G(s)$ to zero. Noting that the Laplace transform of the input is $X(s) = \mathcal{L}\{\cos t u_s(t)\} \stackrel{\text{Table A.1(8)}}{=} s/(s^2 + 1)$, the transformed output and its inverse Laplace transform can be found as

$$Y(s) \stackrel{(4.4)}{=} G(s)X(s) = \frac{2}{s^2 + 1} \frac{s}{s^2 + 1} = \frac{2s}{(s^2 + 1)^2} \stackrel{\text{Table A.1(7)}}{\rightarrow} \stackrel{\text{Table A.2(7)}}{=} y(t) = t \sin t u_s(t)$$

which will diverge to ∞ as time goes by. However, for any other input than having the frequency corresponding to the characteristic roots $s = \pm j\omega_r$ with $\omega_r = 1$ rad/s, the system does not have unbounded output, but has some oscillatory output components with a constant amplitude and of the undamped resonant frequency ω_r . This shows that the output of a neutrally stable system is generally bounded except in the event of the input whose mode coincides with the characteristic roots.

Note. The following MATLAB statements can be typed into the MATLAB command window to check if the above inverse Laplace transform is correct:

```
>>syms s; ilaplace(2*s/(s^2+1)^2)
ans = t*sin(t)
```

5. The story about the stability of a second-order system in connection with its characteristic roots seems to be done. How about the stability of higher-order systems having more than two characteristic roots? If only a single characteristic root is in the RHP (Figures 4.1(5) and (6)), the system is *unstable*. If only a single real root or two complex characteristic roots are on the $j\omega$ axis and the other ones are all in the LHP, the system is *neutrally/marginally stable*. If and only if all the characteristic roots are in the LHP (Figures 4.1(1), (2), and (3)), the system is *stable*. In this context, the imaginary axis, i.e. the $j\omega$ axis on the s -plane ($s = \sigma + j\omega$: a complex variable) is the boundary that determines the stability of a system, where its characteristic roots are plotted on that plane.

4.2 Analysis of Second-Order Circuits

In this section a series *RLC* circuit, a parallel *RLC* circuit, and a circuit with two meshes/nodes are solved, which are described by a second-order (ordinary linear) differential equation. The responses of higher-order circuits can be regarded as a linear combination of the responses of first/second-order circuits. The Laplace transform method together with the symbolic computation of MATLAB may alleviate the computational difficulty involved in solving higher-order circuits.

4.2.1 A Series *RLC* Circuit

Consider the circuit of Figure 4.2.1(a) in which the initial values of the inductor current and the capacitor voltage are

$$i_L(0) = I_0 \quad \text{and} \quad v_C(0) = V_0$$

respectively. To find the mesh current $i(t)$, we apply KVL to the *RLC* loop to set up the mesh equation in the time domain as

$$v_R(t) + v_L(t) + v_C(t) = R i(t) + L \frac{di(t)}{dt} + \frac{1}{C} \int_{-\infty}^t i(\tau) d\tau = v_i(t)$$

and take its Laplace transform (Table A.2(5) and (6)) to write the transformed mesh equation as

$$RI(s) + L[sI(s) - I_0] + \frac{1}{C} \left[\frac{1}{s} I(s) + \frac{1}{s} \int_{-\infty}^0 i(\tau) d\tau \right] = V_i(s)$$

$$\left(R + sL + \frac{1}{sC} \right) I(s) = V_i(s) - \frac{1}{s} \left[\frac{1}{C} \int_{-\infty}^0 i(\tau) d\tau \right] + LI_0 = V_i(s) - \frac{V_0}{s} + LI_0 \quad (4.15)$$

A better way to get this equation is to transform the circuit into its s -domain equivalent (see Figure 3.6), as depicted in Figure 4.2.1(b) and apply the mesh analysis as if the circuit were made of just resistors and sources. In either case, Equation (4.15) is solved to obtain the transformed mesh current as

$$I(s) = \frac{V_i(s) - V_0/s + LI_0}{R + sL + 1/(sC)} = \frac{[sV_i(s) - V_0]/L + I_0s}{s^2 + sR/L + 1/(LC)} \quad (4.16)$$

and can take its inverse Laplace transform to find $i(t)$.

If the voltages across the inductor/capacitor are needed, they can be found by using the V - I relationships (3.15a) and (3.16b):

$$V_L(s) \stackrel{(3.15a)}{=} sLI(s) - Li_L(0)$$

$$V_C(s) \stackrel{(3.16b)}{=} \frac{1}{sC} I(s) + \frac{v_C(0)}{s}$$

Note. Be careful not to make the mistake of missing out the initial condition terms.

The transfer function of this circuit with the source voltage as the input and the mesh current as the output is

$$G(s) = \frac{I(s)}{V_i(s)} \bigg|_{\substack{\text{with zero initial conditions} \\ V_0=0, I_0=0}} \stackrel{(4.16)}{=} \frac{s/L}{s^2 + sR/L + 1/(LC)}$$

and the characteristic equation obtained by setting its denominator to zero is

$$s^2 + s\frac{R}{L} + \frac{1}{LC} = 0 \quad (4.17)$$

Note. The notation $G(s)$ denoting a transfer function should not be confused with G denoting a conductance.

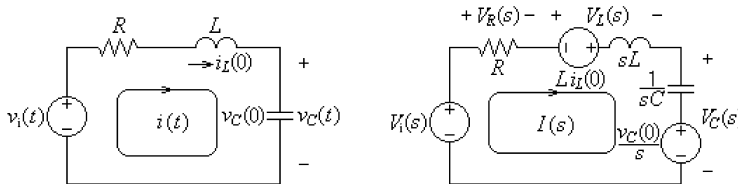


Figure 4.2.1 The circuit for Example 4.1

(Example 4.1) Time Responses of a Series RLC Circuit

Consider the series RLC circuit of Figure 4.2.1(a) in which the source voltage and the initial conditions of the capacitor and inductor are

$$v_i(t) = V_i u_s(t) = 2u_s(t) \text{ [V]} \rightarrow V_i(s) = \frac{2}{s}, \quad I_0 = 1 \text{ [A]}, \text{ and } V_0 = \frac{1}{2} \text{ [V]} \quad (\text{E4.1.1})$$

respectively. Noting that the discriminant of the characteristic equation (4.17) is $D = (R/L)^2 - 4/(LC)$, find the mesh current and the voltages across the inductor and capacitor for four different sets of values of R , L , and C .

(a) $R = 3/2 \Omega$, $L = 1/2 \text{ H}$, and $C = 1 \text{ F} \rightarrow D = (R/L)^2 - 4/(LC) > 0$ (overdamped)

The transformed mesh current (4.16) is expanded into the partial fraction form as

$$I(s) = \frac{[sV_i(s) - V_0]/L + I_0s}{s^2 + sR/L + 1/(LC)} = \frac{3 + s}{s^2 + 3s + 2} = \frac{K_1}{s + 1} + \frac{K_2}{s + 2} \quad (\text{E4.1.2})$$

where the coefficients are obtained by using the formula (A.28) in Appendix A as

$$K_1 \stackrel{(\text{A.28a})}{=} (s + 1)I(s)|_{s=-1} = \frac{s + 3}{s + 2} \Big|_{s=-1} = 2 \quad (\text{E4.1.3a})$$

$$K_2 \stackrel{(\text{A.28a})}{=} (s + 2)I(s)|_{s=-2} = \frac{s + 3}{s + 1} \Big|_{s=-2} = -1 \quad (\text{E4.1.3b})$$

Thus the inverse Laplace transform of $I(s)$ is taken to get the mesh current $i(t)$ as

$$\begin{aligned} I(s) &= \frac{K_1}{s + 1} + \frac{K_2}{s + 2} = \frac{2}{s + 1} - \frac{1}{s + 2} \\ i(t) &= \mathcal{L}^{-1}\{I(s)\} \stackrel{\text{Table A.1(5)}}{=} \begin{cases} I_0 = 1 \text{ [A]} & \text{for } t = 0^- \\ 2e^{-t} - e^{-2t} \text{ [A]} & \text{for } t \geq 0 \end{cases} \end{aligned} \quad (\text{E4.1.4})$$

Noting that this mesh current flows through the inductor and the capacitor in series, the s -domain V - I relationships (3.15a) and (3.16b) are used to obtain the voltages across them as

$$\begin{aligned} V_L(s) &\stackrel{(3.15a)}{=} sLI(s) - Li_L(0) \stackrel{(\text{E4.1.1})(\text{E4.1.2})}{=} \frac{s(s + 3)}{2(s^2 + 3s + 2)} - \frac{1}{2} \\ &= \frac{-1}{(s + 1)(s + 2)} = -\frac{1}{s + 1} + \frac{1}{s + 2} \\ v_L(t) &= \mathcal{L}^{-1}\{V_L(s)\} \stackrel{\text{Table A.1(5)}}{=} -e^{-t} + e^{-2t} \text{ [V]} \quad \text{for } t \geq 0 \end{aligned} \quad (\text{E4.1.5})$$

$$\begin{aligned} V_C(s) &\stackrel{(3.16b)}{=} \frac{1}{sC}I(s) + \frac{V_C(0)}{s} \stackrel{(\text{E4.1.1})(\text{E4.1.2})}{=} \frac{s + 3}{s(s + 1)(s + 2)} + \frac{1/2}{s} \\ &= \frac{3/2}{s} - \frac{2}{s + 1} + \frac{1/2}{s + 2} + \frac{1/2}{s} \\ v_C(t) &= \mathcal{L}^{-1}\{V_C(s)\} \stackrel{\text{Table A.1(5)}}{=} \begin{cases} V_0 = \frac{1}{2} \text{ [V]} & \text{for } t = 0^- \\ 2 - 2e^{-t} + \frac{1}{2}e^{-2t} \text{ [V]} & \text{for } t \geq 0 \end{cases} \end{aligned} \quad (\text{E4.1.6})$$

These results might be obtained by using the time-domain v - i relationships (3.1a) and (3.4b):

$$v_L(t) \stackrel{(3.1a)}{=} L \frac{di(t)}{dt} \stackrel{(E4.1.4)}{=} \frac{1}{2} [2(-1)e^{-t} - (-2)e^{-2t}] = -e^{-t} + e^{-2t} \text{ [V]} \quad \text{for } t \geq 0 \quad (E4.1.7)$$

$$\begin{aligned} v_C(t) &\stackrel{(3.4b)}{=} \frac{1}{C} \int_{-\infty}^t i(\tau) d\tau = \frac{1}{C} \int_{-\infty}^t i(\tau) d\tau + \frac{1}{C} \int_0^t i(\tau) d\tau \\ &\stackrel{(E4.1.4)}{=} v_C(0) + \frac{1}{C} \int_0^t (2e^{-\tau} - e^{-2\tau}) d\tau \stackrel{(F.33)}{=} V_0 - 2e^{-\tau} \Big|_0^t + 2\frac{1}{2}e^{-2\tau} \Big|_0^t \\ &= \frac{1}{2} - 2(e^{-t} - 1) + \frac{1}{2}(e^{-2t} - 1) = 2 - 2e^{-t} + \frac{1}{2}e^{-2t} \text{ [V]} \quad \text{for } t \geq 0 \end{aligned} \quad (E4.1.8)$$

However, this method does not seem to be the first choice, because it takes more time and effort than the Laplace transform approach.

(b) $R = 1 \Omega$, $L = 1/2 \text{ H}$, and $C = 2 \text{ F} \rightarrow D = (R/L)^2 - 4/(LC) = 0$ (critically damped)

The transformed mesh current (4.16) is expanded into the partial fraction form as

$$I(s) = \frac{[sV_i(s) - V_0]/L + I_0s}{s^2 + sR/L + 1/(LC)} = \frac{3 + s}{s^2 + 2s + 1} = \frac{K_1}{s + 1} + \frac{K_2}{(s + 1)^2} \quad (E4.1.9)$$

where the coefficients are obtained by using the formula (A.28) as

$$K_1 \stackrel{(A.28b)}{=} \frac{d}{ds}(s + 1)^2 I(s) \Big|_{s=-1} = \frac{d}{ds}(s + 3) \Big|_{s=-1} = 1 \quad (E4.1.10a)$$

$$K_2 \stackrel{(A.28b)}{=} (s + 1)^2 I(s) \Big|_{s=-1} = s + 3 \Big|_{s=-1} = 2 \quad (E4.1.10b)$$

Thus the inverse Laplace transform of $I(s)$ is taken to get the mesh current $i(t)$ as

$$\begin{aligned} I(s) &= \frac{K_1}{s + 1} + \frac{K_2}{(s + 1)^2} = \frac{1}{s + 1} + \frac{2}{(s + 1)^2} \\ i(t) &= \mathcal{L}^{-1}\{I(s)\} \stackrel{\text{Table A.1(5),(6)}}{=} \begin{cases} I_0 = 1 \text{ [A]} & \text{for } t = 0^- \\ e^{-t} + 2te^{-t} \text{ [A]} & \text{for } t \geq 0 \end{cases} \end{aligned}$$

The s -domain V - I relationships (3.15a) and (3.16b) are used to obtain the voltages across the inductor and capacitor as follows:

$$\begin{aligned} V_L(s) &\stackrel{(3.15a)}{=} sLI(s) - Li_L(0) \stackrel{(E4.1.1),(E4.1.9)}{=} \frac{s(s + 3)}{2(s + 1)^2} - \frac{1}{2} = \frac{1/2}{s + 1} - \frac{1}{(s + 1)^2} \\ v_L(t) &= \frac{1}{2}e^{-t} - te^{-t} \text{ [V]} \quad \text{for } t \geq 0 \end{aligned} \quad (E4.1.12)$$

$$\begin{aligned} V_C(s) &\stackrel{(3.16b)}{=} \frac{1}{sC} I(s) + \frac{v_C(0)}{s} \stackrel{(E4.1.1),(E4.1.9)}{=} \frac{s + 3}{2s(s + 1)^2} + \frac{1/2}{s} \\ &= \frac{3/2}{s} - \frac{1}{(s + 1)^2} - \frac{3/2}{s + 1} + \frac{1/2}{s} \\ v_C(t) &= \mathcal{L}^{-1}\{V_C(s)\} \stackrel{\text{Table A.1(5),(6)}}{=} \begin{cases} V_0 = \frac{1}{2} \text{ [V]} & \text{for } t = 0^- \\ 2 - \frac{3}{2}e^{-t} - te^{-t} \text{ [V]} & \text{for } t \geq 0 \end{cases} \end{aligned} \quad (E4.1.13)$$

These results could be obtained by using the time-domain v - i relationships (3.1a) and (3.4b) of the inductor and capacitor. However, the method is not recommended since it takes more time and effort.

(c) $R = 1\ \Omega$, $L = 1/2\ \text{H}$, and $C = 1\ \text{F} \rightarrow D = (R/L)^2 - 4/(LC) < 0$ (underdamped)

The transformed mesh current (4.16) can be decomposed into the following form:

$$I(s) = \frac{[sV_1(s) - V_0]/L + I_0s}{s^2 + sR/L + 1/LC} = \frac{3 + s}{s^2 + 2s + 2} = \frac{K_1(s+1)}{(s+1)^2 + 1^2} + \frac{K_2 \times 1}{(s+1)^2 + 1^2} \quad (\text{E4.1.14})$$

where the inverse Laplace transform of each term can be found from the Laplace transform table. Instead of the formula (A.28), we use the coefficient comparison method, i.e. make the terms on the right-hand side (RHS) have a common denominator and equate the numerators on both sides to find the coefficients as

$$s + 3 = K_1s + (K_1 + K_2); \quad K_1 = 1, K_2 = 2 \quad (\text{E4.1.15})$$

Thus the inverse Laplace transform of $I(s)$ is taken to get the mesh current $i(t)$ as

$$\begin{aligned} I(s) &= \frac{K_1(s+1)}{(s+1)^2 + 1^2} + \frac{K_2 \times 1}{(s+1)^2 + 1^2} = \frac{s+1}{(s+1)^2 + 1^2} + \frac{2 \times 1}{(s+1)^2 + 1^2} \\ i(t) &= \mathcal{L}^{-1}\{I(s)\} \stackrel{\text{Table A.1(9),(10)}}{=} \begin{cases} I_0 = 1\ [\text{A}] & \text{for } t = 0^- \\ e^{-t} \cos t + 2e^{-t} \sin t\ [\text{A}] & \text{for } t \geq 0 \end{cases} \quad (\text{E4.1.16}) \end{aligned}$$

The s -domain V - I relationships (3.15a) and (3.16b) are used to obtain the voltages across the inductor and capacitor as

$$\begin{aligned} V_L(s) &\stackrel{(3.15a)}{=} sL I(s) - L i_L(0) \stackrel{(\text{E4.1.1}), (\text{E4.1.14})}{=} \frac{s(s+3)}{2(s^2 + 2s + 2)} - \frac{1}{2} \\ &= \frac{1}{2} \left[\frac{s+1}{(s+1)^2 + 1^2} - \frac{3 \times 1}{(s+1)^2 + 1^2} \right] \\ v_L(t) &= \frac{1}{2} e^{-t} \cos t - \frac{3}{2} e^{-t} \sin t\ [\text{V}] \quad \text{for } t \geq 0 \quad (\text{E4.1.17}) \end{aligned}$$

$$\begin{aligned} V_C(s) &\stackrel{(3.16b)}{=} \frac{1}{sC} I(s) + \frac{v_C(0)}{s} \stackrel{(\text{E4.1.1}), (\text{E4.1.14})}{=} \frac{s+3}{s(s^2 + 2s + 2)} + \frac{1/2}{s} \\ &= \frac{K_0}{s} + \frac{K_1(s+1)}{s^2 + 2s + 2} + \frac{K_2 \times 1}{s^2 + 2s + 2} + \frac{1/2}{s} \\ &= \frac{3/2}{s} - \frac{(3/2)(s+1)}{s^2 + 2s + 2} - \frac{(1/2) \times 1}{s^2 + 2s + 2} + \frac{1/2}{s} = \frac{2}{s} - \frac{(3/2)(s+1)}{s^2 + 2s + 2} - \frac{(1/2) \times 1}{s^2 + 2s + 2} \\ v_C(t) &= \mathcal{L}^{-1}\{V_C(s)\} \stackrel{\text{Table A.1(3),(9),(10)}}{=} \begin{cases} V_0 = \frac{1}{2}\ [\text{V}] & \text{for } t = 0^- \\ 2 - \frac{3}{2} e^{-t} \cos t - \frac{1}{2} e^{-t} \sin t\ [\text{V}] & \text{for } t \geq 0 \end{cases} \quad (\text{E4.1.18}) \end{aligned}$$

Here the formula (A.28a) is used to find K_0 as

$$K_0 \stackrel{(\text{A.28a})}{=} s \frac{s+3}{s(s^2 + 2s + 2)} \Big|_{s=0} = \frac{3}{2}$$

and then the coefficient comparison method is used; i.e. we make the terms on the RHS have the common denominator and equate the numerators on both sides to write a set of equations and solve it

for the coefficients as follows:

$$\begin{aligned} s + 3 &= K_0(s^2 + 2s + 2) + s(K_1(s + 1) + K_2) \\ &= (K_0 + K_1)s^2 + (2K_0 + K_1 + K_2)s + 2K_0 \end{aligned} \quad (\text{E4.1.19})$$

The coefficient of the second-degree term: $K_0 + K_1 = 0$; $K_1 = -K_0 = -3/2$

The coefficient of the first-degree term: $2K_0 + K_1 + K_2 = 1$; $K_2 = 1 - 2K_0 - K_1 = -1/2$

The coefficient of the constant term: $2K_0 = 3$; $K_0 = 3/2$ (for crosscheck)

The following statements can be typed into the MATLAB command window to get the same result:

```
>> A=[1 1 0; 2 1 1; 2 0 0] ; b=[0;1;3] ;
>> K=A\b
    K = 1.5000
        -1.5000
        -0.5000
```

Note. MATLAB could help much more than just solving a set of equations, which will be discussed at the end of this example.

(d) $R = 0\Omega$, $L = 1/2\text{H}$, and $C = 1/2\text{F} \rightarrow D = (R/L)^2 - 4/(LC) < 0$ (undamped)

The transformed mesh current (4.16) is decomposed into the following form:

$$I(s) = \frac{[sV_i(s) - V_0]/L + I_0s}{s^2 + sR/L + 1/(LC)} = \frac{3 + s}{s^2 + 4} = \frac{s}{s^2 + 2^2} + \frac{(3/2) \times 2}{s^2 + 2^2} \quad (\text{E4.1.20})$$

the inverse Laplace transform of which is

$$i(t) = \mathcal{L}^{-1}\{I(s)\} \stackrel{\text{Table A.1(7),(8)}}{=} \begin{cases} I_0 = 1 [\text{A}] & \text{for } t = 0^- \\ \cos 2t + (3/2) \sin 2t [\text{A}] & \text{for } t \geq 0 \end{cases} \quad (\text{E4.1.21})$$

The s -domain V - I relationships (3.15a) and (3.16b) are used to obtain the voltages across the inductor and capacitor as

$$\begin{aligned} V_L(s) &\stackrel{(3.15a)}{=} sLI(s) - Li_L(0) \stackrel{(\text{E4.1.1}),(\text{E4.1.20})}{=} \frac{s(s+3)}{2(s^2+4)} - \frac{1}{2} = \frac{1}{2} \left(\frac{3s}{s^2+2^2} - \frac{2 \times 2}{s^2+2^2} \right) \\ v_L(t) &= \frac{3}{2} \cos 2t - \sin 2t [\text{V}] \quad \text{for } t \geq 0 \end{aligned} \quad (\text{E4.1.22})$$

$$\begin{aligned} V_C(s) &\stackrel{(3.16b)}{=} \frac{1}{sC} I(s) + \frac{v_C(0)}{s} \stackrel{(\text{E4.1.1}),(\text{E4.1.20})}{=} \frac{2(s+3)}{2(s^2+4)} + \frac{1/2}{s} = \frac{2}{s} - \frac{(3/2)s-2}{2(s^2+2^2)} \\ v_C(t) &= \mathcal{L}^{-1}\{V_C(s)\} \stackrel{\text{Table A.1(3),(7),(8)}}{=} \begin{cases} V_0 = \frac{1}{2} [\text{V}] & \text{for } t = 0^- \\ 2 - \frac{3}{2} \cos 2t + \sin 2t [\text{V}] & \text{for } t \geq 0 \end{cases} \end{aligned} \quad (\text{E4.1.23})$$

(e) Compose the following MATLAB program, save it as an M-file named `cir04e01.m`, and run it to get the solutions and plot them for all the cases given above as depicted in Figure 4.2.2.

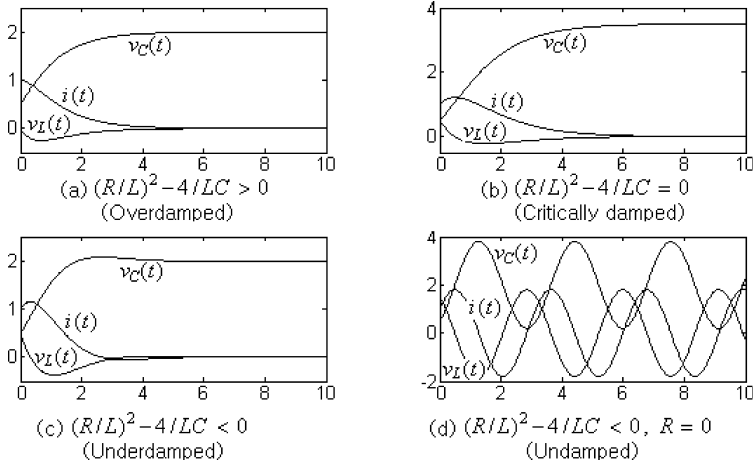


Figure 4.2.2 The output voltage/current of the circuit depicted in Figure 4.2.1 (Example 4.1)

```
%cir04e01.m for Example 4.1
clear, clf
syms s; Vi=2; Vis=Vi/s; I0=1; V0=1/2;
tt=[0:500]*0.02; % the time vector for the time interval [0,10]
for m=1:4
    if m==1, R=3/2; L=1/2; C=1;
    elseif m==2, R=1; L=1/2; C=2;
    elseif m==3, R=1; L=1/2; C=1;
    else R=0; L=1/2; C=1/2;
end
Is = ((s*Vis-V0)/L + I0*s) / (s^2 + s*R/L + 1/L/C); % Eq. (4.16)
i = ilaplace(Is) % the inverse Laplace transform i(t)
VLs = s*L*Is - L*I0; vL = ilaplace(VLs) % the inductor voltage
VCs = Is/s/C + V0/s; vC = ilaplace(VCs) % the capacitor voltage
for n=1:length(tt)
    t=tt(n); it(n)=eval(i); vLt(n)=eval(vL); vCt(n)=eval(vC);
end
subplot(220+m), plot(tt,it, tt,vLt, tt,vCt), hold on
end
```

Note. The result makes us happy with the Laplace transform and MATLAB or equivalent software. On the other hand it makes us feel sorry for those who have not experienced the amazing usefulness and convenience of such tools.

(Example 4.2) A Series RLC Circuit for Arcing (Ignition)

Consider the series RLC circuit for an ignition system of Figure 4.3.1(a) in which the values of the voltage source, the resistor R , the inductor L , and the capacitor C are

$$v_i(t) = V_i u_s(t) = 12u_s(t) \text{ [V]} \rightarrow V_i(s) = \frac{12}{s}, \quad R = 3 \Omega, \quad L = 0.01 \text{ H, and } C = 10^{-6} \text{ F} \quad (\text{E4.2.1})$$

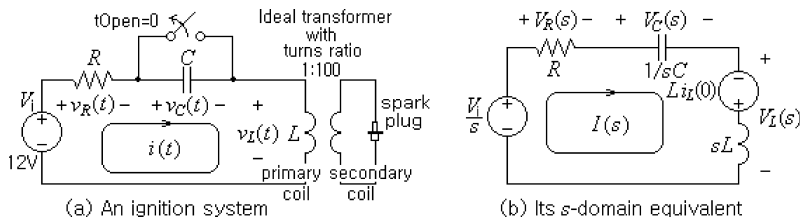


Figure 4.3.1 The RLC circuit for Example 4.2

respectively. In this circuit the voltage across the primary coil of the transformer (with a turns ratio of 1:100) is stepped up to 100 times across the secondary coil, which is expected to be high enough to initiate an arc discharge across the spark plug gap. We will find the voltages across the inductor L and the capacitor C after $t = 0$ when the switch is opened. Note the following point:

As will be discussed in the next chapter about magnetically coupled coils, a transformer can step up/down only the AC voltage, varying with time in its magnitude and polarity. How can the transformer step up the voltage (across the primary coil) in this circuit having only a DC voltage source? It is made possible by an almost undamped RLC circuit with the characteristic roots located close to the $j\omega$ axis ($\omega_d \gg \sigma$), which produces oscillatory (AC-like) voltages across the inductor (see Figure 4.1(3)).

To make a quantitative analysis of this circuit for finding the voltages across the inductor and the capacitor, it is supposed that the switch across the capacitor has been closed for a long time until $t = 0$ when the switch is opened. Then at $t = 0^-$, the circuit is expected to reach DC steady state, where the inductor acts like a short circuit so that the mesh current through R -closed SW- L is

$$i(0) = i_L(0^-) = \frac{V_i}{R} = \frac{12}{3} = 4 \text{ A} \quad (\text{E4.2.2})$$

The capacitor voltage at $t = 0^-$ is $v_C(0^-) = 0 \text{ V}$ since the capacitor has been shorted by the closed switch. Note the following:

1. These values of $i_L(0^-)$ and $v_C(0^-)$ are the final (steady state) values for the circuit with the switch closed for $t < 0$ and are also the initial values for the circuit with the switch opened at $t = 0$ because of the continuity rules on the inductor current and the capacitor voltage.
 2. The s -domain equivalent of this circuit with the initial inductor current $i_L(0)$ represented by a (transformed) voltage source of $L i_L(0)$ (see Figure 3.6(b1)) is shown in Figure 4.3.1(b).
- (a) Find the inductor voltage $v_L(t)$ and its maximum amplitude to make sure that the voltage induced across the secondary coil will be high enough to produce an arc in the air gap of the spark plug. Applying KVL to the transformed circuit in Figure 4.3.1(b) yields the mesh equation as

$$\left(R + sL + \frac{1}{sC}\right)I(s) = \frac{V_i}{s} + L i_L(0) \quad (\text{E4.2.3})$$

This equation is solved to get the mesh current as

$$I(s) = \frac{V_i/L + s i(0)}{s^2 + sR/L + 1/(LC)} \quad (\text{E4.2.4})$$

and then Equation (3.15a) is used to obtain the voltage across the inductor as

$$\begin{aligned} V_L(s) &\stackrel{(3.15a)}{=} sLI(s) - L i_L(0) \stackrel{(E4.2.4)}{=} \frac{sV_i + s^2 L i(0) - L i(0) [s^2 + sR/L + 1/(LC)]}{s^2 + sR/L + 1/(LC)} \\ &\stackrel{(E4.2.2)}{=} \frac{sV_i - L(V_i/R) [sR/L + 1/(LC)]}{s^2 + sR/L + 1/(LC)} = \frac{-V_i/(RC)}{s^2 + sR/L + 1/(LC)} \end{aligned} \quad (E4.2.5)$$

On the premise that

$$\left(\frac{R}{L}\right)^2 - \frac{4}{LC} \ll 0 \quad (\text{underdamped}) \quad (E4.2.6)$$

so that the characteristic equation $s^2 + sR/L + 1/(LC) = 0$ has complex roots, the inverse Laplace transform of Equation (E4.2.5) yields the inductor voltage as follows:

$$V_L(s) \stackrel{(E4.2.5)}{=} \frac{-[V_i/(\omega_d RC)]\omega_d}{(s + \sigma)^2 + \omega_d^2} \quad (E4.2.7)$$

$$\text{with } \sigma = R/(2L) = 3/(2 \times 0.01) = 150, \quad \omega_d = \sqrt{1/(LC) - [R/(2L)]^2} \approx 10^4$$

$$v_L(t) = \mathcal{L}^{-1}\{V_L(s)\} \stackrel{\text{Table A.1(9)}}{=} -\frac{V_i}{\omega_d RC} e^{-\sigma t} \sin(\omega_d t) \quad \text{for } t \geq 0 \quad (E4.2.8)$$

Now the time derivative of $v_L(t)$ is set to zero to find the peak time at which the absolute value of $v_L(t)$ is maximized:

$$\begin{aligned} \frac{dv_L(t)}{dt} &\stackrel{(E4.2.8)}{=} -\frac{V_i}{\omega_d RC} e^{-\sigma t} [-\sigma \sin(\omega_d t) + \omega_d \cos(\omega_d t)] = 0 \\ \tan(\omega_d t) &= \frac{\omega_d}{\sigma}; \quad t = \frac{1}{\omega_d} \left(\tan^{-1} \frac{\omega_d}{\sigma} + k\pi \right); \quad t_{\text{peak}} = \frac{1}{\omega_d} \tan^{-1} \frac{\omega_d}{\sigma} = 0.16 \text{ ms} \end{aligned} \quad (E4.2.9)$$

This peak time is within the first period of $2\pi/\omega_d \simeq 0.63 \text{ ms}$ and much earlier than the time constant $1/\sigma = 1/150 \simeq 6.7 \text{ ms}$. Therefore,

$$t_{\text{peak}} \ll \frac{1}{\sigma} \rightarrow \sigma t_{\text{peak}} \ll 1 \rightarrow e^{-\sigma t_{\text{peak}}} \simeq 1 \quad (E4.2.10)$$

which implies that the amplitude of oscillation decreases little at $t = t_{\text{peak}}$. The peak time (E4.2.9) can be substituted for t into Equation (E4.2.8) to find the maximum amplitude of the inductor voltage as

$$\begin{aligned} V_{L,\text{peak}} &= |v_L(t_{\text{peak}})| = \frac{V_i}{\omega_d RC} e^{-\sigma t_{\text{peak}}} \sin(\omega_d t_{\text{peak}}) \stackrel{(E4.2.10)}{\simeq} \frac{V_i}{\omega_d RC} \sin(\tan^{-1} \frac{\omega_d}{\sigma}) \\ &= \frac{V_i}{\omega_d RC} \frac{\omega_d}{\sqrt{\sigma^2 + \omega_d^2}} = \frac{V_i}{RC \sqrt{1/(LC)}} = \frac{V_i}{R} \sqrt{\frac{L}{C}} = \frac{12}{3} \sqrt{\frac{10^{-2}}{10^{-6}}} = 400 \text{ V} \end{aligned} \quad (E4.2.11)$$

This voltage across the primary coil of the transformer (with a turns ratio of 1:100) is stepped up to 100 times across the secondary coil so that the maximum amplitude of the voltage across the spark plug gap will be

$$v_{\text{sp,peak}} = 100 \times 400 = 40 \text{ kV} \quad (E4.2.12)$$

This voltage may be high enough to break the dielectric strength of the air (in the gap of the spark plug), amounting to about 3 kV/mm. All these computations as well as the plotting job are

```

%cir04e02a.m for Example 4.2 (a)
clear, clf
Vi=12; R=3; L=0.01; C=1e-6; I0=Vi/R; V0=0;
syms s, Vis=Vi/s; Is = (s*Vis-V0)/L + I0*s)/(s^2 + s*R/L + 1/L/C); %Eq. (4.16)
VLs = s*L*Is - L*I0; vL = ilaplace(VLs) % the inductor voltage
dvL = diff(vL), pretty(dvL) % the time derivative of vL(t)
dvL = inline('6e4*exp(-150*t)*sin(1e4*t)-4e6*exp(-150*t)*cos(1e4*t)', 't');
tpeak = fsolve(dvL, 1e-4, optimset('fsolve')) % the peak time Eq. (E4.2.9)
t = tpeak; vLmax = eval(vL) % the peak (maximum) amplitude of vL
t0=0; tf=0.01; N=500;
tt = t0+[0:N]/N*(tf-t0); % the time vector for the time interval [0,0.01s]
for n=1:length(tt)
    t = tt(n); vLt(n) = eval(vL);
end
sigma = R/2/L; wd = sqrt(1/L/C-sigma^2)
vLt1 = -Vi/(wd*R*C)*exp(-sigma*tt).*sin(wd*tt); % Eq. (E4.2.8)
plot(tt, real(vLt), tt, vLt1, 'k:', tpeak*[1 1], [0 vLmax], 'r:')

```

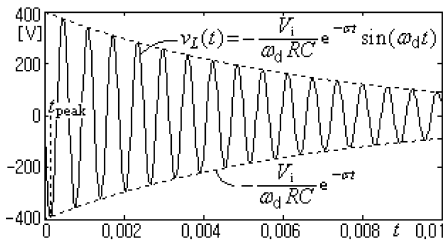
performed in the following MATLAB program `cir04e02a.m`. Figures 4.3.2(a) and (b) show $v_L(t)$ obtained by running this program and that obtained from the PSpice simulation, respectively.

- (b) Find the capacitor voltage $v_C(t)$ and its maximum amplitude to see that it is not so high as to produce an arc between the two contacts of the switch in parallel with the capacitor.

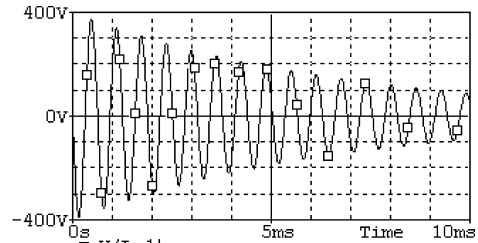
Noting that the (transformed) mesh current has been obtained as Equation (E4.2.4), Equation (3.16b) is used to obtain the voltage across the capacitor as

$$\begin{aligned}
 V_C(s) &\stackrel{(3.16b)}{=} \frac{1}{sC}I(s) + \frac{v_C(0)}{s} \stackrel{(E4.2.4)}{=} \frac{V_i/LC + sV_i/(RC)}{v_C(0)=0 \left[s^2 + sR/L + 1/(LC) \right]} = V_i \left[\frac{1}{s} - \frac{s + R/L - 1/(RC)}{s^2 + sR/L + 1/(LC)} \right] \\
 &= V_i \left(\frac{1}{s} - \frac{(s + \sigma) + \{[\sigma - 1/(RC)]/\omega_d\}\omega_d}{(s + \sigma)^2 + \omega_d^2} \right) \quad (E4.2.13)
 \end{aligned}$$

$$v_C(t) = \mathcal{L}^{-1}\{V_C(s)\} \stackrel{\text{Table A.1(3),(9),(10)}}{=} V_i - V_i e^{-\sigma t} \left[\cos(\omega_d t) + \frac{\sigma - 1/(RC)}{\omega_d} \sin(\omega_d t) \right] \quad \text{for } t \geq 0 \quad (E4.2.14)$$



(a) $v_L(t)$ plotted using MATLAB simulation



(b) $v_L(t)$ obtained from the PSpice simulation

Figure 4.3.2 The simulation results for the circuit in Figure 4.3.1

The time derivative of $v_C(t)$ is now set to zero in order to find the peak time at which the absolute value of $v_C(t)$ is maximized, where $dv_C(t)/dt$ is obtained from taking the inverse Laplace transform of $\mathcal{L}\{dv_C(t)/dt\} = sV_C(s) - v_C(0)$:

$$\begin{aligned}\mathcal{L}\left\{\frac{dv_C(t)}{dt}\right\} &\stackrel{\text{Table A.2(5)}}{=} sV_C(s) - v_C(0) \stackrel{(3.16b)}{=} s\left[\frac{1}{sC}I(s) + \frac{v_C(0)}{s}\right] - v_C(0) = \frac{1}{C}I(s) \\ &\stackrel{(E4.2.4)}{=} \frac{V_i/(LC) + sV_i/(RC)}{s^2 + sR/L + 1/(LC)} = \frac{V_i}{2\sigma LC} \frac{(s + \sigma) + (\sigma/\omega_d)\omega_d}{(s + \sigma)^2 + \omega_d^2} \\ \frac{dv_C(t)}{dt} &= \mathcal{L}^{-1}\left\{\frac{1}{C}I(s)\right\} \stackrel{\text{Table A.1(9),(10)}}{=} \frac{V_i}{2\sigma LC} e^{-\sigma t} \left[\cos(\omega_d t) + \frac{\sigma}{\omega_d} \sin(\omega_d t)\right] = 0 \\ \omega_d t &= \tan^{-1}\left(-\frac{\omega_d}{\sigma}\right) + k\pi; \quad t_{\text{peak},C} = \frac{1}{\omega_d} \left(\pi - \tan^{-1} \frac{\omega_d}{\sigma}\right) \simeq 0.16 \text{ ms}\end{aligned}\quad (\text{E4.2.15})$$

Noting that this peak time is also within the first period of $2\pi/\omega_d \simeq 0.63 \text{ ms}$ and much earlier than the time constant $1/\sigma \simeq 1/150 \simeq 6.7 \text{ ms}$ so that the amplitude of oscillation decreases little at $t = t_{\text{peak},C}$, the maximum amplitude of the capacitor voltage can be found as

$$\begin{aligned}V_{C,\text{peak}} &= |v_C(t_{\text{peak},C})| \stackrel{(E4.2.14)}{=} V_i - V_i e^{-\sigma t} \left[\cos(\omega_d t_{\text{peak},C}) + \frac{\sigma - 1/(RC)}{\omega_d} \sin(\omega_d t_{\text{peak},C})\right] \\ &\left(\begin{aligned}\cos(\omega_d t_{\text{peak},C}) &\stackrel{(E4.2.15)}{=} \cos\left(\pi - \tan^{-1} \frac{\omega_d}{\sigma}\right) = -\cos\left(\tan^{-1} \frac{\omega_d}{\sigma}\right) = \frac{-\sigma}{\sqrt{\sigma^2 + \omega_d^2}} \\ \sin(\omega_d t_{\text{peak},C}) &\stackrel{(E4.2.15)}{=} \sin\left(\pi - \tan^{-1} \frac{\omega_d}{\sigma}\right) = \sin\left(\tan^{-1} \frac{\omega_d}{\sigma}\right) = \frac{\omega_d}{\sqrt{\sigma^2 + \omega_d^2}}\end{aligned}\right) \\ &\stackrel{(E4.2.15)}{\simeq} V_i \left[1 - \frac{-\sigma}{\sqrt{\sigma^2 + \omega_d^2}} - \frac{\sigma - 1/(RC)}{\omega_d} \frac{\omega_d}{\sqrt{\sigma^2 + \omega_d^2}}\right] \\ &= V_i \frac{\sqrt{1/(LC)} + 1/(RC)}{\sqrt{1/(LC)}} \simeq \frac{V_i}{R} \sqrt{\frac{L}{C}} = \frac{12}{3} \sqrt{\frac{10^{-2}}{10^{-6}}} = 400 \text{ V} \\ &\left(\because \frac{1}{RC} = \frac{1}{3 \times 10^{-6}} = 333\,333 \gg \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{10^{-2} \times 10^{-6}}} = 10\,000\right)\end{aligned}\quad (\text{E4.2.16})$$

which is close to the amplitude of the inductor voltage $v_L(t)$.

What is the minimum distance between two contacts of the switch in parallel with the capacitor such that the dielectric strength of the air between them is not broken by $V_{C,\text{peak}} = 400 \text{ V}$? It is

$$d_{\text{sw,min}} = \frac{400 \text{ V}}{3000 \text{ V/mm}} = 0.133 \text{ mm} \quad (\text{E4.2.17})$$

Readers are invited to compose a MATLAB program `cir04e02b.m`, which performs all these computations as well as the plotting job.

Note. It is interesting to note that the ratios of the amplitudes of the voltages across the inductor and the capacitor to that of the input voltage source V_i are commonly close to the voltage magnification ratio given by Equation (8.19) as

$$Q = \frac{v_{L,\text{peak}}}{V_i} \simeq \frac{v_{C,\text{peak}}}{V_i} \stackrel{(E4.2.11),(E4.2.16)}{\simeq} \frac{1}{R} \sqrt{\frac{L}{C}} \quad (4.18)$$

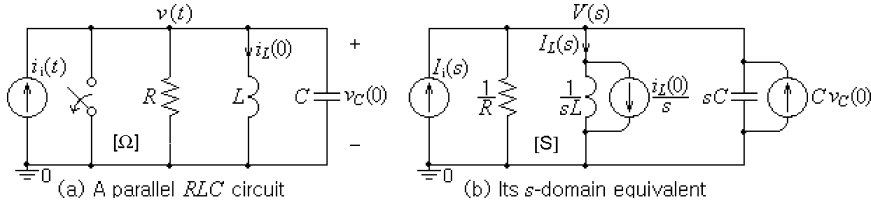


Figure 4.4.1 The circuit for Example 4.3

4.2.2 A Parallel RLC Circuit

Consider the circuit of Figure 4.4.1(a) in which the initial values of the inductor current and the capacitor voltage are

$$i_L(0) = I_0 \quad \text{and} \quad v_C(0) = V_0$$

respectively. To find the voltage $v(t)$ at the top node, KCL can be applied to the top node to set up the node equation in the time domain as

$$i_R(t) + i_L(t) + i_C(t) = \frac{v(t)}{R} + \frac{1}{L} \int_{-\infty}^t v(\tau) d\tau + C \frac{dv(t)}{dt} = i_i(t)$$

and its Laplace transform (Table A.2(5) and (6)) taken to write the transformed node equation as

$$\begin{aligned} \frac{V(s)}{R} + \frac{1}{L} \left[\frac{1}{s} V(s) + \frac{1}{s} \int_{-\infty}^0 v(\tau) d\tau \right] + C[sV(s) - V_0] &= I_i(s) \\ \left(\frac{1}{R} + \frac{1}{sL} + sC \right) V(s) &= I_i(s) - \frac{1}{s} \left[\frac{1}{L} \int_{-\infty}^0 v(\tau) d\tau \right] + CV_0 = I_i(s) - \frac{I_0}{s} + CV_0 \end{aligned} \quad (4.19)$$

A better way to get this equation is to transform the circuit into its s -domain equivalent, as depicted in Figure 4.4.1(b), and apply KCL to the top node to write the node equation (4.19) directly. In either case, Equation (4.19) is solved to obtain the transformed node voltage as

$$V(s) = \frac{I_i(s) - I_0/s + CV_0}{1/R + sC + 1/(sL)} = \frac{[sI_i(s) - I_0]/C + V_0s}{s^2 + s/(RC) + 1/(LC)} \quad (4.20)$$

and its inverse Laplace transform is taken to find $v(t)$.

If the currents through the inductor/capacitor are needed, they can be found by using the V - I relationships (3.15b) and (3.16a):

$$\begin{aligned} I_L(s) &\stackrel{(3.15b)}{=} \frac{1}{sL} V(s) + \frac{i_L(0)}{s} \\ I_C(s) &\stackrel{(3.16a)}{=} sCV(s) - C v_C(0) \end{aligned}$$

Note. Care should be taken here not to make the a mistake of missing out the initial condition terms.

Note that the transfer function of this circuit with the source current as the input and the node voltage as the output is

$$G(s) = \frac{V(s)}{I_1(s)} \bigg|_{\substack{\text{with zero initial conditions} \\ V_0=0, I_0=0}} = \frac{s/C}{s^2 + s/(RC) + 1/(LC)}$$

and the characteristic equation obtained by setting its denominator to zero is

$$s^2 + \frac{s}{RC} + \frac{1}{LC} = 0 \quad (4.21)$$

(Example 4.3) Time Responses of a Parallel RLC Circuit

Consider the parallel RLC circuit of Figure 4.4.1(a) in which the source current and the initial conditions of the inductor and capacitor are

$$i_i(t) = 50 \sin 2t \, u_s(t) \, [\text{A}] \xrightarrow{\text{Table A.1(7)}} I_1(s) = \frac{50 \times 2}{s^2 + 2^2}, \quad I_0 = 0 \, \text{A}, \quad \text{and} \quad V_0 = 0 \, \text{V} \quad (\text{E4.3.1})$$

respectively. Noting that the discriminant of the characteristic equation (4.21) is $D = [1/(RC)]^2 - 4/(LC)$, find the node voltage for four different sets of values of R , L , and C .

(a) $R = 2/3 \, \Omega$, $C = 1/2 \, \text{F}$, and $L = 1 \, \text{H} \rightarrow D = [1/(RC)]^2 - 4/(LC) > 0$ (overdamped)

The transformed node voltage (4.20) is decomposed into the following form:

$$\begin{aligned} V(s) &= \frac{[s I_1(s) - I_0]/C + V_0 s}{s^2 + s/(RC) + 1/(LC)} = \frac{200s}{(s^2 + 2^2)(s^2 + 3s + 2)} \\ &= \frac{K_1}{s+1} + \frac{K_2}{s+2} + \frac{K_3 s + K_4 \times 2}{s^2 + 2^2} \end{aligned} \quad (\text{E4.3.2})$$

where the coefficients are obtained by using the formula (A.28) in Appendix A together with the coefficient comparison method as

$$K_1 \stackrel{(\text{A.28a})}{=} (s+1)V(s)|_{s=-1} = (s+1) \frac{200s}{(s^2 + 2^2)(s+1)(s+2)} \bigg|_{s=-1} = -40 \quad (\text{E4.3.3a})$$

$$K_2 \stackrel{(\text{A.28a})}{=} (s+2)V(s)|_{s=-2} = (s+2) \frac{200s}{(s^2 + 2^2)(s+1)(s+2)} \bigg|_{s=-2} = 50 \quad (\text{E4.3.3b})$$

$$\begin{aligned} 200s &= K_1(s+2)(s^2 + 2^2) + K_2(s+1)(s^2 + 2^2) + (K_3s + 2K_4)(s^2 + 3s + 2) \\ &= (K_1 + K_2 + K_3)s^3 + (2K_1 + K_2 + 3K_3 + 2K_4)s^2 \\ &\quad + (4K_1 + 4K_2 + 2K_3 + 6K_4)s + (8K_1 + 4K_2 + 4K_4) \end{aligned} \quad (\text{E4.3.3c})$$

The coefficient of the third-degree term: $K_1 + K_2 + K_3 = 0$, $K_3 = -K_1 - K_2 = 40 - 50 = -10$

The coefficient of the zeroth-degree term: $8K_1 + 4K_2 + 4K_4 = 0$, $K_4 = -2K_1 - K_2 = 30$

The coefficient of the second-degree term: $2K_1 + K_2 + 3K_3 + 2K_4 = 0$ (for crosscheck)

The coefficient of the first-degree term: $4K_1 + 4K_2 + 2K_3 + 6K_4 = 200$ (for crosscheck)

Thus the inverse Laplace transform of $V(s)$ is taken to get the node voltage $v(t)$ as

$$V(s) = \frac{K_1}{s+1} + \frac{K_2}{s+2} + \frac{K_3 s + K_4 \times 2}{s^2 + 2^2} = \frac{-40}{s+1} + \frac{50}{s+2} + \frac{-10s + 30 \times 2}{s^2 + 2^2}$$

$$v(t) = \mathcal{L}^{-1}\{V(s)\} \stackrel{\text{Table A.1(5),(7),(8)}}{=} -40e^{-t} + 50e^{-2t} - 10 \cos 2t + 30 \sin 2t [\text{V}] \quad \text{for } t \geq 0 \quad (\text{E4.3.4})$$

Noting that this node voltage is applied across the inductor and the capacitor in parallel, the s -domain V - I relationships (3.15b) and (3.16a) or the time-domain v - i relationships (3.1b) and (3.4a) could be used to obtain the currents through each of them if necessary.

(b) $R = 1 \Omega$, $C = 1/2 \text{ F}$, and $L = 2 \text{ H} \rightarrow D = [1/(RC)]^2 - 4/(LC) = 0$ (critically damped)

The transformed node voltage (4.20) is decomposed into the following form:

$$V(s) = \frac{[sI_i(s) - I_0]/C + V_0 s}{s^2 + s/(RC) + 1/(LC)} = \frac{200s}{(s^2 + 2^2)(s^2 + 2s + 1)} = \frac{K_1}{s+1} + \frac{K_2}{(s+1)^2} + \frac{K_3 s + K_4 \times 2}{s^2 + 2^2} \quad (\text{E4.3.5})$$

where

$$K_1 \stackrel{(\text{A.28b})}{=} \frac{d}{ds}(s+1)^2 V(s) \Big|_{s=-1} = \frac{d}{ds} \left(\frac{200s}{s^2 + 2^2} \right) \Big|_{s=-1} = 200 \frac{(s^2 + 4) - s \times 2s}{(s^2 + 4)^2} \Big|_{s=-1} = 24 \quad (\text{E4.3.6a})$$

$$K_2 \stackrel{(\text{A.28b})}{=} (s+1)^2 V(s) \Big|_{s=-1} = \frac{200s}{s^2 + 2^2} \Big|_{s=-1} = -40 \quad (\text{E4.3.6b})$$

$$\begin{aligned} 200s &= K_1(s+1)(s^2 + 2^2) + K_2(s^2 + 2^2) + (K_3 s + 2K_4)(s^2 + 2s + 1) \\ &= (K_1 + K_3)s^3 + (K_1 + K_2 + 2K_3 + 2K_4)s^2 \\ &\quad + (4K_1 + K_3 + 4K_4)s + (4K_1 + 4K_2 + 2K_4) \end{aligned} \quad (\text{E4.3.6c})$$

The coefficient of the third-degree term: $K_1 + K_3 = 0$, $K_3 = -K_1 = -24$

The coefficient of the zeroth-degree term: $4K_1 + 4K_2 + 2K_4 = 0$, $K_4 = -2K_1 - 2K_2 = 32$

The coefficient of the second-degree term: $K_1 + K_2 + 2K_3 + 2K_4 = 0$ (for crosscheck)

The coefficient of the first-degree term: $4K_1 + K_3 + 4K_4 = 200$ (for crosscheck)

Thus the inverse Laplace transform of $V(s)$ is taken to get the node voltage $v(t)$ as

$$V(s) = \frac{K_1}{s+1} + \frac{K_2}{(s+1)^2} + \frac{K_3 s + K_4 \times 2}{s^2 + 2^2} = \frac{24}{s+1} + \frac{-40}{(s+1)^2} + \frac{-24s + 32 \times 2}{s^2 + 2^2}$$

$$v(t) = \mathcal{L}^{-1}\{V(s)\} \stackrel{\text{Table A.1(5),(6),(7),(8)}}{=} 24e^{-t} - 40te^{-t} - 24 \cos 2t + 32 \sin 2t [\text{V}] \quad \text{for } t \geq 0 \quad (\text{E4.3.7})$$

(c) $R = 1 \Omega$, $C = 1/2 \text{ F}$, and $L = 1 \text{ H} \rightarrow D = [1/(RC)]^2 - 4/(LC) < 0$ (underdamped)

The transformed node voltage (4.20) is decomposed into the following form:

$$\begin{aligned} V(s) &= \frac{[sI_i(s) - I_0]/C + V_0 s}{s^2 + s/(RC) + 1/(LC)} = \frac{200s}{(s^2 + 2^2)(s^2 + 2s + 2)} \\ &= \frac{K_1(s+1) + K_2 \times 1}{(s+1)^2 + 1^2} + \frac{K_3 s + K_4 \times 2}{s^2 + 2^2} \end{aligned} \quad (\text{E4.3.8})$$

where

$$\begin{aligned} 200s &= (K_1s + K_1 + K_2)(s^2 + 2^2) + (K_3s + 2K_4)(s^2 + 2s + 2) \\ &= (K_1 + K_3)s^3 + (K_1 + K_2 + 2K_3 + 2K_4)s^2 \\ &\quad + (4K_1 + 2K_3 + 4K_4)s + (4K_1 + 4K_2 + 4K_4) \end{aligned} \quad (\text{E4.3.9})$$

```
>> A=[1 0 1 0; 1 1 2 2; 4 0 2 4; 4 4 0 4]; b=[0 0 200 0]; K=A\ -1*b.'
K=20 -60 -20 40
```

Thus the inverse Laplace transform of $V(s)$ is taken to get the node voltage $v(t)$ as

$$\begin{aligned} V(s) &= \frac{K_1(s+1) + K_2 \times 1}{(s+1)^2 + 1^2} + \frac{K_3s + K_4 \times 2}{s^2 + 2^2} = \frac{20(s+1) - 60 \times 1}{(s+1)^2 + 1^2} + \frac{-20s + 40 \times 2}{s^2 + 2^2} \\ v(t) &= \mathcal{L}^{-1}\{V(s)\} \stackrel{\text{Table A.1(7),(8),(9),(10)}}{=} 20e^{-t} \cos t - 60e^{-t} \sin t - 20 \cos 2t + 40 \sin 2t [\text{V}] \quad \text{for } t \geq 0 \end{aligned} \quad (\text{E4.3.10})$$

```
>> syms s; v= ilaplace (200*s/(s^2+4)/(s^2+2*s+2))
v= -20*cos(2*t) + 40*sin(2*t) + 20*exp(-t)*cos(t) - 60*exp(-t)*sin(t)
```

(d) $R = \infty \Omega$ (open), $C = 1/2 \text{ F}$, and $L = 1/2 \text{ H} \rightarrow D = [1/(RC)]^2 - 4/(LC) < 0$ (undamped)

The transformed node voltage (4.20) is decomposed into the following form:

$$\begin{aligned} V(s) &= \frac{[sI_1(s) - I_0]/C + V_0s}{s^2 + s/(RC) + 1/(LC)} = \frac{200s}{(s^2 + 2^2)(s^2 + 1^2)} \\ &= \frac{K_1s + K_2 \times 1}{s^2 + 1^2} + \frac{K_3s + K_4 \times 2}{s^2 + 2^2} \end{aligned} \quad (\text{E4.3.11})$$

where

$$\begin{aligned} 200s &= (K_1s + K_2)(s^2 + 2^2) + (K_3s + 2K_4)(s^2 + 1) \\ &= (K_1 + K_3)s^3 + (K_2 + 2K_4)s^2 + (4K_1 + K_3)s + (4K_2 + 2K_4) \end{aligned} \quad (\text{E4.3.12})$$

```
>> A=[1 0 1 0; 0 1 0 2; 4 0 1 0; 0 4 0 2]; b=[0 0 200 0]; K=A\b.'
K= 66.6667 0 -66.6667 0
>> format rat, K % for fractional form of numeric values
K= 200/3 0 -200/3 0
```

Thus the inverse Laplace transform of $V(s)$ is taken to get the node voltage $v(t)$ as

$$\begin{aligned} V(s) &= \frac{(200/3)s}{s^2 + 1^2} + \frac{(-200/3)s}{s^2 + 2^2} \\ v(t) &= \mathcal{L}^{-1}\{V(s)\} \stackrel{\text{Table A.1(8)}}{=} (200/3) \cos t - (200/3) \cos 2t [\text{V}] \quad \text{for } t \geq 0 \end{aligned} \quad (\text{E4.3.13})$$

```
>> syms s; v= ilaplace (200*s/(s^2+4)/(s^2+1))
v= 200/3*cos(t) - 200/3*cos(2*t)
```

(e) You may compose the following MATLAB program, save it as an M-file named `cir04e03.m`, and run it to get the solutions and plot them for all the cases (a), (b), (c), and (d) given above, as depicted in Figure 4.4.2.

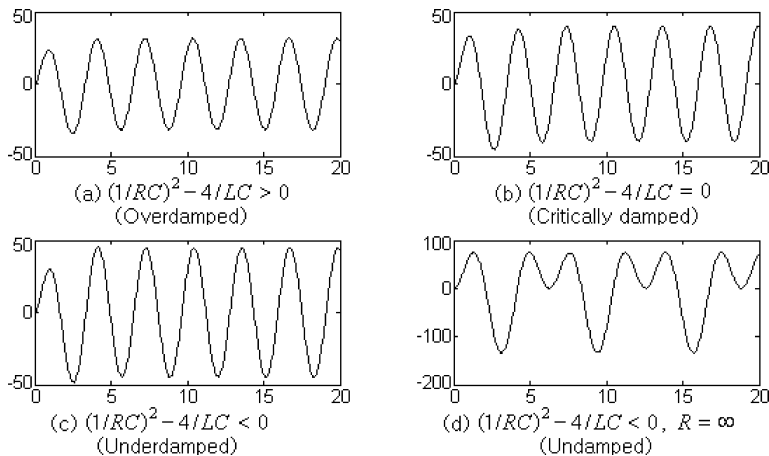


Figure 4.4.2 The output voltage of the circuit depicted in Figure 4.4.1 (Example 4.3)

```
%cir04e03.m
clear, clf
syms s
t0=0; tf=20; N=1000; tt = t0+[0:N] * (tf-t0)/N; % Simulation interval
Iis = 50*2/(s^2+2^2); V0=0; I0=0; % Eq. (E4.3.1)
for m=1:4
    if m==1, R=2/3; C=1/2; L=1;
    elseif m==2, R=1; C=1/2; L=2;
    elseif m==3, R=1; C=1/2; L=1;
    else R=inf; C=1/2; L=2;
    end
    G=1/R; Vs = ((s*Iis-I0)/C+V0*s)/(s^2+s*G/C+1/L/C); % Eq. (4.20)
    v=ilaplace(Vs) % the inverse Laplace transform
    for n=1:length(tt)
        t=tt(n); vt(n) = eval(v);
    end
    subplot(220+m), plot(tt, vt)
end
```

(Example 4.4) Design of a Parallel *RLC* Circuit for Triggering

Consider the parallel *RLC* circuit of Figure 4.5.1(a) in which the capacitor *C* is normally charged from the DC voltage source of 12 V when the switch is connected to position *a* and then is discharged to supply the stored energy to the resistor *R* = 3 Ω after the switch is moved to position *b*. The design objective is to determine the values of *L* and *C* such that *R* dissipates the energy more than 1 J during the first period of 0.1 s just after the switch is flipped to position *b*. More specifically, the voltage *v*(*t*) across or the current through *R* will be made to oscillate 5 times for one time constant of *T* = 0.5 s. This design specification can be expressed in terms of the parameters of the damping constant σ and the damped frequency ω_d as follows:

$$\sigma = \zeta\omega_r = \frac{1}{T} = \frac{1}{0.5} = 2 [1/\text{s}] \quad \text{and} \quad \omega_d = 2\pi\left(\frac{5}{T}\right) = 20\pi [\text{rad/s}] \quad (\text{E4.4.1})$$

If only the circuit conforms to this specification, the oscillatory voltage with an amplitude of about 12 V is expected to make about 2 J of energy dissipated in *R* = 3 Ω for 0.1 s:

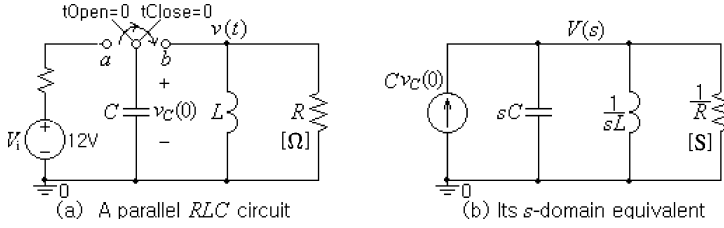


Figure 4.5.1 The circuit for Example 4.4

$$E_R \stackrel{(F.9)}{=} \int_0^{0.1} \frac{1}{R} v^2(\tau) d\tau \approx \frac{1}{3} \int_0^{0.1} (12 \sin \omega_d \tau)^2 d\tau \stackrel{(F.14)}{=} \frac{12^2}{6} \int_0^{0.1} (1 - \cos 2\omega_d \tau) d\tau = 2.4 \text{ J} \quad (\text{E4.4.2})$$

- (a) To determine the values of L and C such that the design specification is met, the characteristic equation (4.21) of the supposedly underdamped parallel RLC circuit will be written as

$$s^2 + \frac{1}{RC}s + \frac{1}{LC} = (s + \sigma)^2 + \omega_d^2 = s^2 + 2\sigma s + (\sigma^2 + \omega_d^2) \stackrel{(E4.4.1)}{=} s^2 + 4s + (4 + 400\pi^2) \quad (\text{E4.4.3})$$

This implies that the values of L and C should be determined as

$$\frac{1}{RC} = 4; \quad C = \frac{1}{4R} = \frac{1}{12} \text{ F} \quad (\text{E4.4.4a})$$

$$\frac{1}{LC} = 4 + 400\pi^2 \simeq 3951.8; \quad L = \frac{1}{3951.8 C} = \frac{12}{3951.8} \simeq 0.003 \text{ H} \quad (\text{E4.4.4b})$$

- (b) With the values of L and C determined in (a), find the node voltage $v(t)$ of the circuit. For the s -domain equivalent in Figure 4.5.1(b), the node voltage that is produced by the current source of $Cv_C(0)$ corresponding to the initial capacitor voltage is

$$\begin{aligned} V(s) &\stackrel{(4.20)}{=} \frac{C v_C(0)}{1/R + sC + 1/(sL)} = \frac{V_0 s}{s^2 + s/(RC) + 1/(LC)} = \frac{12s}{(s + \sigma)^2 + \omega_d^2} \\ &= \frac{K_1(s + \sigma)}{(s + \sigma)^2 + \omega_d^2} + \frac{K_2 \omega_d}{(s + \sigma)^2 + \omega_d^2} \\ \text{with } K_1 &= 12, \quad K_2 = -12\sigma/\omega_d = -24/20\pi = -0.382 \end{aligned} \quad (\text{E4.4.5})$$

Since the absolute value of the coefficient of the second term ($|K_2| = 0.382$) is much less than that of the first term ($|K_1| = 12$), the node voltage can be approximated by just the first term as

$$v(t) = \mathcal{L}^{-1}\{V(s)\} \stackrel{\text{Table A.1(10)}}{=} K_1 e^{-\sigma t} \cos \omega_d t = 12e^{-2t} \cos 20\pi t \text{ [V]} \quad (\text{E4.4.6})$$

- (c) With the node voltage obtained in (b), find the energy dissipated in R for the first period of 0.1 s:

$$\begin{aligned} E_R &= \int_0^{0.1} \frac{1}{R} v^2(t) dt \stackrel{(E4.4.6)}{\simeq} \frac{1}{3} \int_0^{0.1} (12e^{-2t} \cos 20\pi t)^2 dt \stackrel{(F.15)}{=} \frac{12^2}{6} \int_0^{0.1} e^{-4t} (1 + \cos 40\pi t) dt \\ &\simeq 24 \int_0^{0.1} e^{-4t} dt \stackrel{(F.33)}{=} 24 \frac{1}{-4} e^{-4t} \Big|_0^{0.1} = 6(1 - e^{-0.4}) \simeq 1.98 \text{ J} \end{aligned} \quad (\text{E4.4.7})$$

- (d) All of the above computations for analysis can be done by running the following MATLAB program `cir04e04.m`. In Figure 4.5.2 it plots the approximate node voltage (E4.4.6) together with the exact one obtained by using `ilaplace()`, which is the MATLAB function for the inverse Laplace transform.

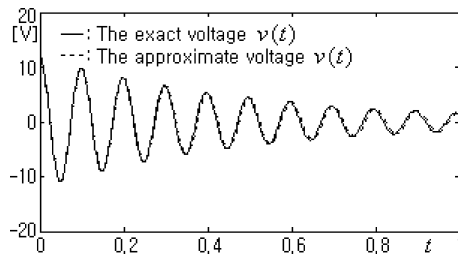


Figure 4.5.2 The output voltage of the circuit depicted in Figure 4.5.1

```
%cir04e04.m
clear, clf
syms s
Vi=12; R=3; L=0.003; C=1/12; vC0= Vi;
Vs= C*vC0/(1/R+1/s/L+s*C); % Eq. (E4.4.5)
v= ilaplace(Vs) % the inverse Laplace transform
t0=0; tf=2; N=500; tt= t0+[0:N]/N*(tf-t0);
for n=1:length(tt)
    t= tt(n); vt(n)= eval(v);
end
sigma= 1/2/R/C; wd= sqrt(1/L/C-sigma^2);
K1=Vi; K2= -Vi*sigma/wd; vt1= K1*exp(-2*tt).*cos(20*pi*tt); % Eq. (E4.4.6)
plot(tt,vt, tt,vt1,'')
Power_of_R= inline('48*exp(-4*t).*cos(20*pi*t).^2','t');
Energy_dissipated_in_R= quad(Power_of_R,0,0.1) % Eq. (E4.4.7)
```

4.2.3 Two-Mesh/Node Circuit

Once a given circuit with its initial conditions is transformed into its s -domain equivalent, it can be dealt with it as if it consisted of sources and resistors only, where the passive elements have impedances (R , sL , or $1/(sC)$) that can be thought of as generalized resistances. The number of inductors/capacitors or meshes/nodes makes no essential difference. The same criterion is used for determining which one of mesh analysis and node analysis has a computational advantage (see Section 2.6):

1. Which is fewer, the number of mesh equations, $(b - n + 1)$, or that of node equations, $(n - 1)$? Note that b is the number of branches having an element (between the two nodes) and n is the number of nodes in a circuit with every source removed (see Section 1.4.4).
2. Which is easier, converting all the sources into voltage sources or current sources? Note that it is easy to set up the mesh equations for circuits having no current sources and the node equations for circuits having no voltage sources. In this context, we had better choose the analysis method before transforming the initial conditions into their s -domain equivalent sources and then transform them into voltage or current sources depending on the analysis method.
3. Which do you want to find, current or voltage?

(Example 4.5) A Two-Mesh/Node Circuit

Consider the circuit of Figure 4.6(a) in which the values of the source voltage, the resistors, the inductor, and the capacitor are

$$V_i = 1 \text{ V}, \quad R_1 = \frac{1}{2} \Omega, \quad R_2 = \frac{1}{2} \Omega, \quad L = \frac{1}{4} \text{ H}, \quad \text{and} \quad C = 1 \text{ F} \quad (\text{E4.5.1})$$

respectively. Suppose the switch has been connected to the DC voltage source V_i for a long time before $t = 0$ when it is flipped to the ground. Since the circuit is supposed to be in the DC steady state where the inductor L is like shorted and the capacitor C is like opened, the (initial) values of the inductor current and the capacitor voltage at $t = 0$ are found to be

$$I_0 = i_L(0) = V_i/R_1 = 1/(1/2) = 2 \text{ A} \quad \text{and} \quad V_0 = v_C(0) = V_i = 1 \text{ V} \quad (\text{E4.5.2})$$

Figures 4.6(b) and (c) show the s -domain equivalents with the initial conditions represented by voltage and current sources that suit the mesh/node analysis, respectively.

(a) Mesh Analysis

The formula (2.12) for the s -domain equivalent in Figure 4.6(b) can be used to write the mesh equation as

$$\begin{bmatrix} s/4 + 1/2 & -1/2 \\ -1/2 & 1/2 + 1/2 + 1/s \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} 1/2 \\ -1/s \end{bmatrix} \quad (\text{E4.5.3})$$

which yields

$$\begin{bmatrix} s + 2 & -2 \\ -s & 2s + 2 \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} 2 \\ -2 \end{bmatrix}$$

$$\begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \frac{1}{2(s^2 + 2s + 2)} \begin{bmatrix} 2(s+1) & 2 \\ s & s+2 \end{bmatrix} \begin{bmatrix} 2 \\ -2 \end{bmatrix} = \frac{2}{(s+1)^2 + 1^2} \begin{bmatrix} (s+1) - 1 \\ -1 \end{bmatrix} \quad (\text{E4.5.4})$$

$$i_1(t) = \mathcal{L}^{-1}\{I_1(s)\} \stackrel{\text{Table A.1(9),(10)}}{=} 2e^{-t}(\cos t - \sin t) u_s(t) [\text{A}] \quad (\text{E4.5.5a})$$

$$i_2(t) = \mathcal{L}^{-1}\{I_2(s)\} \stackrel{\text{Table A.1(9)}}{=} -2e^{-t} \sin t u_s(t) [\text{A}] \quad (\text{E4.5.5b})$$

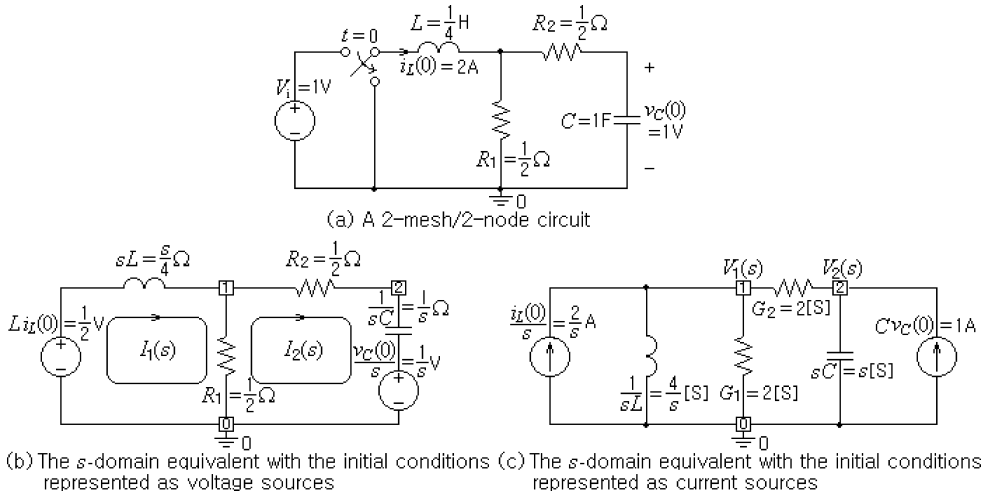


Figure 4.6 The circuit for Example 4.5

The s -domain V - I relationship (3.16b) can also be used to get the capacitor voltage as

$$V_C(s) \stackrel{(3.16b)}{=} \frac{1}{sC} I_2(s) + \frac{v_C(0)}{s} \stackrel{(E4.5.2),(E4.5.4)}{=} \frac{-2}{s(s^2 + 2s + 2)} + \frac{1}{s} = \frac{(s+1) + 1}{(s+1)^2 + 1^2};$$

$$v_C(t) = e^{-t}(\cos t + \sin t) u_s(t) \text{ [V]} \quad (E4.5.6)$$

(b) Node Analysis

The formula (2.10) for the s -domain equivalent in Figure 4.6(c) can be used to write the node equation as

$$\begin{bmatrix} 4/s + 2 + 2 & -2 \\ -2 & 2 + s \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} = \begin{bmatrix} 2/s \\ 1 \end{bmatrix} \quad (E4.5.7)$$

which yields

$$\begin{bmatrix} 4(s+1) & -2s \\ -2 & s+2 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} = \begin{bmatrix} 2 \\ 1 \end{bmatrix}$$

$$\begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} = \frac{1}{4(s^2 + 2s + 2)} \begin{bmatrix} s+2 & 2s \\ 2 & 4(s+1) \end{bmatrix} \begin{bmatrix} 2 \\ 1 \end{bmatrix} = \frac{1}{(s+1)^2 + 1^2} \begin{bmatrix} s+1 \\ (s+1) + 1 \end{bmatrix} \quad (E4.5.8)$$

$$v_1(t) = e^{-t} \cos t u_s(t) \text{ [V]} \quad (E4.5.9a)$$

$$v_2(t) = e^{-t}(\cos t + \sin t) u_s(t) \text{ [V]} \quad (E4.5.9b)$$

The s -domain V - I relationship (3.15b) is also used to obtain the inductor current as

$$I_L(s) \stackrel{(3.15b)}{=} \frac{1}{sL} [-V_1(s)] + \frac{i_L(0)}{s} \stackrel{(E4.5.2),(E4.5.8)}{=} -\frac{4(s+1)}{s(s^2 + 2s + 2)} + \frac{2}{s} = \frac{2(s+1) - 2 \times 1}{(s+1)^2 + 1^2}$$

$$i_L(t) = 2e^{-t}(\cos t - \sin t) u_s(t) \text{ [A]} \quad (E4.5.10)$$

4.2.4 Circuits Having Dependent Sources

The following example illustrates that transformed (s -domain) equivalents are good for analyzing circuits regardless of the existence of dependent sources in the circuits.

(Example 4.6) A Circuit with a Dependent Source

Let us apply the mesh analysis and the node analysis to the circuit of Figure 4.7(a) to find the expression of the transformed output voltage $V_o(s)$ in terms of the transformed input source voltage $V_i(s)$.

(a) Mesh Analysis

First, the controlling variable $V_2(s)$ is expressed in terms of $I_1(s)$ and $I_2(s)$ as

$$V_2(s) \stackrel{(3.16b)}{=} \frac{1}{sC_2} I_{C_2}(s) + \frac{v_{C_2}(0)}{s} = \frac{1}{sC_2} [I_1(s) - I_2(s)] + \frac{v_{C_2}(0)}{s} \quad (E4.6.1)$$

Then the circuit is transformed into the s -domain equivalent with the initial conditions represented by voltage sources as depicted in Figure 4.7(b), and the mesh equation is written as

$$\begin{bmatrix} R_1 + R_2 + 1/(sC_2) & -[R_2 + 1/(sC_2)] \\ -[R_2 + 1/(sC_2)] & 1/(sC_1) + R_2 + 1/(sC_2) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_i(s) - v_{C_2}(0)/s \\ v_{C_2}(0)/s - v_{C_1}(0)/s - KV_2(s) \end{bmatrix} \quad (E4.6.2)$$

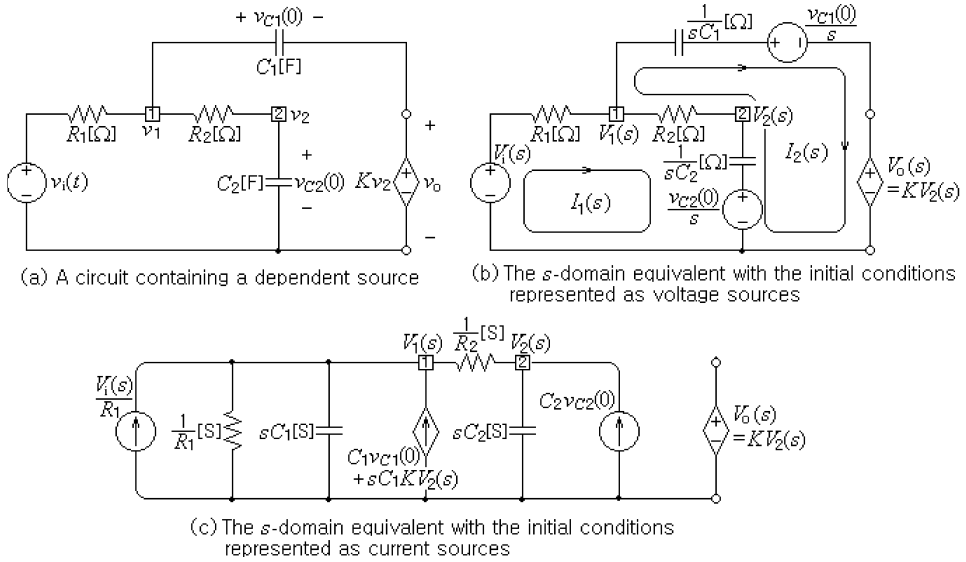


Figure 4.7 The circuit for Example 4.6

Equation (E4.6.1) is substituted for $V_2(s)$ into the right-hand side (RHS) of Equation (E4.6.2), and the unknown terms on the RHS are moved to the LHS to write

$$\begin{bmatrix} R_1 + R_2 + 1/(sC_2) & -[R_2 + 1/(sC_2)] \\ -[R_2 + (1-K)/(sC_2)] & 1/(sC_1) + R_2 + (1-K)/(sC_2) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_1(s) - v_{C2}(0)/s \\ (1-K)v_{C2}(0)/s - v_{C1}(0)/s \end{bmatrix} \quad (\text{E4.6.3})$$

$$\begin{bmatrix} s(R_1 + R_2)C_2 + 1 & -(sR_2C_2 + 1) \\ -[sR_2C_1C_2 + (1-K)C_1] & C_2 + sR_2C_1C_2 + (1-K)C_1 \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} sC_2V_1(s) - C_2v_{C2}(0) \\ (1-K)C_1C_2v_{C2}(0) - C_1C_2v_{C1}(0) \end{bmatrix}$$

With the assumption of zero initial conditions $v_{C1}(0) = 0$ and $v_{C2}(0) = 0$ for simplicity, this equation is solved to obtain the mesh currents as

$$\begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \frac{sC_2V_1(s)}{s^2R_1R_2C_1C_2^2 + s[(R_1 + R_2)C_2^2 + (1-K)R_1C_1C_2] + C_2} \begin{bmatrix} C_2 + sR_2C_1C_2 + (1-K)C_1 \\ sR_2C_1C_2 + (1-K)C_1 \end{bmatrix} \quad (\text{E4.6.4})$$

Thus the output voltage is

$$\begin{aligned} V_o(s) &= KV_2(s) \stackrel{(\text{E4.6.1})}{=} \frac{K}{sC_2} [I_1(s) - I_2(s)] \\ &\stackrel{(\text{E4.6.4})}{=} \frac{K}{s^2R_1R_2C_1C_2 + s[(R_1 + R_2)C_2 + (1-K)R_1C_1] + 1} V_1(s) \end{aligned} \quad (\text{E4.6.5})$$

(b) Node Analysis

To apply the node analysis, the circuit is transformed into the s -domain equivalent with the initial conditions represented by current sources and the independent/dependent sources converted into current sources,

as depicted in Fig. 4.7(c), and then the node equation is written as

$$\begin{bmatrix} 1/R_1 + sC_1 + 1/R_2 & -1/R_2 \\ -1/R_2 & 1/R_2 + sC_2 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} = \begin{bmatrix} V_i(s)/R_1 + C_1 v_{C_1}(0) + sC_1 K V_2(s) \\ C_2 v_{C_2}(0) \end{bmatrix} \quad (\text{E4.6.6})$$

To solve this equation for $V_1(s)$ and $V_2(s)$, we move the unknown term $sC_1 K V_2(s)$ on the RHS to the left-hand side (LHS) and rearrange the equation as

$$\begin{bmatrix} R_2 + sR_1 R_2 C_1 + R_1 & -R_1 - sK R_1 R_2 C_1 \\ -1 & 1 + sR_2 C_2 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} = \begin{bmatrix} R_2 V_i(s) + R_1 R_2 C_1 v_{C_1}(0) \\ R_2 C_2 v_{C_2}(0) \end{bmatrix} \quad (\text{E4.6.7})$$

With the assumption of zero initial conditions $v_{C_1}(0) = 0$ and $v_{C_2}(0) = 0$ for simplicity, this equation is solved for the node voltages as

$$\begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} = \frac{R_2 V_i(s)}{s^2 R_1 R_2^2 C_1 C_2 + s R_2 [(R_1 + R_2) C_2 + (1 - K) R_1 C_1] + R_2} \begin{bmatrix} 1 + s R_2 C_2 \\ 1 \end{bmatrix} \quad (\text{E4.6.8})$$

Thus the output voltage is

$$V_o(s) = K V_2(s) \stackrel{(\text{E4.6.8})}{=} \frac{K}{s^2 R_1 R_2 C_1 C_2 + s[(R_1 + R_2) C_2 + (1 - K) R_1 C_1] + 1} V_i(s) \quad (\text{E4.6.9})$$

(c) Mesh/Node Analysis Using MATLAB

All the above computations can be performed by running the following MATLAB program `cir04e06.m`.

```
%cir04e06.m
clear, clf
syms s R1 R2 C1 C2 K Vis
sC1 = s*C1; sC2 = s*C2;
display('(a)')
Z = [R1+R2+1/sC2 - (R2+1/sC2); - (R2+(1-K)/sC2) 1/sC1+R2+(1-K)/sC2];
Is = Z\[Vis; 0]; % Eq. (E4.6.3) -> (E4.6.4)
Vos = K/sC2*(Is(1)-Is(2)); % Eq. (E4.6.5)
pretty(simplify(Vos))
display('(b)')
Y = [1/R1+sC1+1/R2 -1/R2-sC1*K; -1/R2 1/R2+sC2];
Vs = Y\[Vis/R1; 0]; % Eq. (E4.6.6) -> (E4.6.8)
Vos = K*Vs(2); % Eq. (E4.6.9)
pretty(simplify(Vos))
```

```
>> cir04e06
```

```

              K Vis
-----
              2
R2 s C2 + C2 R2 s  C1 R1 - C1 K R1 s + 1 + R1 s C2 + C1 R1 s
```

4.2.5 Thevenin Equivalent Circuit

The following example illustrates the fact that transformed (s -domain) equivalents are also effective for finding Thevenin equivalents.

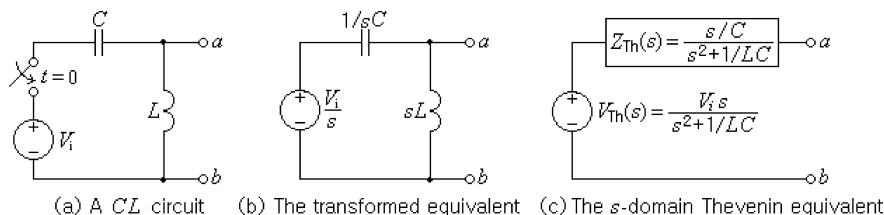


Figure 4.8 The circuit for Example 4.7

(Example 4.7) s -Domain Thevenin Equivalent

Find the Thevenin equivalent seen from the terminals a and b of the circuit in Figure 4.8(a), where the voltage source is applied starting from $t = 0$ when the switch is closed, so that its value can be described as

$$v_i(t) = V_i u_s(t) \xrightarrow{\text{Table A.1(3)}} V_i(s) = \frac{V_i}{s} \quad (\text{E4.7.1})$$

As explained in Section 2.7, the Thevenin voltage source can be found as the voltage with the terminals a and b open, i.e. with no load connected across the two terminals. Referring to the transformed equivalent in Figure 4.8(b), the voltage divider rule (Section 2.2.1) is used to obtain the Thevenin voltage source as

$$V_{\text{Th}}(s) = \frac{V_i}{s} \frac{sL}{sL + 1/(sC)} = \frac{V_i s}{s^2 + 1/(LC)} \quad (\text{E4.7.2})$$

The Thevenin (equivalent) impedance seen from the terminals a and b can be found by removing (short-circuiting) the independent voltage source. It turns out to be the parallel combination of L and C as

$$Z_{\text{Th}}(s) = \frac{sL \times 1/(sC)}{sL + 1/(sC)} = \frac{s/C}{s^2 + 1/(LC)} \quad (\text{E4.7.3})$$

The Thevenin equivalent is depicted in Figure 4.8(c).

Note. The inverse Laplace transform of the s -domain expression (E4.7.2) can be taken to find the time-domain expression of the Thevenin voltage source as

$$v_{\text{Th}}(t) = \mathcal{L}^{-1}\{V_{\text{Th}}(s)\} \xrightarrow[\text{Table A.1(8)}]{(\text{E4.7.2})} V_i \cos(\sqrt{1/(LC)}) u_s(t) \quad (\text{E4.7.4})$$

which is a sinusoidal voltage. Does it imply that an AC voltage can be generated from a DC voltage source? It seems to be possible if only the (input) impedance of the load to be connected at the terminals a and b is infinity. However, since a real-world inductor/capacitor cannot be free from some parasitic/leakage resistance, the output voltage of this CL circuit is expected to be a sinusoidal voltage with exponentially decreasing amplitude, even for a load of infinitely large impedance.

4.3 Second-Order OP AMP Circuits

(Example 4.8) Second-Order OP Amp Circuits

(a) Find the transformed output voltage $V_o(s)$ of the circuit in Figure 4.9(a) with zero initial conditions. KCL is applied to nodes 1 and 2 to write the node equations:

$$\text{Node 1: } \frac{V_1(s) - V_i(s)}{R_1} + \frac{V_1(s) - V_2(s)}{R_2} + sC_1[V_1(s) - V_o(s)] = 0 \quad (\text{E4.8.1a})$$

$$\text{Node 2: } \frac{V_2(s) - V_1(s)}{R_2} + sC_2 V_2(s) = 0 \quad (\text{E4.8.1b})$$

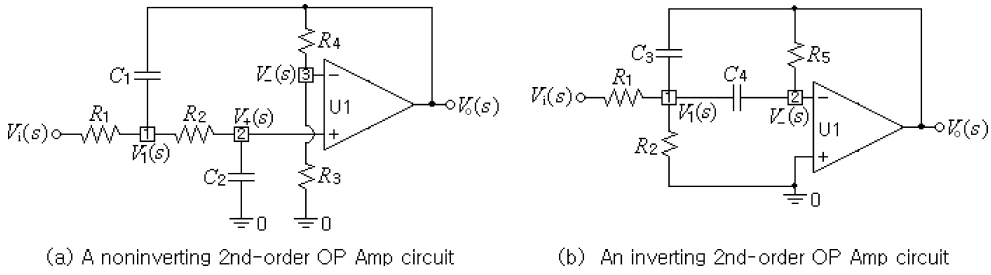


Figure 4.9 Second-order active filters

Regarding the pair of two resistors R_3 and R_4 in series as a voltage divider and applying the virtual short principle (Remark 1.2(2)) for the OP Amp with negative feedback, the voltage at node 2, which is the positive input terminal of the OP Amp, can be written as

$$V_2(s) = V_+(s) \stackrel{\text{virtual short}}{=} V_-(s) \stackrel{\text{voltage divider}}{=} \frac{R_3}{R_3 + R_4} V_o(s)$$

This implies

$$V_o(s) = KV_2(s) \quad \text{with} \quad K = \frac{R_3 + R_4}{R_3} \quad (\text{E4.8.2})$$

Substituting this into Equation (E4.8.1), the node equations can be rewritten in matrix–vector form as

$$\begin{bmatrix} 1/R_1 + sC_1 + 1/R_2 & -1/R_2 - sC_1 K \\ -1/R_2 & 1/R_2 + sC_2 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} = \begin{bmatrix} V_i(s)/R_1 \\ 0 \end{bmatrix} \quad (\text{E4.8.3})$$

and solved for $V(s)$ to obtain the expression of $V_o(s) = KV_2(s)$ in terms of $V_i(s)$ as

$$V_o(s) = KV_2(s) = \frac{K}{s^2 R_1 R_2 C_1 C_2 + s[(R_1 + R_2)C_2 + (1 - K)R_1 C_1] + 1} V_i(s) \quad (\text{E4.8.4})$$

(b) Find the transformed output voltage $V_o(s)$ of the circuit in Figure 4.9(b) with zero initial conditions. KCL is applied to nodes 1 and 2 to write the node equations:

$$\text{Node 1 :} \quad \frac{V_1(s) - V_i(s)}{R_1} + \frac{V_1(s)}{R_2} + sC_3[V_1(s) - V_o(s)] + sC_4 V_1(s) = 0 \quad (\text{E4.8.5a})$$

$$\text{Node 2 :} \quad sC_4[0 - V_1(s)] + \frac{0 - V_o(s)}{R_5} = 0 \quad (\text{E4.8.5b})$$

These node equations can be written in matrix–vector form as

$$\begin{bmatrix} 1/R_1 + 1/R_2 + sC_3 + sC_4 & -sC_3 \\ sC_4 & 1/R_5 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_o(s) \end{bmatrix} = \begin{bmatrix} V_i(s)/R_1 \\ 0 \end{bmatrix} \quad (\text{E4.8.6})$$

$$\begin{bmatrix} R_1 + R_2 + sR_1 R_2 (C_3 + C_4) & -sR_1 R_2 C_3 \\ sR_5 C_4 & 1 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_o(s) \end{bmatrix} = \begin{bmatrix} R_2 V_i(s) \\ 0 \end{bmatrix}$$

and solved to obtain the expression of $V_o(s)$ in terms of $V_i(s)$ as

$$V_o(s) = \frac{-s R_2 R_5 C_4}{s^2 R_1 R_2 R_5 C_3 C_4 + s R_1 R_2 (C_3 + C_4) + R_1 + R_2} V_i(s) \quad (\text{E4.8.7})$$

4.4 Analogy and Duality

4.4.1 Analogy

In Section 4.1 the *transfer* or *system function* is defined as the ratio of the transformed output $Y(s)$ to the transformed input $X(s)$ (with zero initial conditions)

$$G(s) = \left. \frac{Y(s)}{X(s)} \right|_{\text{with zero initial conditions}} \quad (4.22)$$

Even if this concept is defined for a differential equation like Equation (4.1) with the input variable $x(t)$ and the output variable $y(t)$, it is the transformed input–output relationship of a system whose time-domain input–output relationship is described by the differential equation. A question arises. Why is the assumption of zero initial conditions needed for a definition of the transfer function? It is because the transfer function describing the characteristics of a system should be defined so that it does not vary with the initial conditions.

In fact, a differential equation having an input $x(t)$ and an output $y(t)$ can be thought of as an abstract system. However, what is meant by a system is often a physical system such as an electrical system (circuit), a mechanical system, etc. For example, a series *RLC* circuit is described by the time-domain input–output relationship based on Kirchhoff's laws as

$$L \frac{di(t)}{dt} + R i(t) + \frac{1}{C} \int_{-\infty}^t i(\tau) d\tau = v(t)$$

which, in view of the definition of the current $i(t) = dq/dt$, can be rewritten as

$$L \frac{d^2 q(t)}{dt^2} + R \frac{dq(t)}{dt} + \frac{1}{C} q(t) = v(t) \quad (4.23)$$

where L = inductance, R = resistance, C = capacitance, q = electric charge, and v = voltage. Likewise, a mass–dashpot–spring system is described by the time-domain input–output relationship based on Newton's law as

$$M \frac{d^2 y(t)}{dt^2} + B \frac{dy(t)}{dt} + K y(t) = f(t) \quad (4.24)$$

where M = mass, B = damping coefficient, K = spring constant, y = displacement, and f = force.

On the assumption of zero initial conditions, the Laplace transform of the differential equations is taken and then the transfer functions are found as

$$Ls^2 Q(s) + RsQ(s) + \frac{1}{C} Q(s) = V(s) \rightarrow \frac{Q(s)}{V(s)} = \frac{1}{Ls^2 + Rs + 1/C} \quad (4.25a)$$

$$Ms^2 Y(s) + BsY(s) + KY(s) = F(s) \rightarrow \frac{Y(s)}{F(s)} = \frac{1}{Ms^2 + Bs + K} \quad (4.25b)$$

These two systems are said to be *analogous* in the sense that their input–output relationships are described by the differential equations and transfer functions that are mathematically identical, though the physical meanings of their input, output variables, and the coefficients are different.

4.4.2 Duality

While the analogy introduced in the previous section is for systems that are governed by different physical laws, duality is for systems that are governed by the same physical laws. For example, the two circuits in Figures 4.10(a) and (b) are *dual* to each other in the sense that the mesh equation for one circuit is identical to the node equation for the other circuit if every variable such as voltage/current and every parameter such as resistance/conductance and inductance/capacitance are switched to the corresponding variable/parameter listed in Table 4.1.

Note that the mesh equation for the circuit in Figure 4.10(a) and the node equation for the circuit in Figure 4.10(b) are

$$\begin{bmatrix} R_1 + 1/(sC) & -1/(sC) \\ -1/(sC) & R_2 + sL + 1/(sC) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_i(s) - v_C(0)/s \\ v_C(0)/s + Li_L(0) \end{bmatrix} \quad (4.26a)$$

and

$$\begin{bmatrix} G_1 + 1/(sL) & -1/(sL) \\ -1/(sL) & G_2 + sC + 1/(sL) \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} = \begin{bmatrix} I_i(s) - i_L(0)/s \\ i_L(0)/s + Cv_C(0) \end{bmatrix} \quad (4.26b)$$

respectively. They can be obtained from each other by the following exchange:

$$\begin{aligned} V_i[V] &\leftrightarrow I_i[A], & v_C[V] &\leftrightarrow i_L[A], & I_1[A] &\leftrightarrow V_1[V], & I_2[A] &\leftrightarrow V_2[V] \\ L[H] &\leftrightarrow C[F], & R_1[\Omega] &\leftrightarrow G_1[S], & R_2[\Omega] &\leftrightarrow G_2[S] \end{aligned}$$

To construct the dual circuit for a given (primal or original) circuit, the following steps are taken:

1. Assign the mesh current in the same (clockwise) direction for every mesh.
2. Place a node inside every mesh and one additional (reference) node outside the primal circuit.
3. Connect the nodes for neighboring meshes by lines through every element shared by two meshes. Connect each node for an outer mesh to the outside (reference) node through every element that is hanging on an outer branch, not shared with another mesh.

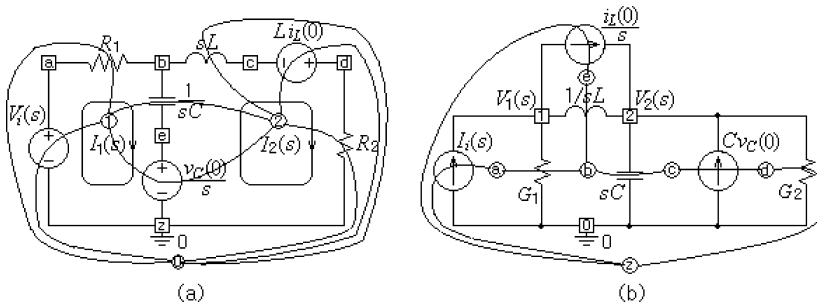


Figure 4.10 Construction of dual circuits

Table 4.1 Variables and parameters dual to each other

Voltage	$v[\text{V}]$	$\leftrightarrow$	$i[\text{A}]$	Current
Resistance	$R[\Omega]$	$\leftrightarrow$	$G[\text{S}]$	Conductance
Inductance	$L[\text{H}]$	$\leftrightarrow$	$C[\text{F}]$	Capacitance
Mesh		$\leftrightarrow$		Node
Series		$\leftrightarrow$		Parallel
Open-circuit		$\leftrightarrow$		Short-circuit

4. Attach the corresponding dual element to the line (branch) drawn at Step 3. If the element in the primal circuit is a capacitor of $C = 10\text{F}$ shared by meshes 1 and 2, the corresponding dual element should be an inductor of $L = 10\text{H}$ connected between nodes 1 and 2 in the dual circuit. If the element in the primal circuit is a resistor of $R_1 = 2\Omega$ on the outside branch of mesh 1, the corresponding dual element should be another resistor of conductance $G_1 = 2\text{S}$ or resistance $R_1 = 1/2\Omega$, which is connected between node 1 and the outside (reference) node in the dual circuit. If the element in the primal circuit is a voltage source of, say, 5 V and with the polarity to increase/decrease the mesh current, the dual element should be a current source of 5 A and with the direction entering/leaving the node corresponding to the mesh. For example, the voltage source $v_C(0)/s$ shared by the two meshes 1 and 2 of the circuit in Figure 4.10(a) has the polarity to decrease the mesh current I_1 and increase I_2 , while the current source $i_L(0)/s$ connected between the two nodes 1 and 2 of the dual circuit in Figure 4.10(b) has the direction of leaving node 1 (to decrease the node voltage V_1) and entering node 2 (to increase V_2).

4.5 Transfer Function, Impulse Response, and Convolution

In Sections 4.1 and 4.4, we take the Laplace transform of the differential equation describing a system on the assumption of zero initial conditions to obtain the ratio of the transformed output to the transformed input as the transfer function. It is, however, possible only for differential equations with the following two features:

1. They are composed of only terms that are proportional to the input, the output, or their derivatives (linearity).
2. All the coefficients are constants not varying with time (time-invariance).

The systems described by such a linear differential equation with constant coefficients are said to be *linear time-invariant* (LTI) systems.

To establish the concept of a transfer function from another point of view, both sides of Equation (4.22), the definition of the transfer function, are multiplied by $X(s)$ to write

$$Y(s) \stackrel{(4.22)}{=} G(s)X(s) \quad (4.27)$$

This transformed input–output relationship will be used to obtain the *impulse response*, i.e. the output of a system having the transfer function $G(s)$, to a unit impulse input $x(t) = \delta(t)$ with the Laplace transform $X(s) = \mathcal{L}\{\delta(t)\} = 1$:

$$Y(s) \stackrel{(4.27)}{=} G(s)X(s) \stackrel{X(s)=1}{=} G(s) \rightarrow \mathcal{L}^{-1}\{G(s)\} = g(t); \quad G(s) = \mathcal{L}\{g(t)\} \quad (4.28)$$

This implies that the *transfer function* of a system can be interpreted as the Laplace transform of the impulse response $g(t)$, which can be regarded as another definition of the transfer function.

Now a question may arise: How is the output $y(t)$ of an LTI system related to a general input $x(t)$ with its impulse response $g(t)$? Is it $y(t) = g(t)x(t)$? No! It is not a multiplication but a *convolution*, as it can be obtained from the inverse Laplace transform of Equation (4.27) (see Equation (A.18) in Appendix A):

$$y(t) = g(t) * x(t) = \int_{-\infty}^{\infty} g(t - \tau)x(\tau)d\tau = \int_{-\infty}^{\infty} x(t - \tau)g(\tau)d\tau \quad (4.29)$$

To appreciate this time-domain input–output relationship, the output of an LTI system to an arbitrary input approximated by a linear combination of rectangular pulses will be found in Section 4.5.4.

4.5.1 Linear Systems

A system is said to be *linear* if the superposition principle holds, i.e. its output to a linear combination of several arbitrary inputs is the same as the linear combination of the outputs to individual inputs.

Superposition Principle

Let the output of a system to each individual input $x_i(t)$ be $y_i(t) = G\{x_i(t)\}$. Then the output of the system to a linearly combined input $\sum a_i x_i(t)$ is

$$y(t) = G\left\{\sum a_i x_i(t)\right\} = \sum a_i G\{x_i(t)\} = \sum a_i y_i(t) \quad (4.30)$$

(Ex.) A linear system: $y(t) = 2x(t)$, $y_1(t) + y_2(t) = 2x_1(t) + 2x_2(t) \equiv 2[x_1(t) + x_2(t)]$

(Ex.) A nonlinear system: $y(t) = x(t) + 1$, $y_1(t) + y_2(t) = [x_1(t) + 1] + [x_2(t) + 1] \neq [x_1(t) + x_2(t)] + 1$

4.5.2 Time-Invariant Systems

Let the output of a system to an arbitrary input $x(t)$ be $y(t) = G\{x(t)\}$. The system is said to be *time-invariant* or *shift-invariant* if its output to the delayed/shifted input $x(t - t_1)$ is the delayed version $y(t - t_1)$ of the original output, i.e.

$$y(t - t_1) = G\{x(t - t_1)\} \quad (4.31)$$

(Ex.) A time-invariant system: $y(t) = \sin[x(t)]$

(Ex.) A time-varying system: $y(t) = (\sin t)x(t)$

4.5.3 The Pulse Response of a Linear Time-Invariant System

Consider a linear time-invariant (LTI) system with the impulse response and the transfer function given by

$$g(t) = e^{-at}u_s(t) \quad \text{and} \quad G(s) \stackrel{(4.28)}{=} \mathcal{L}\{g(t)\} = \mathcal{L}\{e^{-at}u_s(t)\} \stackrel{\text{Table A.1(5)}}{=} \frac{1}{s + a}$$

respectively. Let a unity-area rectangular pulse input of duration (pulsewidth) T and height $1/T$

$$x(t) = \frac{1}{T}r_T(t) = \frac{1}{T}[u_s(t) - u_s(t - T)]$$

$$X(s) = \mathcal{L}\{x(t)\} = \frac{1}{T}\mathcal{L}\{u_s(t) - u_s(t - T)\} \stackrel{\text{Tables A.1(3), A.2(2)}}{=} \frac{1}{T}\left(\frac{1}{s} - e^{-Ts}\frac{1}{s}\right)$$

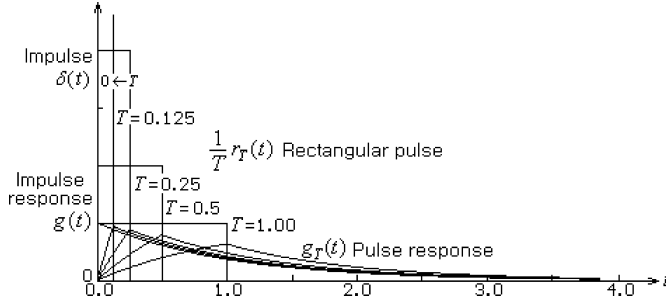


Figure 4.11 The pulse response and the impulse response

be applied to the system. Then the output $g_T(t)$, which is called the *pulse response*, is obtained as

$$Y_T(s) = G(s)X(s) = \frac{1}{T} \left[\frac{1}{s(s+a)} - e^{-Ts} \frac{1}{s(s+a)} \right] = \frac{1}{aT} \left[\frac{1}{s} - \frac{1}{s+a} - e^{-Ts} \left(\frac{1}{s} - \frac{1}{s+a} \right) \right]$$

$$g_T(t) = \mathcal{L}^{-1}\{Y_T(s)\} \stackrel{\text{Tables A.1(3),(5), A.2(2)}}{=} \frac{1}{aT} \left[(1 - e^{-at})u_s(t) - (1 - e^{-a(t-T)})u_s(t-T) \right]$$

If we let $T \rightarrow 0$, i.e. decrease T to an infinitesimal so that the rectangular pulse input becomes an impulse $\delta(t)$ of instantaneous duration and infinite height, how can the output be expressed? Taking the limit of the output equation with $T \rightarrow 0$ yields the impulse response $g(t)$ (see Figure 4.11):

$$g_T(t) \xrightarrow{T \rightarrow 0} \frac{1}{aT} \left[(1 - e^{-at})u_s(t) - (1 - e^{-a(t-T)})u_s(t) \right]$$

$$= \frac{1}{aT} (e^{aT} - 1)e^{-at}u_s(t) \stackrel{\text{(F.25)}}{\underset{aT \rightarrow 0}{\approx}} \frac{1}{aT} (1 + aT - 1)e^{-at}u_s(t) = e^{-at}u_s(t) \equiv g(t) \quad (4.32)$$

This implies that as the input gets close to an impulse, the output becomes close to the impulse response, which is quite natural for any linear time-invariant system.

4.5.4 The Input–Output Relationship of a Linear Time-Invariant System

To find the input–output relationship of a linear time-invariant (LTI) system with the impulse response $g(t)$, an input signal $x(t)$ is approximated as a linear combination of many scaled, time-shifted rectangular pulses and its limit is then taken with $T \rightarrow 0$ (see Figures 4.12(a1) and (a2)):

$$\hat{x}(t) = \sum_{m=-\infty}^{\infty} x(mT) \frac{1}{T} r_T(t-mT)T \quad \text{with } r_T(t-mT) = u_s(t-mT) - u_s(t-mT-T) \quad (4.33)$$

$$\xrightarrow{T \rightarrow d\tau, mT \rightarrow \tau} x(t) = \lim_{T \rightarrow 0} \hat{x}(t) = \int_{-\infty}^{\infty} x(\tau) \delta(t-\tau) d\tau = x(t) * \delta(t) \quad \text{with } \delta(t) = \lim_{T \rightarrow 0} r_T(t)/T \quad (4.34)$$

where the fact was used that the limit of the unity-area rectangular pulse $r_T(t)/T$ with $T \rightarrow 0$ is the unit impulse $\delta(t)$. Now the superposition principle (Equation (4.30)) based on the linearity and time-invariance of the system can be applied to obtain its output $\hat{y}(t)$ to the approximate input $\hat{x}(t)$. Then,

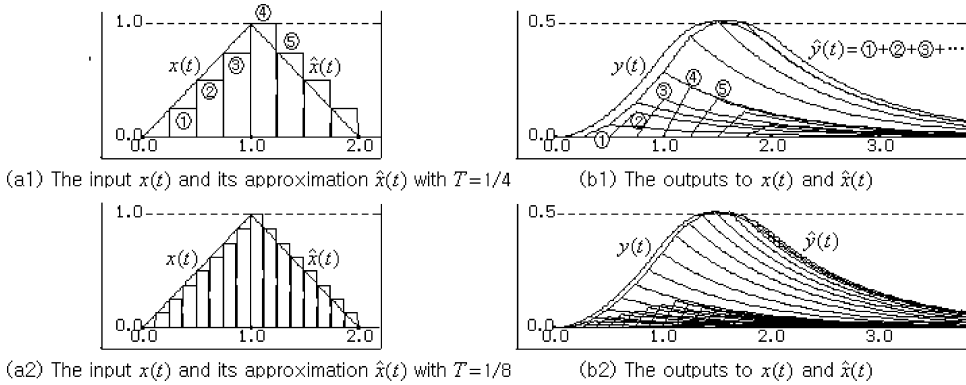


Figure 4.12 The input–output relationship of a linear time-invariant (LTI) system – convolution

noting that the limit of the pulse response $g_T(t)$ with $T \rightarrow 0$ is the impulse response $g(t)$ as illustrated by Equation (4.32), the limit of $\hat{y}(t)$ with $T \rightarrow 0$ is taken to get the output $y(t)$ to the exact input $x(t)$ as

$$\hat{y}(t) = G\{\hat{x}(t)\} = \sum_{m=-\infty}^{\infty} x(mT) g_T(t - mT)T \quad (4.35)$$

$$\xrightarrow{T \rightarrow 0, mT \rightarrow \tau} y(t) = \lim_{T \rightarrow 0} \hat{y}(t) = G\{x(t)\} = \int_{-\infty}^{\infty} x(\tau) g(t - \tau) d\tau = x(t) * g(t) \quad \text{with } g(t) = \lim_{T \rightarrow 0} g_T(t) \quad (4.36)$$

This implies that the output of an LTI system to an input can be expressed as the *convolution* (integral) of the input and the impulse response. Figures 4.12(b1) and (b2) demonstrate the validity of this argument and may enhance understanding of the above equation.

We use the convolution property (A.18) of the Laplace transform to take the Laplace transform of the time-domain input–output relationship (4.36) and find the s -domain input–output relationship as

$$Y(s) = G(s)X(s) \quad (4.37)$$

which agrees with Equation (4.27).

[Remark 4.1] Impulse Response and Transfer (System) Function

The *impulse response* of a system is defined to be the output to a unit impulse input $x(t) = \delta(t)$ and can be expressed as the limit of the pulse response with $T \rightarrow 0$:

$$g(t) = \lim_{T \rightarrow 0} g_T(t) = \lim_{T \rightarrow 0} G\left\{\frac{1}{T}r_T(t)\right\} = G\left\{\lim_{T \rightarrow 0} \frac{1}{T}r_T(t)\right\} = G\{\delta(t)\} \quad (4.38)$$

The *transfer* or *system function* of a linear time-invariant (LTI) system is defined as the ratio of the transformed output to the transformed input and turns out to be the Laplace transform of the impulse response, corresponding to the transformed output to the transformed input $X(s) = 1$:

$$G(s) = \frac{Y(s)}{X(s)} = Y(s)|_{X(s)=1} = \mathcal{L}\{g(t)\} \quad (4.39)$$

4.6 The Steady-State Response to a Sinusoidal Input

The transfer functions of most LTI systems are rational functions of the complex variable s , which are quotients of two polynomials in s as

$$G(s) = \frac{Y(s)}{X(s)} = \frac{Q(s)}{P(s)} = \frac{b_M s^M + b_{M-1} s^{M-1} + \cdots + b_0}{s^N + a_{N-1} s^{N-1} + \cdots + a_0} = \frac{K(s - z_1)(s - z_2) \cdots (s - z_M)}{(s - p_1)(s - p_2) \cdots (s - p_N)} \quad (4.40)$$

where each value of $s = z_m$, $m = 1, 2, \dots, M$, making the numerator $Q(s)$ zero is called a *zero*, and each one of $s = p_n$, $n = 1, 2, \dots, N$, making the denominator $P(s)$ zero is called a *pole* of the transfer function. Note that the poles of the transfer function are the characteristic roots since the characteristic equation is obtained by setting the denominator of the transfer function to zero.

For simplicity, the following assumptions are made about the transfer function of an LTI system:

1. The degree of the numerator polynomial $Q(s)$, M , is less than that of the denominator polynomial $P(s)$, N , i.e. $M < N$.
2. All the poles are in the left-half plane (LHP); i.e. the real parts of all the poles are negative so that the system is stable (see Section 4.1.4).

On these assumptions, let us find the steady state response to a sinusoidal input $x(t)$ such as

$$x(t) = A \cos(\omega t + \phi) = A(\cos \phi \cos \omega t - \sin \phi \sin \omega t) \quad (4.41a)$$

$$X(s) = \mathcal{L}\{x(t)\} \stackrel{\text{Table A.1(7),(8)}}{=} \frac{A(s \cos \phi - \omega \sin \phi)}{s^2 + \omega^2} \quad (4.41b)$$

Substituting this transformed input into the s -domain input–output relationship (4.37), taking the partial fraction expansion, and taking the inverse Laplace transform yields

$$Y(s) \stackrel{(4.37)}{=} G(s)X(s) \stackrel{(4.40),(4.41b)}{=} \frac{Q(s)}{(s - p_1)(s - p_2) \cdots (s - p_N)} \frac{A(s \cos \phi - \omega \sin \phi)}{s^2 + \omega^2}$$

$$\stackrel{\text{partial fraction}}{=} \frac{K_1}{s - p_1} + \frac{K_2}{s - p_2} + \cdots + \frac{K_N}{s - p_N} + \frac{K_0}{s - j\omega} + \frac{K_0^*}{s + j\omega} \quad (4.42)$$

$$y(t) = \mathcal{L}^{-1}\{Y(s)\} \stackrel{\text{Table A.1(5)}}{=} K_1 e^{p_1 t} + K_2 e^{p_2 t} + \cdots + K_N e^{p_N t} + K_0 e^{j\omega t} + K_0^* e^{-j\omega t} \quad (4.43)$$

where

$$K_0 \stackrel{(A.28a)}{=} (s - j\omega)Y(s)|_{s=j\omega} = \left. \frac{G(s)A(s \cos \phi - \omega \sin \phi)}{s + j\omega} \right|_{s=j\omega}$$

$$= \frac{G(j\omega)A(j\omega \cos \phi - \omega \sin \phi)}{j2\omega} = \frac{G(j\omega)A(\cos \phi + j \sin \phi)}{2}$$

$$\stackrel{(F.20)}{=} \frac{1}{2} G(j\omega)A e^{j\phi} \stackrel{(C.4)}{=} \frac{1}{2} |G(j\omega)|A e^{j[\theta(\omega) + \phi]} \quad \text{with} \quad \theta(\omega) = \angle G(j\omega) \quad (4.44)$$

Here, $G(j\omega)$, obtained by substituting $s = j\omega$ (ω = the radian frequency of the input source) into the transfer function, is called the *frequency response*, which will be discussed in detail in Chapter 8. Complying with the assumption of stability that the real parts of all the characteristic roots ($s = p_n$'s) are negative, all the terms (stemming from the characteristic roots) but the last two terms originating from the

sinusoidal input will die out as time goes by. Consequently, the *sinusoidal steady state response* turns out to be

$$\begin{aligned} y_{ss}(t) &= K_0 e^{j\omega t} + K_0^* e^{-j\omega t} = 2\operatorname{Re}\{K_0 e^{j\omega t}\} \stackrel{(4.44)}{=} 2\operatorname{Re}\left\{\frac{1}{2}|G(j\omega)|A e^{j[\omega t + \theta(\omega) + \phi]}\right\} \\ &\stackrel{(F.20)}{=} |G(j\omega)| \cos[\omega t + \theta(\omega) + \phi] \end{aligned} \quad (4.45)$$

where $|G(j\omega)|$ and $\theta(\omega)$ are the magnitude and phase of the *frequency response* $G(j\omega)$. Comparing this steady state response with the sinusoidal input (4.41a), it can be seen that its amplitude is $|G(j\omega)|$ times the amplitude of the input, A , and its phase is $\theta(\omega)$ plus the phase of the input, ϕ , at the source frequency, ω .

The expression for the sinusoidal steady state response can be obtained from the time-domain input–output relationship (4.36); i.e. noting that the sinusoidal input (4.41a) can be written as the sum of two complex conjugate exponential functions

$$x(t) = A \cos(\omega t + \phi) \stackrel{(F.21)}{=} (A/2) (e^{j(\omega t + \phi)} + e^{-j(\omega t + \phi)}) \quad (4.46)$$

$e^{j(\omega t + \phi)}$ is substituted for $x(t)$ into Equation (4.36) to get the partial steady state response as

$$\begin{aligned} y(t) &= G\{x(t)\} \stackrel{(4.36)}{=} \int_{-\infty}^{\infty} x(\tau) g(t - \tau) d\tau = \int_{-\infty}^{\infty} e^{j(\omega \tau + \phi)} g(t - \tau) d\tau \\ &= e^{j(\omega t + \phi)} \int_{-\infty}^{\infty} e^{-j\omega(t - \tau)} g(t - \tau) d\tau = e^{j(\omega t + \phi)} \int_{-\infty}^{\infty} e^{-j\omega t} g(t) dt = e^{j(\omega t + \phi)} G(j\omega) \end{aligned} \quad (4.47)$$

with

$$G(j\omega) = \int_{-\infty}^{\infty} e^{-j\omega t} g(t) dt \stackrel{g(t)=0 \text{ for } t < 0}{\underset{\text{causal system}}{=}} \int_0^{\infty} e^{-j\omega t} g(t) dt = \int_0^{\infty} e^{-s t} g(t) dt \Big|_{s=j\omega} \stackrel{(A.1)}{=} G(s) \Big|_{s=j\omega} \quad (4.48)$$

where we used the definition (A.1) of the Laplace transform relying on another assumption that the system is *causal*, i.e. the impulse response $g(t)$ is zero for all $t < 0$. In fact, all physical systems satisfy the assumption of causality that its output does not precede the input. The total sinusoidal steady state response to the sinusoidal input (4.46) can be expressed as the sum of two complex conjugate terms and finally turns out to be identical with Equation (4.45):

$$\begin{aligned} y_{ss}(t) &= \frac{A}{2} [e^{j(\omega t + \phi)} G(j\omega) + e^{-j(\omega t + \phi)} G(-j\omega)] \\ &= \frac{A}{2} [e^{j(\omega t + \phi)} |G(j\omega)| e^{j\theta} + e^{-j(\omega t + \phi)} |G(-j\omega)| e^{-j\theta}] \\ &= \frac{A}{2} |G(j\omega)| (e^{j(\omega t + \phi + \theta)} + e^{-j(\omega t + \phi + \theta)}) \stackrel{(F.21)}{=} A |G(j\omega)| \cos(\omega t + \phi + \theta) \end{aligned} \quad (4.49)$$

[Remark 4.2] Frequency Response and Stability

For stable systems with all the poles of the transfer function $G(s)$ (i.e. all the characteristic roots) in the left-half plane (LHP), the *frequency response* $G(j\omega)$, obtained by substituting $s = j\omega$ into $G(s)$, determines the steady state response y_{ss} to a sinusoidal input $x(t)$ of radian frequency ω . The amplitude of y_{ss} is the product of $|G(j\omega)|$ and the amplitude of the input sinusoid. The phase of y_{ss} is the sum of $\theta(\omega) = \angle G(j\omega)$ and the phase of the input sinusoid, where ω is the radian frequency of the input. The concept of frequency response is indispensable for the analysis of AC circuits using the phasor method, which will be introduced in Chapter 6.

Note. The transfer function $G(s)$ (Equation (4.39)) and frequency response $G(j\omega)$ (Equation (4.48)) of a system are the Laplace transform and Fourier transform of the impulse response $g(t)$ of the system, respectively.

4.7 An Example of MATLAB Analysis and PSpice Simulation

(Example 4.9) MATLAB Analysis and PSpice Simulation

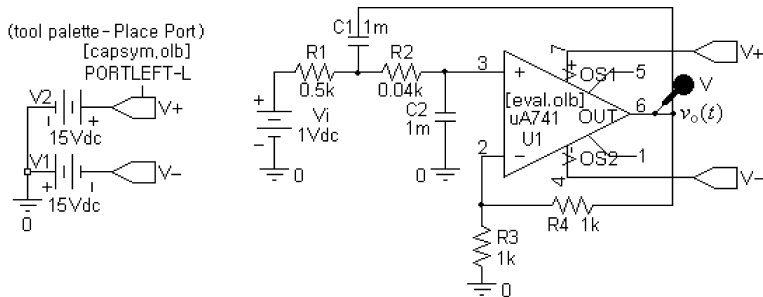
Consider the second-order OP Amp circuit of Figure 4.9(a) in which the values of the parameters are given as follows:

$$V_i = 1 \text{ V} \rightarrow V_i(s) = 1/s, \quad R_1 = 1/2 \text{ k}\Omega, \quad R_2 = 1/25 \text{ k}\Omega, \quad R_3 = R_4 = 1 \text{ k}\Omega, \quad C_1 = C_2 = 1 \text{ mF} \quad (\text{E4.9.1})$$

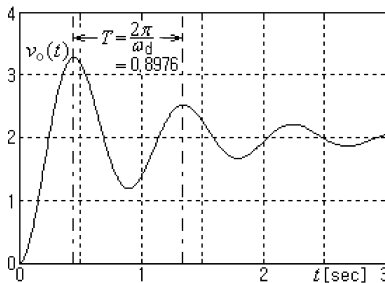
Substituting these parameter values into Equation (E4.8.4) and taking the inverse Laplace transform yields

$$\begin{aligned} V_o(s) &\stackrel{(\text{E4.8.4})}{=} \frac{K = (R_3 + R_4)/R_3}{s^2 R_1 R_2 C_1 C_2 + s[(R_1 + R_2)C_2 + (1 - K)R_1 C_1] + 1} V_i(s) \\ &\stackrel{(\text{E4.9.1})}{=} \frac{K = 2}{s^2 \times (1/2)(1/25) + s(1/2 + 1/25 + (1 - 2)(1/2)) + 1} \frac{1}{s} \\ &= \frac{2 \times 50}{s(s^2 + 2s + 50)} = \frac{2}{s} - \frac{2(s + 1) + (2/7) \times 7}{(s + 1)^2 + 7^2 = (s + \sigma)^2 + \omega_d^2} \end{aligned} \quad (\text{E4.9.2})$$

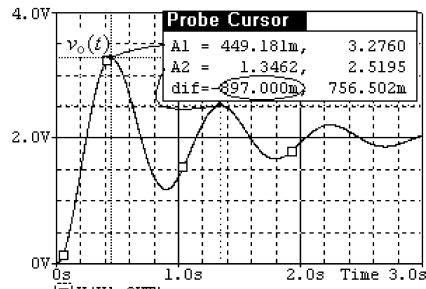
$$v_o(t) = \mathcal{L}^{-1}\{V_o(s)\} \stackrel{\text{Table A.1(3),(9),(10)}}{=} 2 - 2e^{-t} \left(\cos 7t + \frac{1}{7} \sin 7t \right) [\text{V}] \quad (\text{E4.9.3})$$



(a) The PSpice schematic for a second-order active (OP Amp) circuit



(b) The output voltage obtained by using MATLAB



(c) The PSpice simulation result

Figure 4.13 Simulation of the circuit depicted in Figure 4.9(a) (for Example 4.9)

This analytical result indicates that the output voltage has an oscillation of damped frequency $\omega_d = 7 \text{ rad/s}$ and period $2\pi/\omega_d = 0.8976 \text{ s}$, the amplitude decreasing exponentially with the time constant of $1/\sigma = 1 \text{ s}$.

The following MATLAB program `cir04e09.m` is run to get the same result together with the plot of $v_o(t)$ depicted in Figure 4.13(b). Figures 4.13(a) and (c) show the PSpice schematic and the output voltage waveform $v_o(t)$ obtained from the PSpice simulation, respectively.

```
%cir04e09.m
clear, clf
syms s R1 R2 R3 R4 C1 C2 K Vis
sC1 = s*C1; sC2 = s*C2; K = (R3+R4)/R3;
Y = [1/R1+sC1+1/R2 -1/R2-sC1*K; -1/R2 1/R2+sC2];
Vs = Y\[Vis/R1; 0]; % Eq. (E4.8.3)
Vos = K*Vs(2); pretty(Vos) % Eq. (E4.8.4)
% To substitute the numeric values for the parameters
% Vi = 1; R1 = 500; R2 = 40; R3 = 1e3; R4 = 1e3; C1 = 1e-3; C2 = 1e-3; K = (R3+R4)/R3;
Vos = subs(Vos, {Vis, R1, R2, R3, R4, C1, C2}, {1/s, 500, 40, 1e3, 1e3, 1e-3, 1e-3})
% 2000/s / (1000+20*s^2+40*s) = 100/s / (s^2+2*s+50) : Eq. (E4.9.2)
vo = ilaplace(Vos) % 2-2*exp(-t) .* (cos(7*t)+1/7*sin(7*t)) : Eq. (E4.9.3)
t0 = 0; tf = 3; N = 600; tt = t0+(tf-t0)/N*[0:N]; % time vector for [0,3] sec
for n = 1:length(tt)
    t = tt(n); vot(n) = eval(vo);
end
plot(tt, real(vot))
```

```
>> cir04e09
```

$$\frac{(R3 + R4) \text{ Vis}}{R3 + R1 s^2 + C1 R2 R3 C2 + R1 R3 s C2 - R1 s C1 R4 + R3 R2 s C2}$$

```
Vos = 2000/s / (1000 + 20*s^2 + 40*s) % Eq. (E4.9.2)
```

```
vo = 2 - 2*exp(-t)*cos(7*t) - 2/7*exp(-t)*sin(7*t) % Eq. (E4.9.3)
```

Problems

4.1 A Series *RLC* Circuit

Consider the circuit of Figure P4.1

- (a) Let the initial values of the inductor current $i_L(t)$ and the capacitor voltage $v_C(t)$ and the value of the voltage source $v_i(t)$ be

$$i_L(0) = 3.2 \text{ mA}, \quad v_C(0) = 1 \text{ V}, \quad \text{and} \quad v_i(t) = 2u_s(t) [\text{V}] \quad (\text{P4.1.1})$$

respectively. Find the capacitor voltage $v_C(t)$ after $t = 0$ when the switch is closed. Modify and/or complete the following MATLAB program `cir04p01a.m` to find $v_C(t)$ and plot it together with the analytical expression of $v_C(t)$ for $0 \leq t \leq 0.5 \text{ ms}$.

```

%cir04p01a.m
clear, clf
syms s; Vis=2/s; iL0=3.2e-3; vC0=1;
R=100; L=1.5625e-3; C=0.4e-6; sL=s*L; sC=s*C;
t0=0; tf=5e-4; N=500; tt=t0+[0:N]/N*(tf-t0);
Is=(Vis+L*iL0-vC0/s)/(R+sL+1/sC); % Eq. (4.16)
VCs=Is/sC+vC0/s; % Eq. (3.16b)
vC=ilaplace(VCs); pretty(vC)
for n=1:length(tt)
    t=tt(n); vCt(n)=eval(vC);
end
plot(tt,real(vCt))

```

- (b) Let the initial values of the inductor current $i_L(t)$, the capacitor voltage $v_C(t)$, and the value of the voltage source $v_i(t)$ be

$$i_L(0) = 0 \text{ A}, \quad v_C(0) = 0 \text{ V}, \quad \text{and} \quad v_i(t) = 4.8 \cos(40\,000t)u_s(t) [\text{V}] \quad (\text{P4.1.2})$$

respectively. Do the same job as in (a).

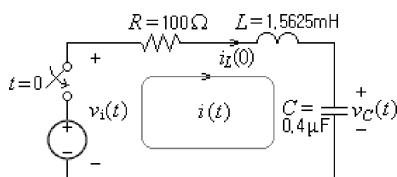


Figure P4.1

4.2 A Parallel RLC Circuit

Consider the circuit of Figure P4.2 in which the switch is closed at $t = 0$ when the initial conditions are

$$i_L(0) = 0.03 \text{ mA} \quad \text{and} \quad v_C(0) = 1 \text{ V} \quad (\text{P4.2.1})$$

- (a) Find the Norton equivalent of the left part (consisting of V_i , R_1 , and R_2) of the circuit seen from terminals a and b .
- (b) Find the top node voltage $v_2(t)$ after $t = 0$ when the switch is closed.

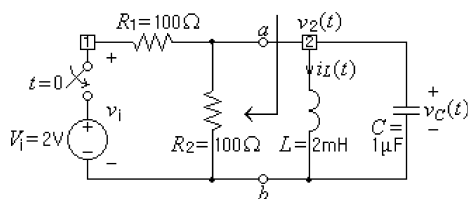


Figure P4.2

4.3 A Second-Order Circuit with Two Meshes/Nodes

Consider the circuit of Figure P4.3 in which the switch has been connected to position *a* for a long time until $t = 0$ when it is flipped to position *b*.

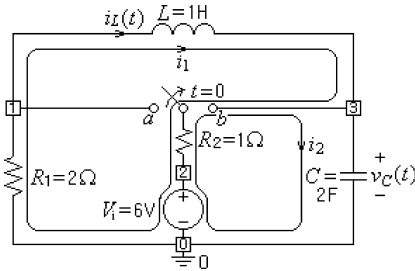


Figure P4.3

(a) Find the initial conditions, i.e. $i_L(0)$ and $v_C(0)$.

(b) Find the capacitor voltage $v_C(t)$ for $t \geq 0$ s.

Note. Readers are encouraged to use MATLAB or its equivalent to obtain the solutions.

4.4 A Second-Order Circuit with Two Meshes/Nodes

Consider the circuit of Figure P4.4 in which the switch has been closed at $t = 0$ when all the initial conditions are zero. Find the capacitor voltage $v_C(t)$ in the following three ways.

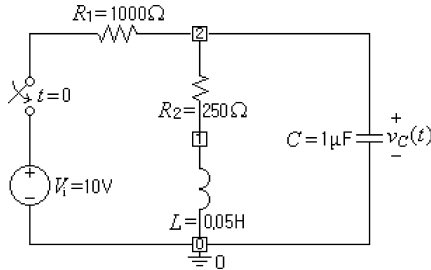


Figure P4.4

(a) Use the voltage divider rule:

$$V_C(s) = \frac{\frac{(R_2 + sL)/(sC)}{(R_2 + sL) + 1/(sC)}}{R_1 + \frac{(R_2 + sL)/(sC)}{(R_2 + sL) + 1/(sC)}} V_i(s) \quad (\text{P4.4.1})$$

(b) Use the mesh analysis.

(c) Use the node analysis.

4.5 A Second-Order Circuit

Consider the circuit of Figure P4.5.

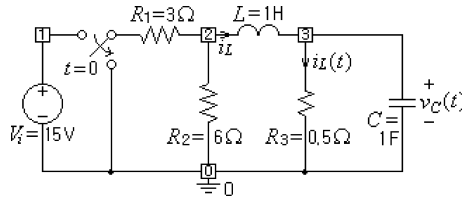


Figure P4.5

- Find $i_L(t)$ and $v_C(t)$ where the switch has been connected to the ground side for a long time until $t = 0$, when it is flipped to the 15 V voltage source side.
- Find $i_L(t)$ and $v_C(t)$ where the switch has been connected to the 15 V voltage source side for a long time until $t = 0$, when it is flipped to the ground side.

4.6 Transfer Function of a Second-Order Circuit with a Dependent Source

Consider the circuit of Figure P4.6 in which all the initial conditions are assumed to be zero. Find the transfer function $V_o(s)/V_i(s)$ in the following two ways.

- Use the mesh analysis.
- Use the node analysis.

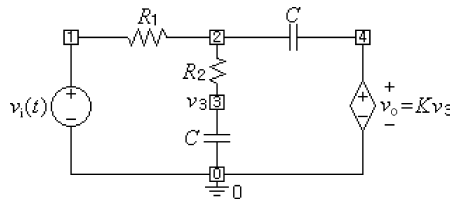


Figure P4.6

4.7 Transfer Function of a Second-Order Circuit with a Dependent Source

Consider the circuit of Figure P4.7 in which all the initial conditions are assumed to be zero. Find the transfer function $V_o(s)/V_i(s)$.

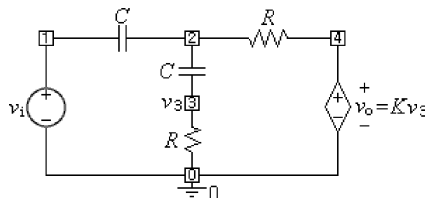


Figure P4.7

4.8 Transfer Function of a Second-Order Circuit with a Dependent Source

Consider the circuit of Figure P4.8 in which all the initial conditions are assumed to be zero. Find the transfer function $V_o(s)/V_i(s)$.

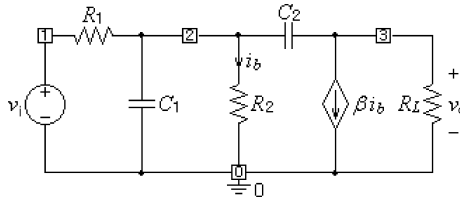


Figure P4.8

4.9 Transfer Function of a Second-Order Circuit with a Dependent Source

Consider the circuit of Figure P4.9 in which all the initial conditions are assumed to be zero. Verify that the transfer function $V_o(s)/V_i(s)$ is

$$\frac{V_o(s)}{V_i(s)} = \frac{KV_3(s)}{V_i(s)} = \frac{sC_1G_1}{(1-K)C_1C_2s^2 + [G_1C_1 + (1-K)G_2(C_1 + C_2)]s + (1-K)G_1G_2} \quad (\text{P4.9.1})$$

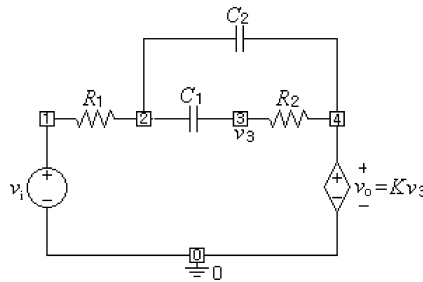


Figure P4.9

4.10 Transient Response in a Second-Order Circuit with Two Dependent Sources

Consider the circuit of Figure P4.10 in which the initial conditions are assumed to be $i_L(0) = 0$ A and $v_C(0) = 8$ V. Find the inductor current $i_L(t)$, the capacitor voltage $v_C(t)$, and the current $i_{R_2}(t)$ through R_2 .

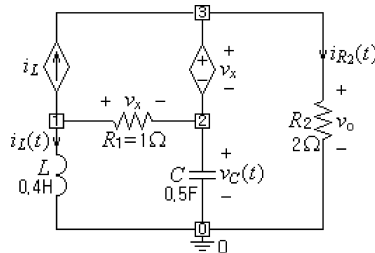
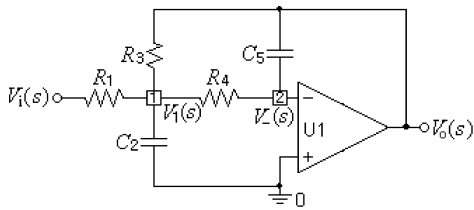


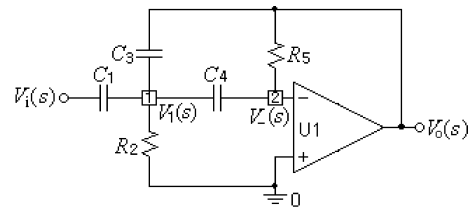
Figure P4.10

4.11 Transfer Function of Second-Order OP Amp Circuits

Consider the OP Amp circuits of Figure P4.11 in which all the initial conditions are assumed to be zero.



(a) A second-order active lowpass filter



(b) A second-order active highpass filter

Figure P4.11

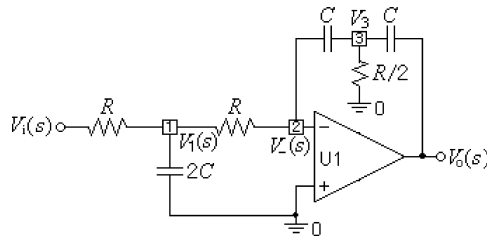
(a) Find the transfer function $V_o(s)/V_i(s)$ of the circuit (a).

(b) Find the transfer function $V_o(s)/V_i(s)$ of the circuit (b).

4.12 A Double Integrator

Consider the OP Amp circuit of Figure P4.12 in which all the initial conditions are assumed to be zero. Apply KCL to nodes 1, 2, and 3 to write a set of node equations in $V_1(s)$, $V_3(s)$, and $V_o(s)$. Then solve it to find the transfer function

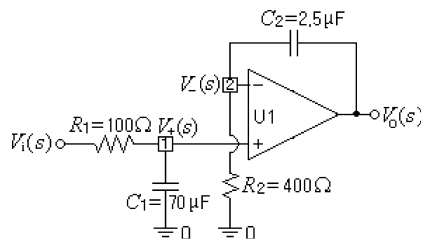
$$G(s) = \frac{V_o(s)}{V_i(s)} = -\frac{1}{(RC)^2 s^2} \quad (\text{P4.12.1})$$

**Figure P4.12**

Note. The circuit in Figure P4.12 saves one OP Amp compared with another (two-stage) double integrator made of two integrators connected in cascade, but instead requires more resistors/capacitors.

4.13 Steady State Response of a Second-Order OP Amp Circuit

Consider the OP Amp circuit of Figure P4.13 in which all the initial conditions are assumed to be zero.

**Figure P4.13**

- (a) Verify that the transfer function of the circuit is

$$G(s) = \frac{V_o(s)}{V_i(s)} = \frac{p(s+z)}{s(s+p)} \quad \text{with} \quad p = \frac{1}{R_1 C_1} \quad \text{and} \quad z = \frac{1}{R_2 C_2} \quad (\text{P4.13.1})$$

- (b) Verify that the steady state output voltage of the circuit to a sinusoidal input
- $v_i(t) = V_{im} \cos \omega t$
- is

$$v_{o,ss}(t) = \frac{\sqrt{1 + [1/(\omega R_2 C_2)]^2}}{\sqrt{1 + (\omega R_1 C_1)^2}} V_{im} \cos(\omega t + \theta) \quad (\text{P4.13.2})$$

$$\text{with } \theta = \tan^{-1}\left(\frac{1}{\omega R_1 C_1}\right) - \tan^{-1}\left(\frac{1}{\omega R_2 C_2}\right) - 90^\circ$$

- (c) With the values of the parameters as

$$R_1 = 100 \Omega, \quad R_2 = 400 \Omega, \quad C_1 = 70 \mu\text{F}, \quad C_2 = 2.5 \mu\text{F}, \quad \text{and} \quad \omega = 1000 \text{ rad/s} \quad (\text{P4.13.3})$$

find the largest amplitude of the input voltage, V_{im} , such that the output voltage is bounded between the dual saturation limit voltages $\pm V_{om} = \pm 11.6\text{V}$ (close to the values of the bipolar supply voltages $V_{CC}/V_{EE} = \pm 12$).

Note. If the output voltage determined by Equation (P4.13.2) exceeds the range upper/lower-bounded by the dual saturation limit voltages $\pm V_{om} = \pm 11.6\text{V}$, the real steady state output voltage will be a clipped sinusoid.

- (d) Noting that
- $V_i(s) = \mathcal{L}\{V_{im} \cos \omega t\} = V_{im}s/(s^2 + \omega^2)$
- , use MATLAB or its equivalent to find the total response

$$v_o(t) = \mathcal{L}^{-1}\{V_o(s)\} = \mathcal{L}^{-1}\{G(s)V_i(s)\} \quad (\text{P4.13.4})$$

to a sinusoidal input $v_i(t) = 58 \cos \omega t$ and plot it together with the steady state response (P4.13.2) for the time interval $[0, 0.05 \text{ s}]$.

- (e) Perform the PSpice simulation to get the response to a sinusoidal input
- $v_i(t) = 58 \cos \omega t$
- .

4.14 A Second-Order OP Amp Circuit Implemented in Hardware

Figure P4.14(a) shows a hardware connection diagram that consists of two chips containing an OP Amp together with two resistors and one capacitor.

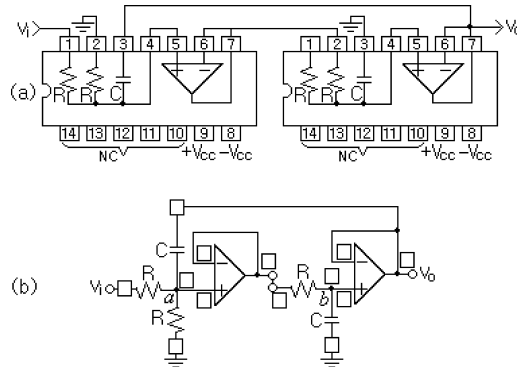


Figure P4.14

- (a) Fill in the square boxes with the corresponding pin numbers in the schematic of Figure P4.14(b).
- (b) On the assumption of zero initial conditions, apply KCL to nodes a and b to write a set of node equations and solve it to find the expression of $V_o(s)$ in terms of $V_i(s)$.

4.15 Wien Bridge Oscillator

Consider the OP Amp circuit of Figure P4.15(a).

- (a) For the OP Amp with a negative feedback path connecting the output terminal to the negative input terminal, the virtual short principle (Remark 1.2(2)) says that the voltages at the positive and negative input terminals are almost equal:

$$v_1 = v_2 = \frac{R_4}{R_3 + R_4} v_o = b v_o \quad (\text{P4.15.1})$$

Let the initial voltages of the capacitors C_1 and C_2 be

$$v_{C_1}(0) = 0 \quad \text{and} \quad v_{C_2}(0) = V_{20} \quad (\text{P4.15.2})$$

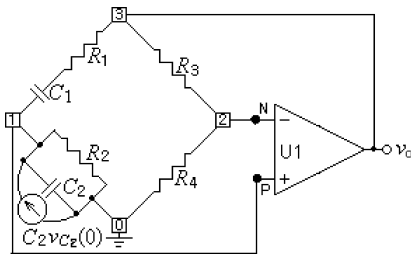
respectively, where the nonzero one of the capacitor C_2 is represented by the current source of $C_2 v_{C_2}(0)$ in parallel with C_2 , as depicted in Figure P4.15(a). Apply KCL to node 1 to write the node equation and solve it to find $V_o(s)$ as

$$\left[\frac{b-1}{R_1 + 1/(sC_1)} + \frac{b}{R_2} + s b C_2 \right] V_o(s) = C_2 V_{20} \quad (\text{P4.15.3})$$

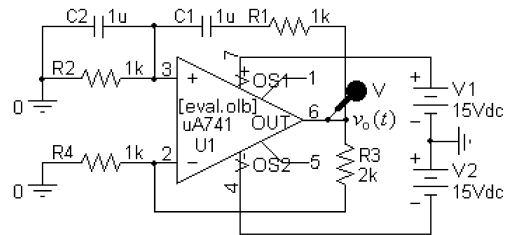
$$V_o(s) = \frac{V_{20} R_2 C_2 (1 + s R_1 C_1)}{b R_1 R_2 C_1 C_2 s^2 + s [b R_1 C_1 + b R_2 C_2 + (b-1) R_2 C_1] + b} \quad (\text{P4.15.4})$$

- (b) With the initial voltage $v_{C_2}(0) = V_{20}$ as a kind of input, Equation (P4.15.4) (excluding V_{20}) can be regarded as a transfer function and its denominator set to zero to obtain the characteristic equation. With reference to Section 4.1.4, verify that the following condition:

$$\frac{R_4}{R_3 + R_4} = b = \frac{R_2 C_1}{R_1 C_1 + R_2 C_2 + R_2 C_1} \quad (\text{P4.15.5})$$



(a) A Wien bridge oscillator



(b) The PSpice schematic

Figure P4.15

guarantees that the characteristic equation has imaginary roots $s = j\omega_r$ so that the output will have an everlasting oscillation of frequency

$$\omega_r = \frac{1}{\sqrt{R_1 R_2 C_1 C_2}} \quad (\text{P4.15.6})$$

(c) Verify that, with

$$v_{C_2}(0) = V_{20} = 1 \text{ V}, \quad R_1 = R_2 = R_4 = 1 \text{ k}\Omega, \quad R_3 = 2 \text{ k}\Omega, \quad \text{and} \quad C_1 = C_2 = 1 \mu\text{F} \quad (\text{P4.15.7})$$

the output voltage is as follows:

$$V_o(s) = \frac{1}{b} \frac{s + \omega_r}{s^2 + \omega_r^2} = \frac{3s}{s^2 + 1000^2} + \frac{3 \times 1000}{s^2 + 1000^2} \quad (\text{P4.15.8})$$

$$v_o(t) = 3(\cos 1000t + \sin 1000t)u_s(t) = 3\sqrt{2} \sin(1000t + 45^\circ)u_s(t) \text{ [V]} \quad (\text{P4.15.9})$$

(d) Use MATLAB or its equivalent to plot $v_o(t)$ for the time interval $[0, 20 \text{ ms}]$.

(e) With reference to the PSpice schematic in Figure P4.15(b), perform the PSpice simulation to get the amplitude and the period of $v_o(t)$.

4.16 Design and Simulation of a Second-Order OP Amp Circuit

Consider the circuit of Figure 4.9(b) in which some values of the parameters are given as

$$V_i = 1.41 \text{ V}, \quad R_1 = R_2, \quad R_5 = 5.1 \text{ k}\Omega, \quad \text{and} \quad C_3 = C_4 \quad (\text{P4.16.1})$$

(a) Choose the values of $C_3 = C_4$ and $R_1 = R_2$ from Table G.3.1 (standard capacitance values) and Table G.2 (5 % tolerance standard resistance values) in Appendix G such that the time constant is close to, but not shorter than, $T = 0.1 \text{ s}$ and the damped frequency is close to, but not higher than, $\omega_d = 2\pi(2/T)$ [rad/s] (corresponding to two oscillations per time constant), or equivalently the poles of the transfer function are located near to

$$-\sigma \pm j\omega_d \stackrel{(4.12)}{=} -\frac{1}{T} \pm j\omega_d = -10 \pm j40\pi \quad (\text{P4.16.2})$$

(b) With the parameter values determined in (a), find the output voltage $v_o(t)$ to the DC input voltage of 1.41 V and plot it for the time interval $[0, 0.5 \text{ s}]$.

(c) Perform the PSpice simulation to get the output voltage $v_o(t)$ to the DC input voltage of 1.41 V for the time interval $[0, 0.5 \text{ s}]$ and find the oscillation period as the time between the first peak/trough time and the second one.

Magnetically Coupled Circuits

A magnetically (inductively) coupled inductor circuit consists of more than one coil of conductive wire wound on the same magnetic core. It presents the basis for the transformers that are used to increase/decrease AC (alternating current) voltages/currents. The coil on the input source side and that on the output load side are called the primary and secondary coils, respectively. The AC voltage on the primary coil causes the flux linkage to be changed continually, which induces the voltage on the secondary coil. The input voltage on the primary coil and the output voltage on the secondary coil are proportional to the number of windings of each coil. What is the difference between DC (direct current) and AC? AC means an electric current whose magnitude and direction change periodically, while DC means an electric current whose magnitude and direction do not change periodically.

It is not by coincidence but due to historical background that this chapter on the basic principle of the transformer falls here between the previous chapters on DC analysis and the next chapters on AC analysis. Late in the nineteenth century, there was a fierce competition, called the ‘War of Currents’, for leadership in the growing market of power transmission and distribution over North America between the two giants of electrical service, the Edison General Electric Company, who had established a DC power service system, and the Westinghouse Corporation, who had developed an AC power distribution system. According to websites such as References [W-1] and [W-5], Edison actively campaigned for the selection of the AC electric chair as a new executioner, hoping that AC would be known to be fatally dangerous and, thereby, consumers would not want to use AC. Edison even provided the AC generators that were needed for the first working electric chairs, although Westinghouse refused to sell any AC generators directly to prison authorities. Despite these attempts by Edison, who had developed the world’s first viable system of centrally generating and distributing electric power, the DC power system was completely defeated by the AC power system, due to transformers (developed by William Stanley) and AC generators/motors (invented by Nikola Tesla). With AC motors making large power use efficient and transformers stepping up/down the voltages of an AC power system for efficient power transmission/distribution, AC power technology was able to consolidate its superiority over DC power technology.

5.1 Self-Inductance

An inductor consists of a coiled conducting wire wound around a core, as illustrated in Figure 5.1(a). In the magnetic circuit made of the core, the *magnetic reluctance* and its reciprocal, called the *permeance*, are determined as

$$\mathcal{R} = \frac{l}{\mu A} [\text{A} \cdot \text{turns/Wb}] \quad \text{and} \quad \mathcal{P} = \frac{1}{\mathcal{R}} = \mu \frac{A}{l} [\text{Wb}/(\text{A} \cdot \text{turns})]$$

respectively, where l [m] and A [m²] are the (mean) length and cross-sectional area of the flux path through the core, and μ is the *permeability* of the core material. The *magnetomotive force* (mmf)

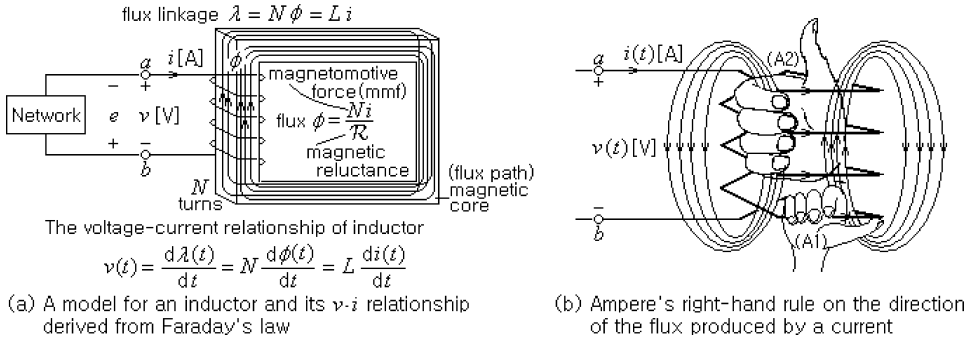


Figure 5.1 A model for an inductor

generated by the current i [A] flowing through the N -turn coil is Ni [A · turns] and produces a magnetic flux

$$\phi = \frac{Ni}{\mathcal{R}} = \mathcal{P} Ni \text{ [Wb]} \quad (5.1)$$

through the core in the direction determined by Ampere's right-hand rule (see Figure 5.1(b)). This is analogous to the electric current $i = V/R$ produced by an electromotive force (emf) V in the electric circuit having a resistance R .

The *flux linkage* of the N -turn coil is defined to be the flux times the number of turns as

$$\lambda = N\phi \text{ [Wb · turns]} = \mathcal{P} N^2 i = Li \quad (5.2)$$

where the (*self*)-*inductance* of the coil (inductor) is defined to be the constant of proportionality of the flux linkage to the current as

$$L[\text{H}] = \frac{\lambda}{i} \text{ [Wb · turns/A]} = \mathcal{P} N^2 \quad (5.3)$$

The unit of inductance is the henry (denoted by H), named in honor of the American physicist Joseph Henry (1797–1878). The flux 'linkage' stems from the fact that the flux ϕ through the core is linked with the current i through the N -turn coil.

Faraday's law states that the change in the flux linkage induces a voltage across the conductor (coil) linked with the flux, which equals the time rate of change of the flux linkage:

$$v(t) = \frac{d\lambda(t)}{dt} \stackrel{(5.2)}{=} N \frac{d\phi(t)}{dt} = L \frac{di(t)}{dt} \quad (5.4)$$

In fact, *Faraday's law* together with *Lenz's law* is described by the following equation:

$$e(t) = - \frac{d\lambda(t)}{dt}$$

where the negative sign means that the polarity of the induced voltage (emf) is such that it opposes the change of the flux linkage; i.e. it generates a current (through the external network) to produce a magnetic flux in the direction opposing the change of the flux linkage.

[Remark 5.1] Ampere's Right-Hand Rule on the Direction of the Magnetic Flux Produced by the Current

Ampere's right-hand rule describes the direction of the magnetic flux produced by a current flowing through a conductor. If the conductor is grasped with the right hand in such a way that the thumb points in the direction of the current, your fingers wrapping around the conductor curl in the direction of the magnetic flux (see (A1) in Figure 5.1(b)). If you curl the fingers of your right hand around a coil in the direction of the current, the thumb points in the direction of the magnetic flux (see (A2) in Figure 5.1(b)).

5.2 Mutual Inductance

In this section two coils are considered that are placed in proximity to each other or wound around a core as in Figure 5.2, where they have N_1 turns and N_2 turns, respectively. The flux ϕ_i linking each coil i is the sum or difference of two components, the *leakage flux* ϕ_{ii} produced by the current through the coil itself and the *mutual flux* ϕ_{ij} produced by the current through the other coil j :

$$\phi_1 = \phi_{11} \pm \phi_{12}, \quad \phi_2 = \pm \phi_{21} + \phi_{22} \quad (5.5)$$

where

ϕ_{11} = the leakage flux linking coil 1 and produced by the current i_1 through coil 1

ϕ_{12} = the mutual flux linking coil 1 and produced by the current i_2 through coil 2

ϕ_{21} = the mutual flux linking coil 2 and produced by the current i_1 through coil 1

ϕ_{22} = the leakage flux linking coil 2 and produced by the current i_2 through coil 2

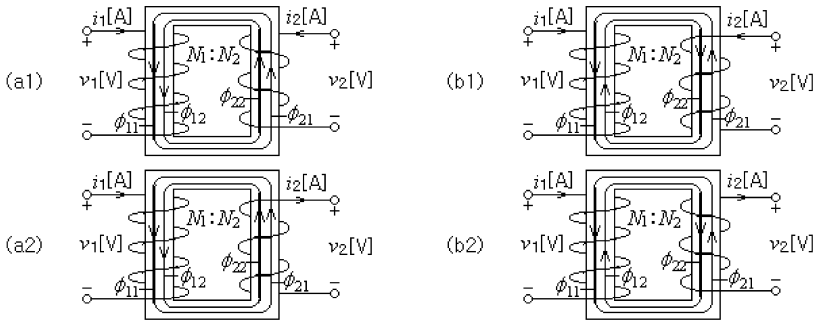
Thus the flux linkage of each coil can be written as

$$\lambda_1 = N_1 \phi_1 = N_1 (\phi_{11} \pm \phi_{12}) \stackrel{(5.2)}{=} L_1 i_1 \pm M_{12} i_2 \quad (5.6a)$$

$$\lambda_2 = N_2 \phi_2 = N_2 (\pm \phi_{21} + \phi_{22}) \stackrel{(5.2)}{=} \pm M_{21} i_1 + L_2 i_2 \quad (5.6b)$$

with the self-inductances L_1 and L_2 , the mutual inductance $M_{12} = M_{21} = M = k\sqrt{L_1 L_2}$, (5.7)

and the *coefficient of coupling* $k = \frac{M}{\sqrt{L_1 L_2}}$ ($0 \leq k \leq 1$) (5.8)



(a) Positive mutual inductance for additive coupling (b) Negative mutual inductance for subtractive coupling

Figure 5.2 Relative winding directions of magnetically coupled coils

where the signs of the mutual inductance terms are plus or minus depending on whether the fluxes produced by currents through two coils in the reference directions are additive or subtractive. Consequently, the induced voltages are the sum or difference of a self-induced one and a mutually induced one as

$$v_1(t) \stackrel{(5.4)}{=} \frac{d\lambda_1}{dt} \stackrel{(5.6a)}{=} L_1 \frac{di_1}{dt} \pm M \frac{di_2}{dt} \quad (5.9a)$$

$$v_2(t) \stackrel{(5.4)}{=} \frac{d\lambda_2}{dt} \stackrel{(5.6b)}{=} \pm M \frac{di_1}{dt} + L_2 \frac{di_2}{dt} \quad (5.9b)$$

5.3 Relative Polarity of Induced Voltages and Dot Convention

5.3.1 Dot Convention and Sign of Mutual Inductance Terms

The polarity of a mutually induced voltage relative to a self-induced one across a coil coupled magnetically with another coil depends on the relative winding and current directions of the two coils. Since it is cumbersome to draw the winding details as depicted in Figure 5.2, the *dot convention* is used to indicate the relative coil winding direction in the following way:

Dot (Marking) Convention:

The self-induced voltage and the mutually induced one are additive, i.e. have the same polarity if both coil currents enter/leave the dotted or undotted ends of the coils (Figure 5.3(a)). They are subtractive, i.e. opposed to each other, if one coil current enters/leaves the dotted end of a coil while the other coil current enters/leaves the undotted end of the other coil (Figure 5.3(b)).

5.3.2 Measurement of the Relative Winding Direction

Figure 5.4(a) shows a testing circuit to determine the relative winding direction of a pair of magnetically coupled coils that can be indicated by the dot marks. Let the switch be closed so that $i_1 > 0$ through coil 1 (with the dot on the upper side) produces some flux ϕ_{11} through the core. Then a current i_2 is supposed to be induced through coil 2 in such a direction that it will produce a flux ϕ_{22} opposing ϕ_{11} . There are two possible cases where a dot should be marked on the coil, on its upper or lower side:

1. If the voltmeter indicates a positive secondary voltage $v_2 > 0$, this implies that the current i_2 flows upward through coil 2. Since this current $i_2 < 0$ must have produced the flux ϕ_{22} opposing ϕ_{11} (Lenz's law), the dot on coil 2 should be marked on the upper side so that $i_2 < 0$ enters the undotted terminal of coil 2, while $i_1 > 0$ enters the dotted terminal of coil 1 (see Figure 5.4(b1)).
2. If the voltmeter indicates a negative secondary voltage $v_2 < 0$, it implies that the current i_2 flows downward through coil 2. Since this current $i_2 > 0$ must have produced the flux ϕ_{22} opposing ϕ_{11}

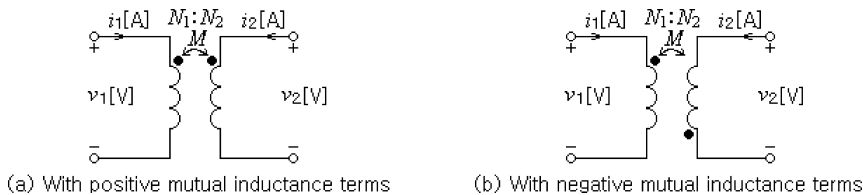


Figure 5.3 The sign of the mutual inductance terms depending on the current reference directions and relative winding directions of magnetically coupled coils expressed by the dot convention

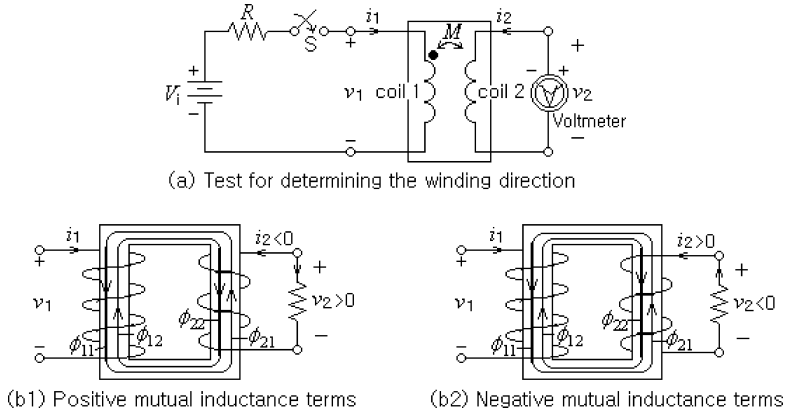


Figure 5.4 To find the relative winding directions of magnetically coupled coils

(Lenz's law), the dot on coil 2 should be marked on the lower side so that $i_2 > 0$ enters the undotted terminal of coil 2, while $i_1 > 0$ enters the dotted terminal of coil 1 (see Figure 5.4(b2)).

5.3.3 Measurement of Mutual Inductance

To find the mutual inductance of a pair of coupled coils, there is a need to find the difference between two resulting inductances measured for the two connections in Figures 5.5(a) and (b). The overall voltage–current relationship and the resulting inductance of the circuit connected as in Figure 5.5(a) is

$$\begin{aligned}
 v(t) &= v_1(t) - v_2(t) = \left(L_1 \frac{di_1}{dt} \pm M \frac{di_2}{dt} \right) - \left(\pm M \frac{di_1}{dt} + L_2 \frac{di_2}{dt} \right) \\
 (i_1=i, i_2=-i) &\left(L_1 \frac{di}{dt} \mp M \frac{di}{dt} \right) - \left(\pm M \frac{di}{dt} - L_2 \frac{di}{dt} \right) = (L_1 + L_2 \mp 2M) \frac{di}{dt} \\
 L_a &= L_1 + L_2 \mp 2M
 \end{aligned} \tag{5.10a}$$

The overall voltage–current relationship and the resulting inductance of the circuit connected as in Figure 5.5(b) is

$$\begin{aligned}
 v(t) &= v_1(t) + v_2(t) = \left(L_1 \frac{di_1}{dt} \pm M \frac{di_2}{dt} \right) + \left(\pm M \frac{di_1}{dt} + L_2 \frac{di_2}{dt} \right) \\
 (i_1=i, i_2=i) &\left(L_1 \frac{di}{dt} \pm M \frac{di}{dt} \right) + \left(\pm M \frac{di}{dt} + L_2 \frac{di}{dt} \right) = (L_1 + L_2 \pm 2M) \frac{di}{dt} \\
 L_b &= L_1 + L_2 \pm 2M
 \end{aligned} \tag{5.10b}$$

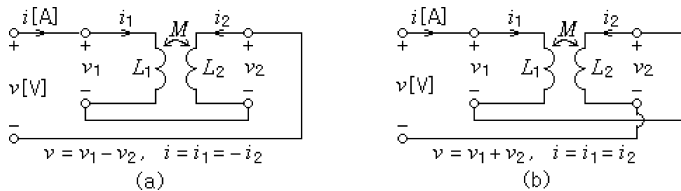


Figure 5.5 Test to measure a mutual inductance

Combining these two equations, $1/4$ is multiplied by the difference between these two resulting inductances to obtain the mutual inductance as

$$M = \frac{|L_a - L_b|}{4} \quad (5.11)$$

5.3.4 Energy in Magnetically Coupled Coils

Let us find the energy stored in a pair of magnetically coupled coils with the self-inductances L_1 and L_2 and the mutual inductance M . On the assumption of zero initial conditions, the total power delivered to the two coils is integrated to find the energy as

$$\begin{aligned} p_X(t) &= v_1(t)i_1(t) + v_2(t)i_2(t) \\ &= \left[L_1 \frac{di_1(t)}{dt} \pm M \frac{di_2(t)}{dt} \right] i_1(t) + \left[\pm M \frac{di_1(t)}{dt} + L_2 \frac{di_2(t)}{dt} \right] i_2(t) \\ &= L_1 i_1(t) \frac{d}{dt} i_1(t) \pm M \frac{d}{dt} [i_1(t) i_2(t)] + L_2 i_2(t) \frac{d}{dt} i_2(t) \\ w_X(t) &= \int_0^t p_X(\tau) d\tau \\ &= \int_0^t \left\{ L_1 i_1(\tau) \frac{d}{d\tau} i_1(\tau) \pm M \frac{d}{d\tau} [i_1(\tau) i_2(\tau)] + L_2 i_2(\tau) \frac{d}{d\tau} i_2(\tau) \right\} d\tau \\ &= \int_0^{i_1(t), i_2(t)} [L_1 i_1 di_1 \pm M d(i_1 i_2) + L_2 i_2 di_2] \\ &= \frac{1}{2} L_1 i_1^2 \pm M i_1 i_2 + \frac{1}{2} L_2 i_2^2 \end{aligned} \quad (5.12)$$

As a by-product, it can be shown that the (magnetic) *coupling coefficient* k cannot be greater than unity, based on this energy equation together with the fact that a pair of coupled coils is a passive element and so its energy can never be negative; i.e. noting that the above energy equation can be written as

$$w_X(t) = \frac{1}{2} \left(\sqrt{L_1} i_1 \pm \frac{M}{\sqrt{L_1}} i_2 \right)^2 + \frac{1}{2} \left(L_2 - \frac{M^2}{L_1} \right) i_2^2 \geq 0 \quad \forall i_1 \text{ and } i_2$$

and this must be nonnegative even for $i_1 = \mp(M/L_1)i_2$ (making the first squared term zero), the desired result is obtained:

$$L_2 - \frac{M^2}{L_1} \geq 0; \quad M^2 \leq L_1 L_2; \quad M \leq \sqrt{L_1 L_2} \xrightarrow{(5.7)} k \sqrt{L_1 L_2} \leq \sqrt{L_1 L_2}; \quad 0 \leq k \leq 1$$

5.4 Equivalent Models of Magnetically Coupled Coils

It is not straightforward to set up mesh equations or node equations for circuits containing coupled coils. It would be much easier if a pair of coupled coils is replaced by an equivalent that contains dependent voltage sources to account for the mutual inductance effect (Figure 5.6.1) or another equivalent that consists of three uncoupled inductors (Figure 5.6.2). Especially, the equivalents in Figures 5.6.1 and 5.6.2(a) are good for setting up mesh equations and the equivalent in Figure 5.6.2(b) is suitable for setting up node equations, where the signs of M should be the upper or lower ones of the double signs ($\pm$ or $\mp$) depending on whether the dots of both coils are on the same or opposite sides.

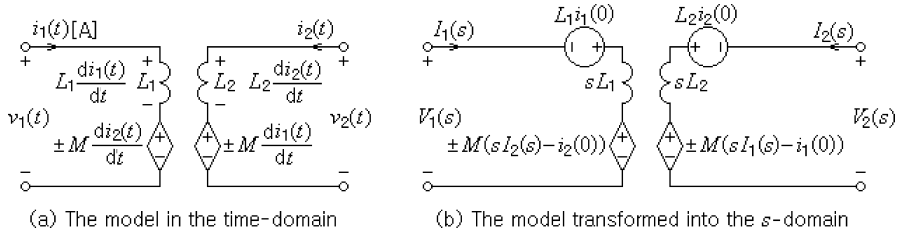


Figure 5.6.1 The equivalent model of two magnetically coupled coils with dependent sources

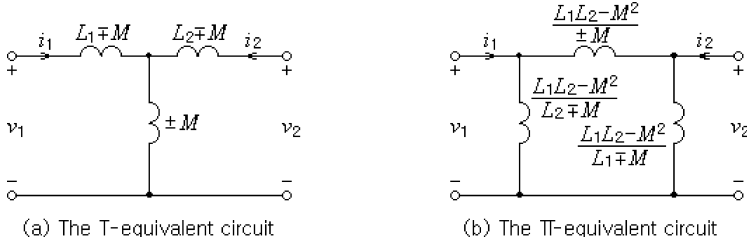


Figure 5.6.2 The equivalent model of two magnetically coupled coils with no dependent source

Note. Coupled inductors are mainly used for AC applications since coils are just like short-circuits in the DC steady state (Remark 3.2(2)).

5.4.1 T-Equivalent Circuit

Taking the Laplace transform of the time-domain voltage–current relationship (Equations (5.9)) of a pair of magnetically coupled coils yields

$$\begin{aligned} \begin{bmatrix} V_1(s) + L_1 i_1(0) \pm M i_2(0) \\ V_2(s) + L_2 i_2(0) \pm M i_1(0) \end{bmatrix} &= \begin{bmatrix} sL_1 & \pm sM \\ \pm sM & sL_2 \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} \\ \xrightarrow{\text{zero initial conditions}} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} &= \begin{bmatrix} sL_1 & \pm sM \\ \pm sM & sL_2 \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} \\ &= \begin{bmatrix} s(L_1 \mp M \pm M) & \pm sM \\ \pm sM & s(L_2 \mp M \pm M) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} \end{aligned} \quad (5.13)$$

This equation can be obtained by taking the s -domain equivalent of the model in Figures 5.6.1(a) or 5.6.2(a) and setting up the mesh equation for it. That is why the models are called the ‘equivalents’ of coupled circuits.

Figure 5.7.1(a) shows a circuit containing a pair of coupled coils with both dots marked on the same side. The mesh equation for its s -domain equivalent can be set up with the coupled coils replaced by the model in Figure 5.6.2(a) (with the upper one of the double signs before M), as depicted in Figure 5.7.1(b) as

$$\begin{bmatrix} R_1 + sL_1 & +sM \\ +sM & R_2 + sL_2 + 1/(sC_2) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_1(s) + L_1 i_1(0) + M i_2(0) \\ L_2 i_2(0) + M i_1(0) \end{bmatrix} \quad (5.14)$$

where the signs of the mutual inductance terms having M are positive since both currents $I_1(s)$ and $I_2(s)$ enter the dotted terminals of coil 1 and coil 2, respectively.

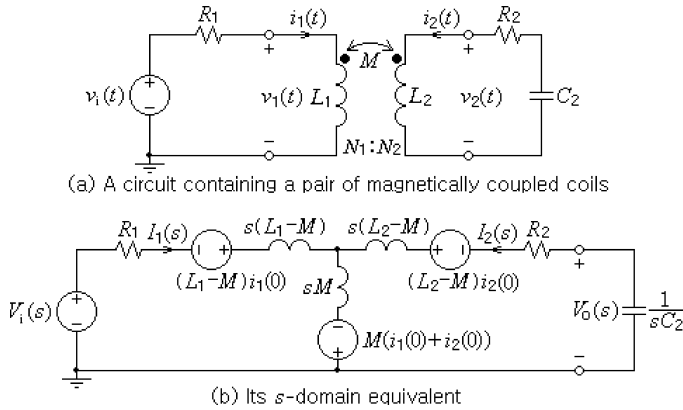


Figure 5.7.1 A circuit containing a pair of coupled coils and its s -domain equivalent

Figure 5.7.2(a) shows a circuit containing a pair of magnetically coupled coils with the two dots marked on opposite sides. The mesh equation for its s -domain equivalent can be set up with the coupled coils replaced by the model in Figure 5.6.2(a) (with the lower one of the double signs before M) as

$$\begin{bmatrix} R_1 + sL_1 & -sM \\ -sM & R_2 + sL_2 + 1/(sC_2) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_1(s) + L_1 i_1(0) - M i_2(0) \\ L_2 i_2(0) - M i_1(0) \end{bmatrix} \quad (5.15)$$

where the signs of the mutual inductance terms having M are negative since one current $I_1(s)$ enters the dotted terminal of coil 1 and the other current $I_2(s)$ enters the undotted terminal of coil 2. This is the same as obtained by negating the terms in $I_2(s)$ and $i_2(0)$ in Equation (5.14) (see Figure 5.7.2(b)):

$$\begin{bmatrix} R_1 + sL_1 & +sM \\ +sM & R_2 + sL_2 + 1/(sC_2) \end{bmatrix} \begin{bmatrix} I_1(s) \\ -I_2(s) \end{bmatrix} = \begin{bmatrix} V_1(s) + L_1 i_1(0) - M i_2(0) \\ -L_2 i_2(0) + M i_1(0) \end{bmatrix}$$

It is implied that switching the winding direction of one coil has the same effect as switching the reference direction of the current through the secondary coil (on the load side).

[Remark 5.2] Relative Winding and Current Reference Directions versus the Sign of Mutual Inductance

Consider the four quantities, i.e. the current reference directions and the winding directions (described by the dot convention) of the two coils. If two or four of them are changed, it makes no difference in the circuit equations for the coupled coils, but if one or three of them are changed, the sign of the terms involving the mutual inductance will be reversed.

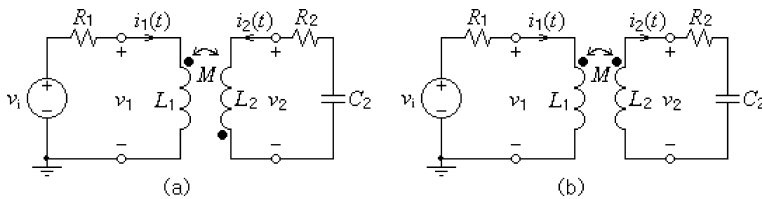


Figure 5.7.2 The same circuit as that of Figure 5.7.1(a), but with different winding direction or current reference direction

(Example 5.1) Mesh Analysis and Simulation of a Circuit Containing Coupled Coils

- (a) Consider the circuit of Figure 5.8.1(a) in which the switch is closed at $t = 0$ when the initial conditions are $i_{L1}(0) = 10/(1 + 1) = 5$ A and $i_{L2}(0) = 0$ A. Find $i_{L1}(t)$ and $i_{L2}(t)$ for $t \geq 0$.

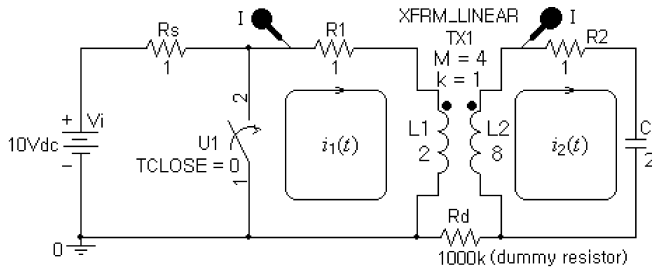
Like Equation (5.15), the mesh equation can be written and solved as

$$\begin{bmatrix} 1 + 2s & -4s \\ -4s & 1 + 8s + 1/(2s) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} L_1 i_1(0) - M i_2(0) \\ L_2 i_2(0) - M i_1(0) \end{bmatrix} = \begin{bmatrix} 2 \times 5 \\ -4 \times 5 \end{bmatrix} \quad (\text{E5.1.1})$$

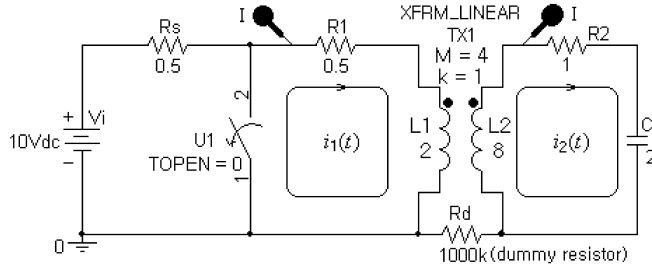
$$\begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \frac{1}{10s + 2 + 1/(2s)} \begin{bmatrix} 1 + 8s + 1/(2s) & 4s \\ 4s & 1 + 2s \end{bmatrix} \begin{bmatrix} 10 \\ -20 \end{bmatrix} = \frac{1}{20s^2 + 4s + 1} \begin{bmatrix} 10s + 5 \\ -20s \end{bmatrix}$$

$$= \frac{1}{(s + 1/10)^2 + (1/5)^2} \begin{bmatrix} (s + 1/10) + 2(1/5) \\ -2(s + 1/10) + (1/5) \end{bmatrix}$$

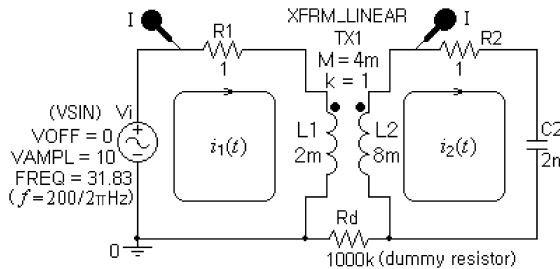
$$\begin{bmatrix} i_1(t) \\ i_2(t) \end{bmatrix} = \begin{bmatrix} e^{-t/10} [\cos(t/5) + 2 \sin(t/5)] \\ e^{-t/10} [-2 \cos(t/5) + \sin(t/5)] \end{bmatrix} u_s(t) \text{ [A]} \quad (\text{E5.1.2})$$



(a) A circuit containing a pair of coupled coils excited by an initial condition



(b) A circuit containing a pair of coupled coils excited by a DC voltage source



(c) A circuit containing a pair of coupled coils excited by an AC voltage source

Figure 5.8.1 Circuits containing a pair of coupled coils for Example 5.1

The following MATLAB program `cir05e01a.m` can be run to obtain the same results and plot them for the time interval $[0, 100\text{ s}]$, as depicted in Figure 5.8.2(a):

```
>>cir05e01a
    i1=exp(-1/10*t)*cos(1/5*t)+2*exp(-1/10*t)*sin(1/5*t)
    i2=-2*exp(-1/10*t)*cos(1/5*t)+exp(-1/10*t)*sin(1/5*t)

%cir05e01a.m
% To solve a circuit with magnetically coupled coils and SW (Ex 5.1a)
clear, clf
syms s
Rs=1; R1=1; R2=1; L1=2; L2=8; M=4; C2=2;
i10=5; i20=0; % Initial conditions
Zs=[R1+s*L1 -s*M; -s*M R2+1/s/C2+s*L2]
Vs=[L1*i10-M*i20; L2*i20-M*i10]
Is=Zs\Vs % solution of Eq. (E5.1.1)
i1=ilaplace(Is(1)), i2=ilaplace(Is(2))
t0=0; tf=100; N=500; tt=t0+[0:N]*(tf-t0)/N;
for n=1:length(tt)
    t=tt(n); i1t(n)=eval(i1); i2t(n)=eval(i2);
end
subplot(221), plot(tt,i1t, tt,i2t), axis([0 100 -2 1.5])
```

- (b) Consider the circuit of Figure 5.8.1(b) in which the switch is open at $t = 0$ with zero initial conditions $i_{L1}(0) = 0\text{ A}$ and $i_{L2}(0) = 0\text{ A}$. Find $i_{L1}(t)$ and $i_{L2}(t)$ for $t \geq 0$.

Like Equation (5.15), the mesh equation can be written and solved as

$$\begin{bmatrix} 1+2s & -4s \\ -4s & 1+8s+1/(2s) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_i(s) \\ 0 \end{bmatrix} = \begin{bmatrix} 10/s \\ 0 \end{bmatrix} \quad (\text{E5.1.3})$$

$$\begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \frac{1}{10s+2+1/(2s)} \begin{bmatrix} 1+8s+1/(2s) & 4s \\ 4s & 1+2s \end{bmatrix} \begin{bmatrix} 10/s \\ 0 \end{bmatrix}$$

$$= \frac{1}{s[(s+1/10)^2 + (1/5)^2]} \begin{bmatrix} 8s^2 + s + 1/2 \\ 4s^2 \end{bmatrix} = \begin{bmatrix} \frac{10}{s} - \frac{2(s+1/10) + 4(1/5)}{(s+1/10)^2 + (1/5)^2} \\ \frac{4(s+1/10) - 2(1/5)}{(s+1/10)^2 + (1/5)^2} \end{bmatrix}$$

$$\begin{bmatrix} i_1(t) \\ i_2(t) \end{bmatrix} = \begin{bmatrix} 10 - e^{-t/10}[2\cos(t/5) + 4\sin(t/5)] \\ e^{-t/10}[4\cos(t/5) - 2\sin(t/5)] \end{bmatrix} u_s(t) \text{ [A]} \quad (\text{E5.1.4})$$

MATLAB may be used to obtain the same results and plot them for the time interval $[0, 100\text{ s}]$, as depicted in Figure 5.8.2(b):

```
>>cir05e01b
    i1 = -2*exp(-1/10*t)*cos(1/5*t)-4*exp(-1/10*t)*sin(1/5*t)+10
    i2 = 4*exp(-1/10*t)*cos(1/5*t)-2*exp(-1/10*t)*sin(1/5*t)
```

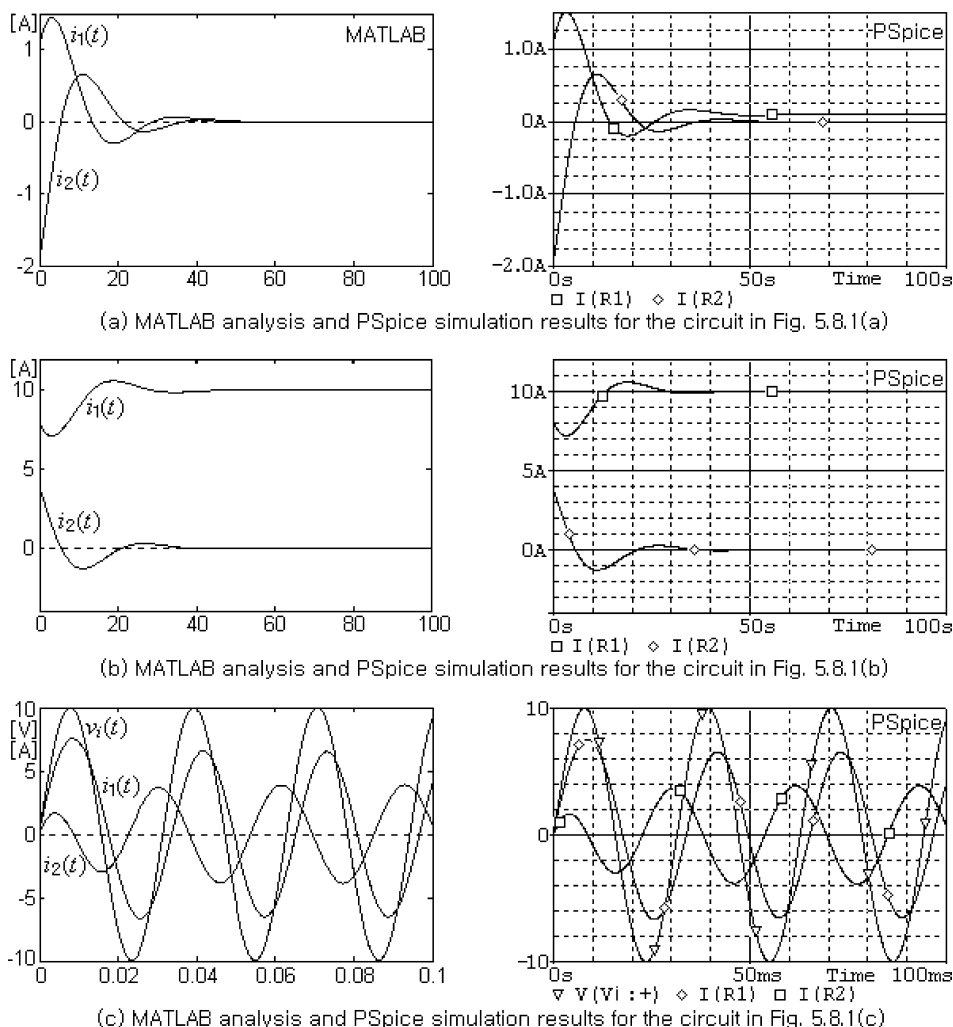



Figure 5.8.2 MATLAB analysis and PSpice simulation results for Example 5.1

- (c) Consider the circuit of Figure 5.8.1(c), where a sinusoidal voltage source $v_1(t) = 10 \sin(\omega t)$ [V] with $\omega = 200 = 2\pi f$ [rad/s] ($f = 200/2\pi \simeq 31.83$ Hz) is applied at $t = 0$ when the initial conditions are zero, i.e. $i_{L1}(0) = 0$ A and $i_{L2}(0) = 0$ A. Find $i_{L1}(t)$ and $i_{L2}(t)$ for $t \geq 0$.

Like Equation (5.15), the mesh equation can be written and solved as

$$\begin{bmatrix} 1 + 0.002s & -0.004s \\ -0.004s & 1 + 0.008s + 10^3/(2s) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_1(s) = 2000/(s^2 + 200^2) \\ 0 \end{bmatrix} \quad (\text{E5.1.5})$$

Even with such unrealistically simple values of the parameters, the computation involved in solving this equation and taking the inverse Laplace transform to obtain $i_{L1}(t)$ and $i_{L2}(t)$ seems to be quite involved and there might be a need to resort to MATLAB. The MATLAB program `cir05e01c.m` that follows is run to get the following results and plot them as depicted in Figure 5.8.2(c):

```
>>cir05e01c
i1=-52/17*cos(200t)+98/17*sin(200t)+52/17*exp(-100t)*cos(200t)+64/17*exp(-100t)*sin(200t)
i2=64/17*cos(200t)-16/17*sin(200t)-64/17*exp(-100t)*cos(200t)+52/17*exp(-100t)*sin(200t)
```

```
%cir05e01c.m
% To solve a circuit with coupled coils & a sinusoidal input (Ex 5.1c)
clear, clf
syms s
Rs=0; R1=1; R2=1; L1=0.002; L2=0.008; M=0.004; C2=0.002;
Zs=[Rs+R1+s*L1 -s*M; -s*M R2+1/s/C2+s*L2]
w=200; Vs=[10*w/(s^2+w^2); 0]
Is=Zs\Vs % Eq. (E5.1.5)
i1=ilaplace(Is(1)); i2=ilaplace(Is(2)); pretty(i1), pretty(i2)
t0=0; tf=0.1; N=500; tt=t0+[0:N]*(tf-t0)/N;
for n=1:length(tt)
    t=tt(n); i1t(n)=eval(i1); i2t(n)=eval(i2);
end
vit = 10*sin(w*tt);
subplot(223), plot(tt,vit, tt,i1t, tt,i2t)
```

Note. The circuit diagrams of Figure 5.8.1 are PSpice schematics and the PSpice simulation results are depicted side by side with the graphs obtained by using MATLAB in Figure 5.8.2.

Note. The coupled coils in Figure 5.8.1 are connected via a dummy resistor of extra-large resistance as required by the PSpice rule that every node should have a DC path to the ground (see Remark H.2 in Appendix H).

Note. Figure 5.8.2 illustrates that the currents in the coupled coils may change instantaneously at $t = 0$, which is a surprising violation of the continuity rule of inductor currents. See Problem 5.12 for details.

5.4.2 Π -Equivalent Circuit

Equation (5.13) with zero initial conditions can be solved to give the expression of $[I_1(s) \ I_2(s)]$ in terms of $[V_1(s) \ V_2(s)]$ as

$$\begin{aligned} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} &= \frac{1}{s^2(L_1L_2 - M^2)} \begin{bmatrix} sL_2 & \mp sM \\ \mp sM & sL_1 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} \\ &= \frac{1}{s(L_1L_2 - M^2)} \begin{bmatrix} L_2 \mp M \pm M & \mp M \\ \mp M & L_1 \mp M \pm M \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} \\ \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} &= \begin{bmatrix} 1/(sL_a) + 1/(sL_c) & -1/(sL_c) \\ -1/(sL_c) & 1/(sL_b) + 1/(sL_c) \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} \end{aligned} \quad (5.16)$$

$$\text{with } L_a = \frac{L_1L_2 - M^2}{L_2 \mp M}, \quad L_b = \frac{L_1L_2 - M^2}{L_1 \mp M}, \quad \text{and } L_c = \frac{L_1L_2 - M^2}{\mp M} \quad (5.17)$$

This node equation directly corresponds to the Π -model for a pair of two magnetically coupled coils in Figure 5.6.2(b), which can be used to set up the node equation for circuits containing coupled coils. Note

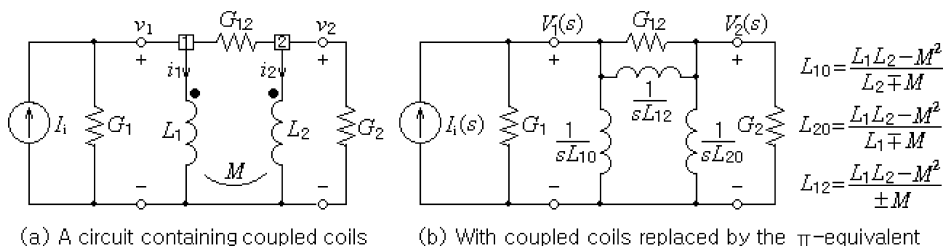


Figure 5.9 A circuit containing a pair of coupled coils

that the sign of M should be negative or positive depending on whether the dots denoting the relative winding directions of two coils are on the same or opposite sides.

For confirmation and practice, set up the node equation for the circuit in Figure 5.9(a). First, regarding the two coil currents i_1 and i_2 as given, KCL is applied to nodes 1 and 2 to write

$$\begin{aligned} I_1(s) + (G_1 + G_{12})V_1(s) - G_{12}V_2(s) &= I_i(s) \\ I_2(s) - G_{12}V_1(s) + (G_2 + G_{12})V_2(s) &= 0 \end{aligned}$$

Substituting Equation (5.16) for $I_1(s)$ and $I_2(s)$ into this equation yields

$$\begin{bmatrix} G_1 + G_{12} + \frac{L_2}{s(L_1L_2 - M^2)} & -G_{12} \mp \frac{M}{s(L_1L_2 - M^2)} \\ -G_{12} \mp \frac{M}{s(L_1L_2 - M^2)} & G_2 + G_{12} + \frac{L_1}{s(L_1L_2 - M^2)} \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} = \begin{bmatrix} I_i(s) \\ 0 \end{bmatrix} \quad (5.18)$$

This is identical to the node equation for the circuit of Figure 5.9(b), in which the pair of two coupled circuits is replaced by the Π -model in Figure 5.6.2(b) consisting of three uncoupled coils.

(Example 5.2) Node Analysis and Simulation of a Circuit Containing Coupled Circuits

Consider the circuit in Figure 5.10.1(a1) where $R_1 = 10\Omega$, $R_{12} = 10\Omega$, $R_2 = 5\Omega$, $L_1 = 1\text{ H}$, $L_2 = 2\text{ H}$, $M = 1\text{ H}$, and the current source of 10 A is applied at $t = 0$ when the initial conditions are zero; i.e. $i_{L1}(0) = 0\text{ A}$ and $i_{L2}(0) = 0\text{ A}$. Find $v_1(t)$ and $v_2(t)$ for $t \geq 0$.

From Equation (5.18) with $G_1 = 1/10\text{ S}$, $G_{12} = 1/10\text{ S}$, $G_2 = 1/5\text{ S}$, $L_1 = 1\text{ H}$, $L_2 = 2\text{ H}$, $M = 1\text{ H}$, and $I_i(s) = 10/s[\text{A}]$, the node equation can be written and solved as

$$\begin{bmatrix} 0.1 + 0.1 + 2/s & -0.1 - 1/s \\ -0.1 - 1/s & 0.1 + 0.2 + 1/s \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} = \begin{bmatrix} 10/s \\ 0 \end{bmatrix} \quad (\text{E5.2.1})$$

$$\begin{aligned} \begin{bmatrix} V_1(s) \\ V_2(s) \end{bmatrix} &= \frac{1}{0.05 + 0.06/s + 1/s^2} \begin{bmatrix} 0.3 + 1/s & 0.1 + 1/s \\ 0.1 + 1/s & 0.2 + 2/s \end{bmatrix} \begin{bmatrix} 10/s \\ 0 \end{bmatrix} \\ &= \frac{20}{(s+2)(s+10)} \begin{bmatrix} 3s+10 \\ s+10 \end{bmatrix} = \begin{bmatrix} 10/(s+2) + 50/(s+10) \\ 20/(s+2) \end{bmatrix} \\ \begin{bmatrix} v_1(t) \\ v_2(t) \end{bmatrix} &= \begin{bmatrix} 10e^{-2t} + 50e^{-10t} \\ 20e^{-2t} \end{bmatrix} u_s(t) [\text{V}] \quad (\text{E5.2.2}) \end{aligned}$$

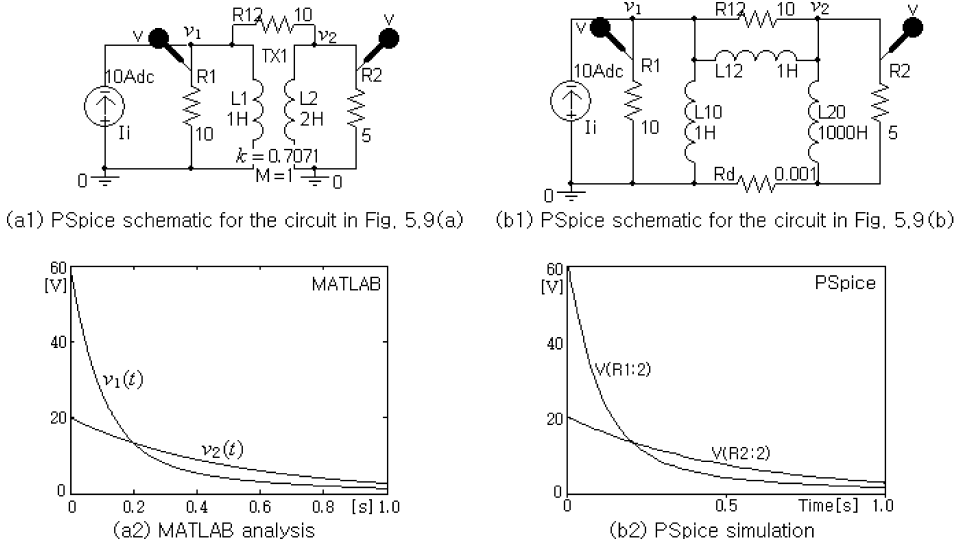


Figure 5.10.1 Simulation of a circuit containing a pair of coupled coils

Readers are invited to compose a MATLAB program named, say, `cir05e02.m` to get the same results and plot them for the time interval $[0, 1\text{s}]$, as depicted in Figure 5.10.1(a2):

```
>>cir05e02
v1 = 20*exp(-6*t)*(3*cosh(4*t) - 2*sinh(4*t))
v2 = 20*exp(-2*t)
```

Note. Note that $\cosh 4t = (e^{4t} + e^{-4t})/2$ and $\sinh 4t = (e^{4t} - e^{-4t})/2$.

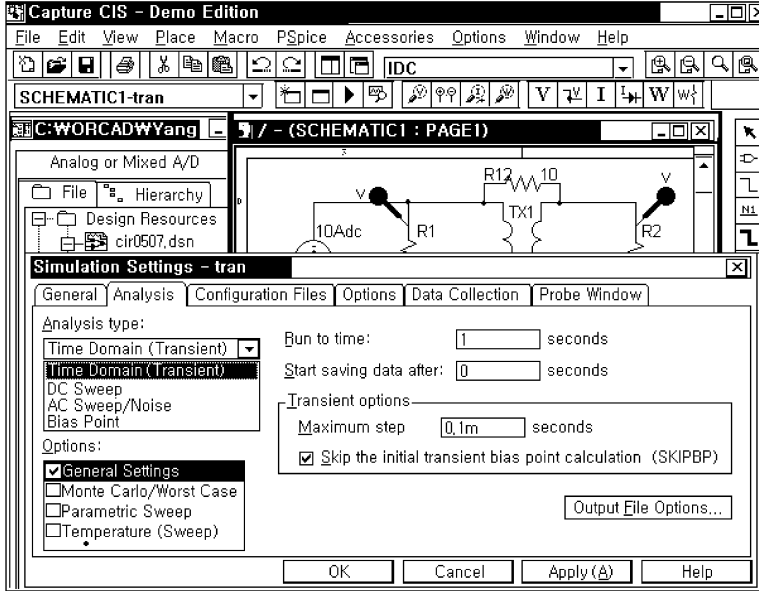
Figures 5.10.1(a1) and (b1) are PSpice schematics themselves, where the latter one has the Π -model for the pair of coupled coils. Since the two coupled coils are interconnected via R_{12} , unlike those in Figure 5.8.1, they do not have to be connected via a dummy resistor of extra-large resistance, but have to be connected via a dummy resistor of extra-small resistance, or each of the two coils can be grounded separately. Especially when the two coils L_{10} and L_{20} are shorted between their lower parts in the PSpice schematic in Figure 5.10.1(b1), it will cause a run-time error since any loop consisting of inductors only (L_{10} – L_{12} – L_{20}) is rejected by PSpice (Remark H.2(3)). Figure 5.10.2(a) shows the Property Editor spreadsheet for the pair of coupled coils in Figure 5.10.1(a1), where the two inductances are set to $L_1 = 1\text{ H}$ and $L_2 = 2\text{ H}$, respectively and the coefficient of coupling is set to

$$k = \frac{M}{\sqrt{L_1 L_2}} = \frac{1}{\sqrt{2}} \simeq 0.7071 \quad (\text{E5.2.3})$$

Figure 5.10.2(b) shows the Simulation Settings dialog box, where the square box before SKIPBP (Skip the initial transient bias point calculation) is checked.

Property Editor					
New Column...		Apply	Display...	Delete Property	Filter
		< Current properties >			
		COUPLING	L1_VALUE	L2_VALUE	Part Reference
1	SCHEMATIC1 : PAGE1 : TX1	0, 7071	1H	2H	TX1
					Value
					XFRM_LINEAR

(a) Property Editor spreadsheet for the pair of coupled coils in Fig. 5.10.1(a1)



(b) Simulation Settings dialog box to set Analysis type to Time Domain (Transient)

Figure 5.10.2 Property Editor spreadsheet and simulation setting dialog box for PSpice simulation of Figure 5.10.1(a1)

5.5 Ideal Transformer

The conditions for a pair of coupled coils to be an *ideal transformer* are as follows:

1. The two coils are perfectly coupled with the unity coefficient of coupling $k = 1$:

$$M \stackrel{(5.7)}{\text{with } k=1} \sqrt{L_1 L_2} \quad (5.19a)$$

2. The permeance $\mathcal{P}$ of the magnetic core around which the two coils are wound is ∞ or, equivalently, the reluctance is $\mathcal{R} = 1/\mathcal{P} = 0$ so that the magnetomotive force (mmf) around the magnetic circuit is zero however large the flux ϕ may be:

$$N_1 i_1 \pm N_2 i_2 \stackrel{(5.1)}{=} \phi \mathcal{R} = 0; \quad \frac{i_2}{i_1} = \mp \frac{N_1}{N_2} \quad (5.19b)$$

3. No energy is stored or dissipated in the two coils, which means that all the power received at one (primary or source) side is instantly transferred to the other (secondary or load) side:

$$p(t) = v_1 i_1 + v_2 i_2 = 0, \quad v_1 i_1 = -v_2 i_2, \quad \frac{v_2}{v_1} = -\frac{i_1}{i_2} \quad (5.19c)$$

Noting from Equation (5.3) that the self-inductance of each coil wound around a common core (with permeance $\mathcal{P}$) is proportional to the square of the number of turns, we have

$$L_2 = \frac{N_2^2}{N_1^2} L_1, \quad M \stackrel{(5.19a)}{=} \sqrt{L_1 L_2} = \frac{N_2}{N_1} L_1, \quad \text{and} \quad L_2 = \frac{N_2}{N_1} M \quad (5.20)$$

so that the voltage–current relationship of an ideal transformer can be written as

$$\begin{aligned} v_1(t) &\stackrel{(5.9a)}{=} L_1 \frac{di_1}{dt} \pm M \frac{di_2}{dt} \\ v_2(t) &\stackrel{(5.9b)}{\stackrel{(5.20)}{=}} \pm \frac{N_2}{N_1} L_1 \frac{di_1}{dt} + \frac{N_2}{N_1} M \frac{di_2}{dt} = \pm \frac{N_2}{N_1} v_1(t) \end{aligned}$$

This relationship between the primary–secondary voltages can be obtained by substituting Equation (5.19b) into Equation (5.19c). It is implied by this result and Equation (5.19c) that the primary–secondary voltages of an ideal transformer are proportional to the number of turns of coil winding, while the primary–secondary currents are inversely proportional to the number of turns of coil winding, which can be summarized as below, where the *turns ratio* is defined as $a = N_1/N_2$.

Relationships between the primary–secondary voltages and currents of an ideal transformer:

$$\frac{v_2}{v_1} = \pm \frac{N_2}{N_1} = \pm \frac{1}{a} \quad (5.21a)$$

$$\frac{i_2}{i_1} = \mp \frac{N_1}{N_2} = \mp a \quad \text{with the turns ratio } a = \frac{N_1}{N_2} \quad (5.21b)$$

Note. The upper/lower signs apply for the case of positive/negative mutual inductance, respectively.

These relationships between the primary–secondary voltages and currents can be modeled by the circuits containing dependent sources, as shown in Figures 5.11(a) and (b).

Based on the voltage–current transformation properties (5.21a) and (5.21b), another important property of *impedance transformation* (or *multiplication* or *scaling*) possessed by an ideal transformer

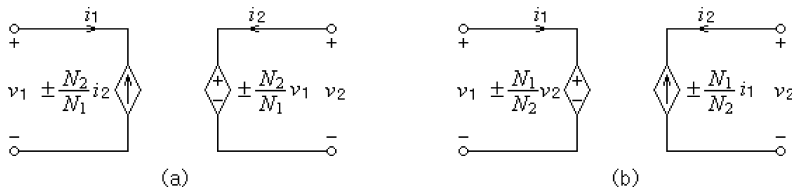


Figure 5.11 Dependent source models for an ideal transformer

will be derived. For this purpose, consider the ideal transformer circuit of Figure 5.12(a) in which the voltage–current relationship of the load is written as

$$V_2(s) = Z_L(s)I_L(s) = -Z_L(s)I_2(s)$$

With Equation (5.21b) this can be substituted into Equation (5.21a) to get

$$\begin{aligned} V_1(s) &\stackrel{(5.21a)}{=} \frac{N_1}{N_2} V_2(s) = \frac{N_1}{N_2} [-Z_L(s)I_2(s)] \stackrel{(5.21b)}{=} -\frac{N_1}{N_2} Z_L(s) \left[-\frac{N_1}{N_2} I_1(s) \right] \\ V_1(s) &= \left(\frac{N_1}{N_2} \right)^2 Z_L(s) I_1(s) \end{aligned} \quad (5.22)$$

Thus the secondary (load) impedance reflected to the primary (source) side is

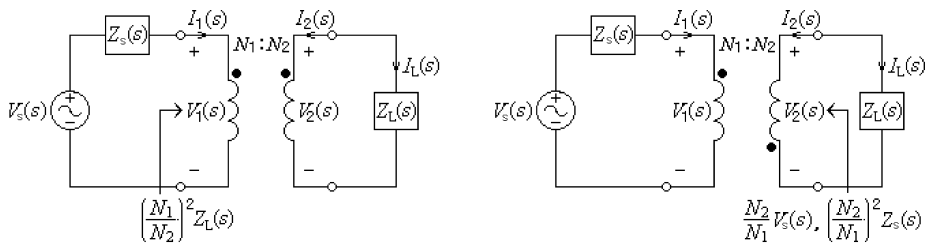
$$Z_{12}(s) = \frac{V_1(s)}{I_1(s)} = \left(\frac{N_1}{N_2} \right)^2 Z_L(s) = a^2 Z_L(s) \quad \text{with } a = \frac{N_1}{N_2} \quad (5.23)$$

This *reflected impedance* implies that, from the primary (source) side, the secondary (load) impedance is seen to be multiplied by the squared turns ratio (a^2). This property is not only helpful in understanding the basic function of a transformer but is also useful in the realization of maximum power transfer or impedance matching, which will be discussed in Section 6.7.

Similarly, referring to Figure 5.12(b), use can be made of Equations (5.21a) and (5.21b) to find the Thevenin equivalent of the transformer circuit seen from the secondary (load) side as

$$\begin{aligned} V_2(s) &\stackrel{(5.21a)}{=} -\frac{N_2}{N_1} V_1(s) = -\frac{N_2}{N_1} [V_s(s) - Z_s(s)I_1(s)] \stackrel{(5.21b)}{=} -\frac{N_2}{N_1} \left[V_s(s) - Z_s(s) \frac{N_2}{N_1} I_2(s) \right] \\ V_2(s) &= -\frac{N_2}{N_1} V_s(s) + \left(\frac{N_2}{N_1} \right)^2 Z_s(s) I_2(s) = -\frac{1}{a} V_s(s) + \frac{1}{a^2} Z_s(s) I_2(s) \end{aligned} \quad (5.24)$$

This implies that from the secondary (load) side, the source voltage and the primary (source) impedance are seen to be multiplied by the reverse turns ratio $1/a$ and the reverse squared turns ratio $1/a^2$, respectively. This property can be used to eliminate the transformer in order to simplify the analysis of a transformer circuit. However, it does not apply in the case where there is some external connection between the two coils.



(a) Load impedance reflected into the primary side (b) Source impedance reflected into the secondary side

Figure 5.12 Impedance transformation (multiplication) by an ideal transformer

(Example 5.3) Electric Power Transmission with High Voltage Using Transformers

Consider the circuit of Figure 5.12(a) in which the turns ratio of the ideal transformer is $a = N_1/N_2 = 30/1$, the primary voltage is $V_1 = 6600$ V, and the load impedance is a resistance of $Z_L = 22 \Omega$. The secondary voltage/current and the primary current are

$$V_2 \stackrel{(5.21a)}{=} \frac{N_2}{N_1} V_1 = \frac{1}{30} 6600 = 220 \text{ V}, \quad I_2 = -I_L = -\frac{V_2}{Z_L} = -\frac{220}{22} = -10 \text{ A}$$

and

$$I_1 \stackrel{(5.21b)}{=} -\frac{N_2}{N_1} I_2 = \frac{1}{3} \text{ A} \quad (\text{E5.3.1})$$

respectively. The load impedance reflected to the primary (source) side is

$$Z_{12} = \frac{V_1}{I_1} \stackrel{(5.23)}{=} a^2 Z_L = 30^2 \times 22 = 19\,800 \Omega \quad (\text{E5.3.2})$$

This implies that despite the high primary voltage, the load impedance seen from the primary side is magnified $a^2 = 900$ times the original value, so that the primary current is merely 1/3 A, much less than the secondary (load) current $I_L = V_2/Z_L = 10$ A.

Note. If the primary current flows through a long transmission line from the generator, this small current I_1 will be good for decreasing the transmission loss as well as the voltage drop. This is why the high voltage transmission is adopted for large power systems.

5.6 Linear Transformer

Figure 5.13 shows a more realistic model for a transformer, which contains the internal coil resistances. To find the reflected impedance, the voltage gain, and the current gain, all the initial conditions are neglected, the mesh equation is set up in the primary/secondary currents $I_1(s)$ and $I_2(s)$, and it is solved as

$$\begin{aligned} [Z_s(s) + R_1 + sL_1]I_1(s) + sM I_2(s) &= V_s(s) \\ sM I_1(s) + [R_2 + sL_2 + Z_L(s)]I_2(s) &= 0 \\ \begin{bmatrix} Z_{11}(s) & sM \\ sM & Z_{22}(s) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} &= \begin{bmatrix} V_s(s) \\ 0 \end{bmatrix} \\ \text{with } Z_{11}(s) &= Z_s(s) + R_1 + sL_1 \text{ and } Z_{22}(s) = R_2 + sL_2 + Z_L(s) \\ \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} &= \frac{1}{\Delta} \begin{bmatrix} Z_{22}(s) & -sM \\ -sM & Z_{11}(s) \end{bmatrix} \begin{bmatrix} V_s(s) \\ 0 \end{bmatrix} = \frac{V_s(s)}{Z_{11}(s)Z_{22}(s) - s^2M^2} \begin{bmatrix} Z_{22}(s) \\ -sM \end{bmatrix} \end{aligned} \quad (5.25)$$

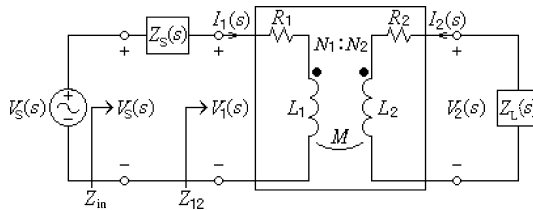


Figure 5.13 The s -domain linear transformer model

Thus the input impedance of the overall circuit seen from the source is

$$\begin{aligned} Z_{in}(s) &= \frac{V_s(s)}{I_1(s)} = \frac{Z_{11}(s)Z_{22}(s) - s^2M^2}{Z_{22}(s)} = Z_{11}(s) - \frac{s^2M^2}{Z_{22}(s)} \\ &= Z_s(s) + R_1 + sL_1 - \frac{s^2M^2}{sL_2 + R_2 + Z_L(s)} \end{aligned} \quad (5.26)$$

Neglecting the source impedance $Z_s(s)$ and the internal resistances R_1 and R_2 and substituting the conditions of an ideal transformer

$$L_1 = N_1^2 \mathcal{P}, \quad L_2 = N_2^2 \mathcal{P} (\mathcal{P} = \infty), \quad \text{and} \quad M \stackrel{(5.19a)}{=} \sqrt{L_1 L_2} \quad (5.27)$$

into Equation (5.26) yields the load impedance reflected to the primary (source) side as

$$Z_{12}(s) = sL_1 - \frac{s^2M^2}{sL_2 + Z_L(s)} \stackrel{L_1 L_2 = M^2}{=} \frac{sL_1 Z_L(s)}{sL_2 + Z_L(s)} \stackrel{L_2 \gg Z_L}{\approx} \frac{L_1 Z_L(s)}{L_2} = \left(\frac{N_1}{N_2} \right)^2 Z_L(s) \quad (5.28)$$

which agrees with Equation (5.23) for an ideal transformer.

The *voltage gain*, i.e. the ratio of the secondary (load) voltage to the source voltage, is

$$\begin{aligned} A_v &= \frac{V_2(s)}{V_s(s)} = \frac{-I_2 Z_L(s)}{V_s(s)} \stackrel{(5.25)}{=} \frac{sM Z_L(s)}{Z_{11}(s)Z_{22}(s) - s^2M^2} \\ &= \frac{sM Z_L(s)}{[Z_s(s) + R_1 + sL_1][R_2 + sL_2 + Z_L(s)] - s^2M^2} \end{aligned} \quad (5.29)$$

and the *current gain*, i.e. the ratio of the secondary (load) current to the primary (source) current is

$$A_i = \frac{I_2(s)}{I_1(s)} \stackrel{(5.25)}{=} \frac{-sM}{Z_{22}(s)} = -\frac{sM}{R_2 + sL_2 + Z_L(s)} \quad (5.30)$$

Neglecting the source impedance $Z_s(s)$ and the internal resistances R_1 and R_2 and substituting the ideal transformer conditions (5.27) into these two equations, (5.29) and (5.30), yields

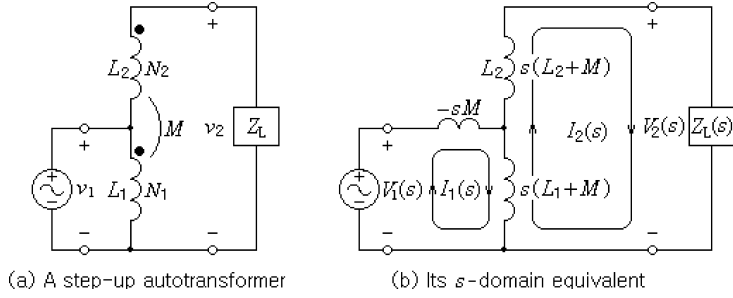
$$A_v = \frac{V_2(s)}{V_s(s)} \simeq \frac{sM Z_L(s)}{sL_1 Z_L(s)} = \frac{N_2}{N_1} \quad (5.31a)$$

$$A_i = \frac{I_2(s)}{I_1(s)} \simeq \frac{-sM}{sL_2} = -\frac{N_1}{N_2} \quad (5.31b)$$

which agree with the primary–secondary voltage and current relationship, (5.21a) and (5.21b), for an ideal transformer.

5.7 Autotransformers

Figures 5.14(a) and 5.15(a) show a step-up autotransformer and a step-down one, respectively, in which the primary and secondary coils have some or all windings in common and the turns ratio depends on the position of the connection point called a *tap*. Compared with a conventional two-winding transformer with the same turns ratio, an autotransformer is lighter, smaller, and less costly because it requires both


Figure 5.14 A step-up autotransformer and its s -domain equivalent

fewer windings and a smaller core. On the other hand, it does not have the function of electrical isolation to reduce the risk of shock hazard or to remove the DC influence of one side on the other.

The coupled coils in Figure 5.14(a) can be replaced by the T-equivalent (Figure 5.6.2(a)), as depicted in Figure 5.14(b), and the mesh equation set up as

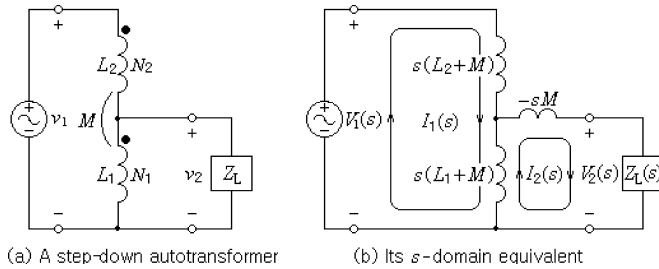
$$\begin{bmatrix} sL_1 & -s(L_1 + M) \\ -s(L_1 + M) & s(L_1 + 2M + L_2) + Z_L(s) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_1(s) \\ 0 \end{bmatrix} \quad (5.32)$$

which is solved to find the primary/secondary currents, the voltage gain, and the current gain as

$$\begin{aligned} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} &= \frac{1}{s(L_1L_2 - M^2) + sL_1Z_L(s)} \begin{bmatrix} s(L_1 + 2M + L_2) + Z_L(s) & s(L_1 + M) \\ s(L_1 + M) & sL_1 \end{bmatrix} \begin{bmatrix} V_1(s) \\ 0 \end{bmatrix} \\ &= \frac{V_1(s)}{s(L_1L_2 - M^2) + sL_1Z_L(s)} \begin{bmatrix} s(L_1 + 2M + L_2) + Z_L(s) \\ s(L_1 + M) \end{bmatrix} \end{aligned} \quad (5.33)$$

$$A_i = \frac{I_2(s)}{I_1(s)} \stackrel{(5.33)}{=} \frac{s(L_1 + M)}{s(L_1 + 2M + L_2) + Z_L(s)} \quad (5.34)$$

$$A_v = \frac{V_2(s)}{V_1(s)} = \frac{I_2(s)Z_L(s)}{V_1(s)} \stackrel{(5.33)}{=} \frac{s(L_1 + M)Z_L(s)}{s(L_1L_2 - M^2) + sL_1Z_L(s)} \quad (5.35)$$


Figure 5.15 A step-down autotransformer and its s -domain equivalent

Substituting the ideal transformer conditions (5.27) into these two equations yields

$$A_i = \frac{I_2(s)}{I_1(s)} \stackrel{(5.34) \text{ with } (5.27)}{=} \frac{L_1 + \sqrt{L_1 L_2}}{L_1 + 2\sqrt{L_1 L_2} + \sqrt{L_2}} = \frac{\sqrt{L_1}}{\sqrt{L_1} + \sqrt{L_2}} = \frac{N_1}{N_1 + N_2} \quad (5.36)$$

$$A_v = \frac{V_2(s)}{V_1(s)} \stackrel{(5.35) \text{ with } (5.27)}{=} \frac{L_1 + M}{L_1} = \frac{N_1^2 + N_1 N_2}{N_1^2} = \frac{N_1 + N_2}{N_1} \quad (5.37)$$

Likewise, the coupled coils in Figure 5.15(a) can be replaced by the T-equivalent (Figure 5.6.2(a)), as depicted in Figure 5.15(b), and the mesh equation is set up as

$$\begin{bmatrix} s(L_1 + 2M + L_2) & -s(L_1 + M) \\ -s(L_1 + M) & sL_1 + Z_L(s) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_1(s) \\ 0 \end{bmatrix} \quad (5.38)$$

which is solved to find the primary/secondary currents, the voltage gain, and the current gain as

$$\begin{aligned} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} &= \frac{1}{\Delta} \begin{bmatrix} sL_1 + Z_L(s) & s(L_1 + M) \\ s(L_1 + M) & s(L_1 + 2M + L_2) \end{bmatrix} \begin{bmatrix} V_1(s) \\ 0 \end{bmatrix} \\ &= \frac{V_1(s)}{s^2(L_1 L_2 - M^2) + s(L_1 + 2M + L_2)Z_L(s)} \begin{bmatrix} sL_1 + Z_L(s) \\ s(L_1 + M) \end{bmatrix} \end{aligned} \quad (5.39)$$

$$A_i = \frac{I_2(s)}{I_1(s)} \stackrel{(5.39)}{=} \frac{s(L_1 + M)}{sL_1 + Z_L(s)} \quad (5.40)$$

$$A_v = \frac{V_2(s)}{V_1(s)} = \frac{I_2(s)Z_L(s)}{V_1(s)} \stackrel{(5.39)}{=} \frac{s(L_1 + M)Z_L(s)}{s^2(L_1 L_2 - M^2) + s(L_1 + 2M + L_2)Z_L(s)} \quad (5.41)$$

Substituting the ideal transformer conditions (5.27) into these two equations yields

$$A_i = \frac{I_2(s)}{I_1(s)} \stackrel{(5.40) \text{ with } (5.27)}{=} \frac{L_1 + \sqrt{L_1 L_2}}{L_1} = \frac{N_1^2 + N_1 N_2}{N_1^2} = \frac{N_1 + N_2}{N_1} \quad (5.42)$$

$$A_v = \frac{V_2(s)}{V_1(s)} \stackrel{(5.41) \text{ with } (5.27)}{=} \frac{L_1 + M}{L_1 + 2M + L_2} = \frac{N_1^2 + N_1 N_2}{N_1^2 + 2N_1 N_2 + N_2^2} = \frac{N_1}{N_1 + N_2} \quad (5.43)$$

Note. A failure of insulation for windings of an autotransformer may cause the full source voltage/current to be applied to the load side.

Problems

5.1 Series Connections of Coupled Coils

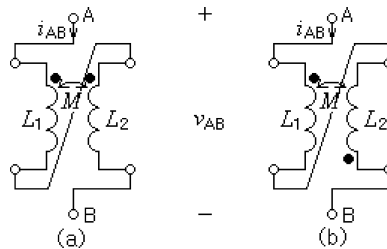


Figure P5.1 Series connections of two coupled coils

Find the voltage–current relationships for the two series connections of two coupled coils, one with positive mutual inductance and the other with negative mutual inductance, in Figures P5.1(a) and (b) and verify that their equivalent inductances are

$$L_{eq1} = L_1 + L_2 + 2M \quad \text{and} \quad L_{eq2} = L_1 + L_2 - 2M \quad (\text{P5.1.1})$$

respectively.

5.2 Parallel Connections of Coupled Coils

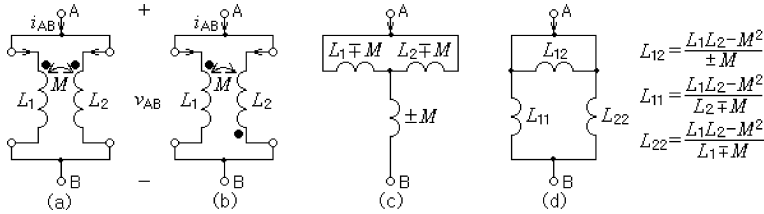


Figure P5.2 Parallel connections of two coupled coils and their equivalent circuits

- (a) Find the voltage–current relationships for the two parallel connections of two coupled coils, one with positive mutual inductance and the other with negative mutual inductance, in Figures P5.2(a) and (b) and verify that their equivalent inductances are

$$L_{eq1} = \frac{L_1 L_2 - M^2}{L_1 + L_2 - 2M} \quad \text{and} \quad L_{eq2} = \frac{L_1 L_2 - M^2}{L_1 + L_2 + 2M} \quad (\text{P5.2.1})$$

respectively.

- (b) Noting that the circuit of Figure P5.2(c) is obtained by replacing the coupled coils with its T-equivalent in Figure 5.6.2(a), find the parallel–series combination of the three inductances to verify that it is identical with what is obtained in (a).
(c) Noting that the circuit of Figure P5.2(d) is obtained by replacing the coupled coils with its Π -equivalent in Figure 5.6.2(b), find the parallel combination of the two inductances L_{11} and L_{22} to verify that it is identical with what is obtained in (a).
(d) Referring to the Δ -Y conversion formula (6.22), show that the two circuits in Figures P5.2(c) and (d) are equivalent to each other.

5.3 A Circuit Containing a Pair of Coupled Coils

Consider the circuit of Figure P5.3(a).

- (a) Figures P5.3(b) and (c) show the two equivalent circuits of the circuit in Figure P5.3(a), one with the pair of coupled coils replaced by its T-equivalent in Figure 5.6.2(a) and the other with the pair of coupled coils replaced by its Π -equivalent in Figure 5.6.2(b). Determine the sign of the mutual inductance in each of them (see Sections 5.4.1 and 5.4.2).
(b) Suppose the switch has been connected to the source side for a long time and is then flipped to the ground side at $t = 0$. Referring to the s -domain equivalent in Figure P5.3(b), write a set of mesh equations in $I_1(s)$ and $I_2(s)$ and solve it to get $I_2(s)$ and finally $V_3(s)$. Also, referring to the s -domain equivalent in Figure P5.3(c), write a set of node equations in $V_1(s)$, $V_2(s)$, and $V_3(s)$ and solve it to get $V_3(s)$. Also find $v_3(t)$.
(c) Suppose the switch has been connected to the ground side for a long time and is then flipped to the source side at $t = 0$. Repeat the same job as in (b) to get $V_3(s)$.

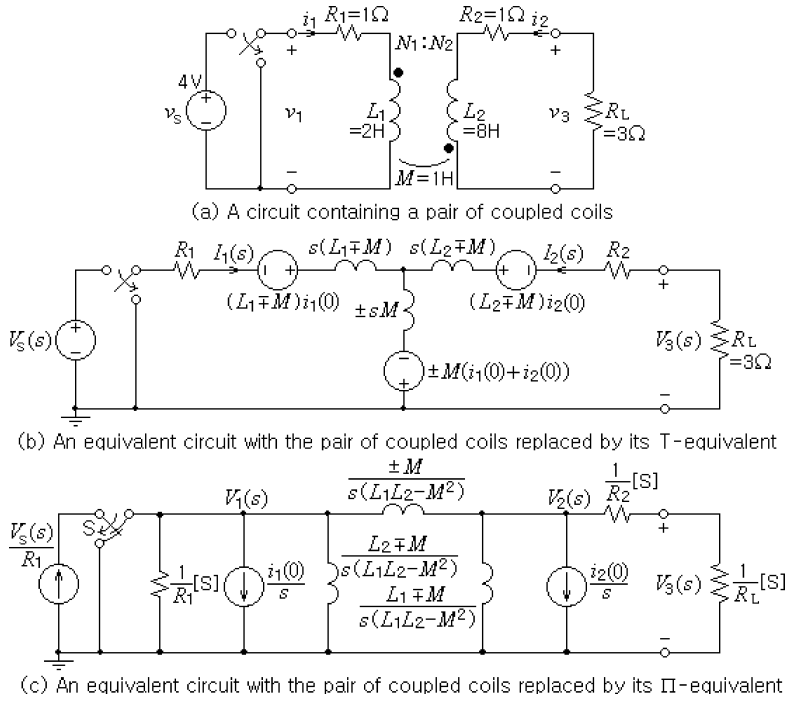


Figure P5.3

Note. Readers are encouraged to use MATLAB or its equivalent to solve these problems.

5.4 Perfectly Coupled Coils

Consider the circuit of Figure P5.4.

- With zero initial conditions, write a set of mesh equations and solve it for $I_1(s)$ and $I_2(s)$.
- Find the primary and secondary voltages $V_1(s)$ and $V_2(s)$ using the following relationships:

$$V_1(s) = sL_1 I_1(s) + sM I_2(s), \quad V_2(s) = -R_2 I_2(s) \quad (\text{P5.4.1})$$

- Verify that if only the perfect coupling condition $M = \sqrt{L_1 L_2}$ is satisfied, the following hold:
 - The denominators of $I_1(s)$, $I_2(s)$, $V_1(s)$, and $V_2(s)$ are all first-degree polynomials, implying that the circuit is not a second-order system, but a first-order system.
 - The ratio of the primary and secondary voltages equals the turns ratio $N_1 : N_2$.

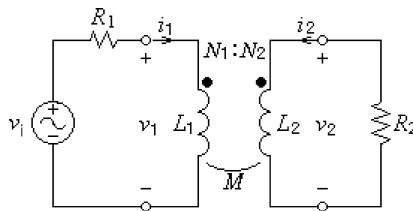


Figure P5.4 Perfectly coupled coils

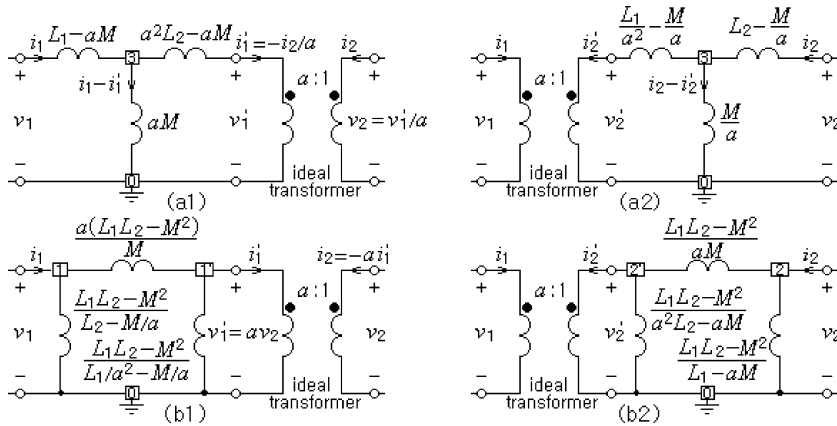


Figure P5.5.1 Equivalent models for coupled coils, each consisting of inductors and an ideal transformer with $M/L_2 < a < L_1/M$

5.5 Equivalents of a Pair of Coupled Coils Using an Ideal Transformer

It is difficult to realize the equivalents of a pair of coupled coils in Figure 5.6.2 (Section 5.4) when they need a negative (mutual) inductance. By contrast, there is no such problem with the equivalents in Figure P5.5. Verify that the circuits in Figures P5.5.1(a) and (b) have the same voltage–current relationships as Equations (5.13) and (5.16), implying that they are exactly the equivalents of a pair of coupled coils. Note that the sign of the mutual inductance is taken over by the relative winding direction of the ideal transformer.

- (a) Consider the circuit of Figure P5.5.1(a1). The voltage at node 3 is the voltage drop across the coil of inductance aM (between node 3 and node 0) caused by the current $I_1 - I_1' = I_1 + I_2/a$ as

$$V_3 = saM(I_1 + I_2/a) \quad (\text{P5.5.1})$$

Adding the voltage drop across the coil of inductance $L_1 - aM$ (between the positive input terminal and node 3) caused by the current I_1 to V_3 yields

$$V_1 = V_3 + s(L_1 - aM)I_1 \stackrel{(\text{P5.5.1})}{=} saM(I_1 + I_2/a) + s(L_1 - aM)I_1 \quad (\text{P5.5.2})$$

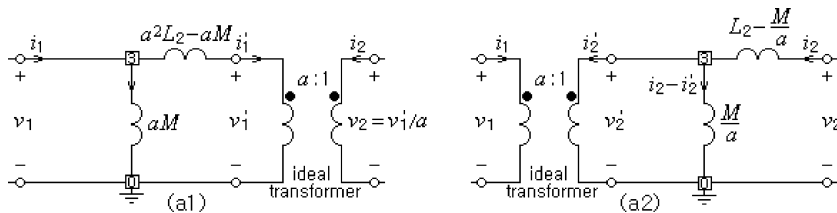


Figure P5.5.2 Equivalent models for coupled coils, each consisting of inductors and an ideal transformer with $a = L_1/M$

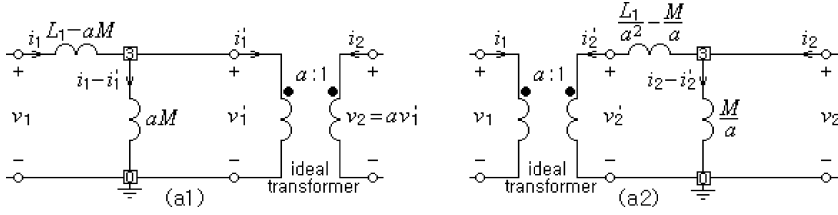


Figure P5.5.3 Equivalent models for coupled coils, each consisting of inductors and an ideal transformer with $a = M/L_2$

The primary voltage of the ideal transformer is obtained by subtracting the voltage drop across the coil of inductance $a^2L_2 - aM$ caused by the current $I'_1 = -I_2/a$ as

$$\begin{aligned} V'_1 &= V_3 - s(a^2L_2 - aM)(-I_2/a) \stackrel{(P5.5.1)}{=} (saM)(I_1 + I_2/a) + s(a^2L_2 - aM)I_2/a \\ &= (saM)I_1 + (sL_2)I_2 \end{aligned} \quad (P5.5.3)$$

Thus the secondary voltage V_2 of the ideal transformer, which is proportional to the primary voltage V'_1 with the proportionality constant $N_2/N_1 = 1/a$ (the reverse turns ratio), is obtained as

$$V_2 \stackrel{(5.21a)}{=} \frac{1}{a} V'_1 \stackrel{(P5.5.3)}{=} sM I_1 + sL_2 I_2 \quad (P5.5.4)$$

Verify that Equations (P5.5.2) and (P5.5.4) conform with Equation (5.13) and all the inductances of the circuit in Figure P5.5.1(a2) are $(1/a^2)$ times those of the circuit in Figure P5.5.1(a1).

(b) Consider the circuit of Figure P5.5.1(b1). KCL can be applied to node 1 to get I_1 as

$$\begin{aligned} I_1 &= \frac{V_1}{s(L_1L_2 - M^2)/(L_2 - M/a)} + \frac{V_1 - aV_2}{sa(L_1L_2 - M^2)/M} \\ &= \frac{L_2}{s(L_1L_2 - M^2)} V_1 - \frac{M}{s(L_1L_2 - M^2)} V_2 \end{aligned} \quad (P5.5.5)$$

KCL can also be applied to node 1' to get the primary current I'_1 of the ideal transformer as

$$I'_1 = \frac{V_1 - aV_2}{sa(L_1L_2 - M^2)/M} - \frac{aV_2}{s(L_1L_2 - M^2)/(L_1/a^2 - M/a)} = \frac{MV_1/a - L_1V_2/a}{s(L_1L_2 - M^2)} \quad (P5.5.6)$$

Thus the secondary current I_2 of the ideal transformer, which is proportional to the primary current I'_1 with the proportionality constant $N_1/N_2 = a$ (the turns ratio), is obtained as

$$I_2 \stackrel{(5.21b)}{=} -aI'_1 \stackrel{(P5.5.6)}{=} -\frac{M}{s(L_1L_2 - M^2)} V_1 + \frac{L_1}{s(L_1L_2 - M^2)} V_2 \quad (P5.5.7)$$

Verify that Equations (P5.5.5) and (P5.5.7) conform with Equation (5.16) and all the inductances of the circuit in Figure P5.5.1(b2) are $(1/a^2)$ times those of the circuit in Figure P5.5.1(b1).

(c) In fact, it was shown in (a) and (b) that the circuits in Figure P5.5.1 are all equivalent to a pair of coupled coils, each with self-inductance L_1 and L_2 and mutual inductance M , and that it is valid irrespective of the value of the turns ratio a . Besides, none of the inductances are negative as long as the turns ratio satisfies the following condition:

$$\frac{M}{L_2} \leq a \leq \frac{L_1}{M} \quad (P5.5.8)$$

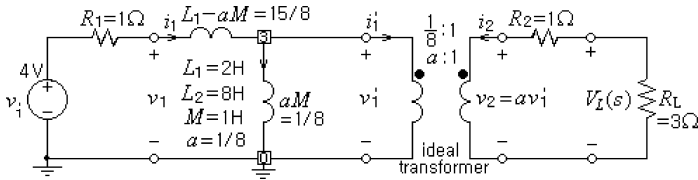


Figure P5.6

Verify that, especially for $a = L_1/M$, the equivalents in Figures P5.5.1(a1) and (a2) become the circuits with two inductors as depicted in Figure P5.5.2(a1) and (a2). Verify that especially for $a = M/L_2$, the equivalents in Figures P5.5.1(a1) and (a2) become the circuits with two inductors as depicted in Figure P5.5.3(a1) and (a2).

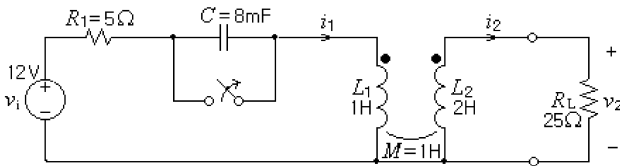
5.6 A Circuit with a Pair of Coupled Coils Replaced by Its Equivalent Having an Ideal Transformer

The pair of coupled coils in Figure P5.3 can be replaced by its equivalent (with $a = 1/8$) in Figure P5.5.3(a1) to obtain the circuit of Figure P5.6. To test for the validity, find the voltage $V_L(s)$ across the load resistor $R_L = 3\Omega$.

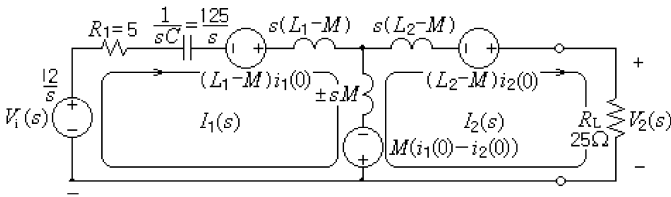
5.7 A Circuit Containing a Pair of Coupled Coils

Consider the circuit of Figure P5.7(a) in which the switch has been closed for a long time before being opened at $t = 0$.

- Find the initial conditions and represent them by the voltage sources in the s -domain equivalent as depicted in Figure P5.7(b), where the pair of coupled coils is replaced by its T-equivalent.
- Determine the sign of the mutual inductance and write a set of mesh equations for the s -domain circuit depicted in Figure P5.7(b) and solve it to find $I_2(s)$ and then $V_2(s) = R_L I_2(s)$. Take the inverse Laplace transform of $V_2(s)$ to get $v_2(t)$ and plot it for the time interval $[0, 1\text{ s}]$ by using MATLAB or its equivalent.
- Support the analysis result obtained in (b) with the PSpice simulation. Do not check the square box before SKIPBP (Skip the initial transient bias point calculation) in the Simulation Settings dialog box since the initial conditions should be calculated.



(a) A circuit containing a set of coupled coils



(b) An equivalent circuit with the pair of coupled coils replaced by its T-equivalent

Figure P5.7

5.8 A Circuit Containing a Pair of Coupled Coils

Consider the circuit of Figure P5.8(a) in which the initial conditions are all assumed to be zero.

- Figure P5.8(b) shows an equivalent of the circuit in Figure P5.8(a) with the pair of coupled coils replaced by its T-equivalent in Figure 5.6.2(a). Write a set of mesh equations and solve it to find $I_2(s)$ and then $V_2(s) = R_L I_2(s)$.
- Figure P5.8(c) shows another equivalent of the circuit in Figure P5.8(a) with the pair of coupled coils replaced by its Π -equivalent. Write a set of node equations and solve it to find $V_2(s)$.
- Take the inverse Laplace transform of $V_2(s)$ to get $v_2(t)$ and plot it for the time interval $[0, 20\text{ s}]$ by using MATLAB or its equivalent. Does the `ilaplace()` function work properly? If not, you can use the function `[r,p,k]=residue()` or `ilaplace_my()` as illustrated in the following program `cir05p08c.m`, where the user needs to type in the coefficient vectors of the numerator and denominator polynomials of the rational function in s to be taken for the inverse Laplace transform.

```
%cir05p08c.m
clear, clf
t0=0; tf=40; N=500; t=t0+[ 0:N] * (tf-t0)/N; % time vector
B=[4 0 2], A=[6 4 4 1] % coefficient vectors of numerator/denominator
[r,p,k]=residue(B,A) % partial fraction expansion  $\sum r(i)/(s-p(i))$ 
vRLt=real(r.*exp(p*t)); % make sure vRL(t) real
% Alternatively,
vRLt1=eval(ilaplace_my(B,A)); vRLt2=eval(ilaplace_my(VRLs));
plot(t,vRLt, t,vRLt1, 'r', t,vRLt2, 'm')
```

```
function x=ilaplace_my(B,A)
% B,A: the coefficient vectors of numerator/denominator polynomials in s
% Copyleft: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only
if ~isnumeric(B), [B,A]=numden(simple(B)); B=sym2poly(B); A=sym2poly(A); end
[r,p,k]=residue(B,A); N=length(r); x=[]; EPS=1e-15;
for n=1:N
    if n<N & abs(imag(r(n)))>EPS & abs(sum(imag(r([n n+1]))))<EPS
        sgm=num2str(real(p(n))); w=num2str(imag(p(n)));
        Kc=num2str(2*real(r(n))); Ks=num2str(-2*imag(r(n)));
        if abs(sgm)>EPS % exp(sgm*t)*(Kc*cos(w*t)+Ks*sin(w*t))
            x=[x '+' exp(' sgm '*t).*( ' Kc '*cos(' w '*t) + ' Ks '*sin(' w '*t)')'];
        else % Kc*cos(w*t)+Ks*sin(w*t)
            x=[x '+' Kc'*cos(' w '*t) + ' Ks '*sin(' w '*t)'];
        end
    elseif n<=N & abs(imag(r(n)))<EPS
        if abs(p(n))>EPS % r(n)*exp(p(n)*t)
            x=[x '+' num2str(r(n)) ' *exp(' num2str(p(n)) '*t)'];
        else % r(n) with the pole of s=0
            x=[x '+' num2str(r(n))];
        end
    end
end
if ~isempty(k), x=[x '+' num2str(k(end)) ' *dirac(t)']; end
```

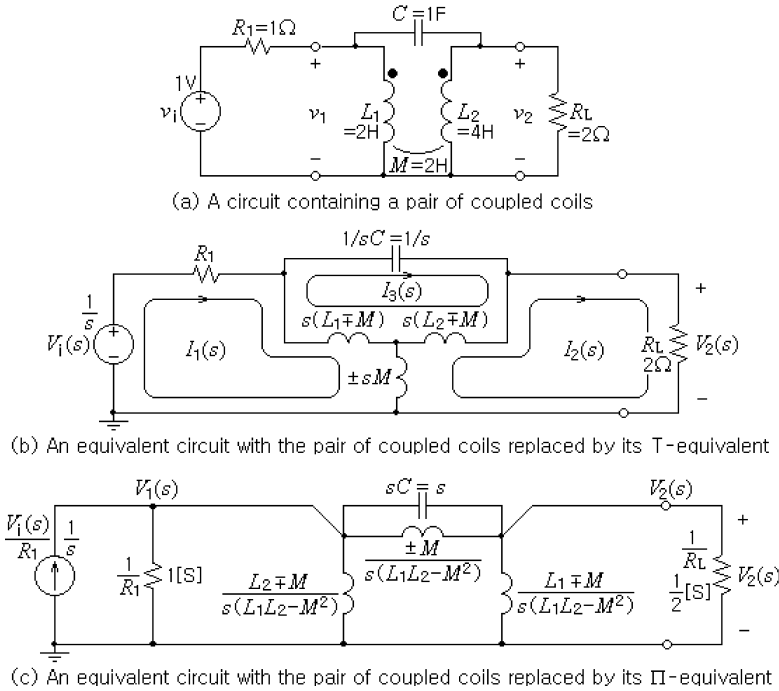


Figure P5.8

- Perform the PSpice simulation to get $v_2(t)$ for the time interval $[0, 20 \text{ s}]$. Do not forget to check the square box before SKIPBP (Skip the initial transient bias point calculation) in the Simulation Settings dialog box. Does the waveform of $v_2(t)$ look similar to that obtained in (c)?
- On the assumption that the dots denoting the relative winding directions of the two coupled coils are on the opposite side, say one on the upper side and the other on the lower side, do the same things as were done in (a) to (d).

5.9 A Circuit Containing a Pair of Coupled Coils

Consider the circuit of Figure P5.9(a) in which the initial conditions are all assumed to be zero.

- Figure P5.9(b) shows an equivalent of the circuit of Figure P5.9(a) with the pair of coupled coils replaced by its T-equivalent as in Figure 5.6.2(a) and Figure P5.9(c) another one with the mutually induced voltages replaced by the dependent voltage sources as in Figure 5.6.1. Show that a set of mesh equations in $I_1(s)$ and $I_2(s)$ can be written as follows:

$$\begin{bmatrix} R_1 + 1/(sC) + sL_1 & -s(L_1 + M) \\ -s(L_1 + M) & R_2 + s(L_1 + L_2 + 2M) \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_1(s) \\ 0 \end{bmatrix} \quad (\text{P5.9.1})$$

- Substitute the values of the components in Equation (P5.9.1) and solve it to find $I_2(s)$. Take the inverse Laplace transform of $I_2(s)$ to get $i_2(t)$ and plot it for the time interval $[0, 20 \text{ s}]$ by using MATLAB or its equivalent.
- Referring to Figure P5.9(d), perform the PSpice simulation to get $i_2(t)$ for the time interval $[0, 20 \text{ s}]$. Do not forget to check the square box before SKIPBP (Skip the initial transient bias point

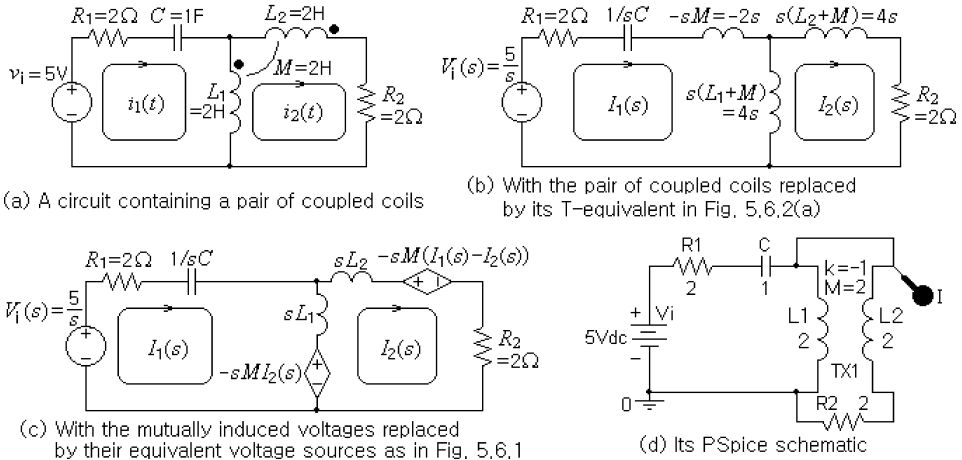


Figure P5.9

calculation) in the Simulation Settings dialog box. Does the waveform of $i_2(t)$ look similar to that obtained in (b)?

5.10 A Circuit Containing a Pair of Coupled Coils

Consider the circuit of Figure P5.10(a) in which the initial conditions are all assumed to be zero.

- (a) Figure P5.10(b) shows an equivalent of the circuit of Figure P5.10(a) with the mutually induced voltages replaced by the dependent voltage sources as in Figure 5.6.1. Show that a set of mesh equations in $I_1(s)$ and $I_2(s)$ can be written as follows:

$$\begin{bmatrix} R_1 + sL_1 & -s(L_1 - M) \\ -s(L_1 - M) & 1/(sC) + s(L_1 + L_2 - 2M) + R_2 \end{bmatrix} \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \begin{bmatrix} V_1(s) \\ 0 \end{bmatrix} \quad (\text{P5.10.1})$$

- (b) Substitute the values of the components into Equation (P5.10.1) and solve it to find $I_2(s)$. Take the inverse Laplace transform of $I_2(s)$ to get $i_2(t)$ and plot it for the time interval $[0, 20]$ s by using MATLAB or its equivalent.
- (c) Perform the PSpice simulation to get $i_2(t)$ for the time interval $[0, 20]$ s. Do not forget to check the square box before SKIPBP (Skip the initial transient bias point calculation) in the Simulation Settings dialog box. Does the waveform of $i_2(t)$ look similar to that obtained in (b)?
- (d) On the assumption that the dots denoting the relative winding directions of the two coupled coils are on the opposite side, say one on the upper side and the other on the lower side, do the same things as were done in (a) to (c).

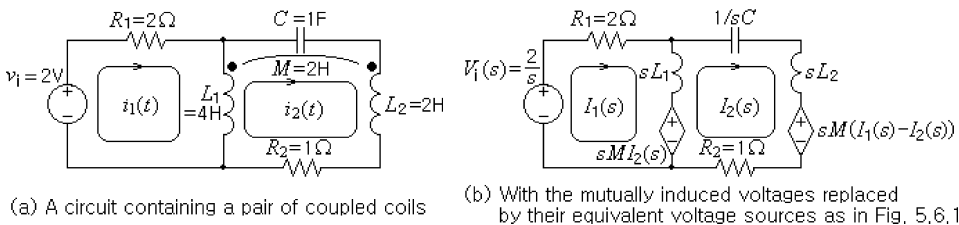


Figure P5.10

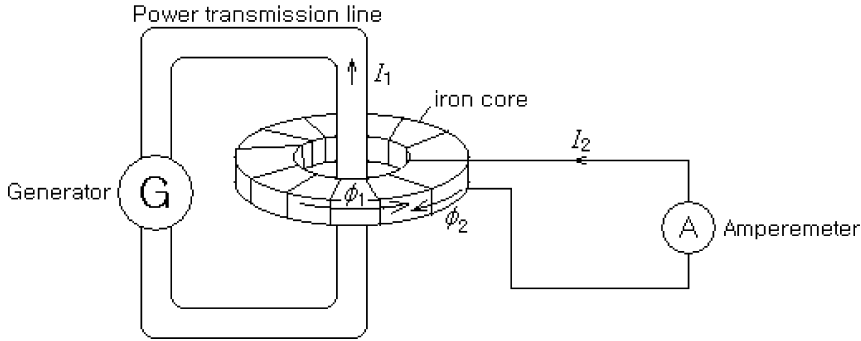


Figure P5.11 A current transformer used for measuring a large current

5.11 Current Transformer

Consider the current transformer in Figure P5.11, where a (large) primary current I_1 produces a magnetic flux $\phi_1 = N_1 I_1 / \mathcal{R}$ in the core against which a (small) secondary current I_2 is supposed to produce $\phi_2 = N_2 I_2 / \mathcal{R}$. In order to make $\phi_1 + \phi_2 = 0$, what should the relationship between the primary/secondary currents be? Find the number of windings N_1 for the primary coil of the current transformer, i.e. how many turns the power transmission line is wound around the core.

Note. The secondary coil of a current transformer must not be opened because the open-circuit voltage across the secondary coil may become dangerously high on the same principle as an abrupt interruption of an inductor current could result in an excessively high voltage across the inductor.

5.12 Continuity Rule on Inductor Currents and Flux Linkage Conservation for Coupled Coils

- (a) Consider Example 5.1(a), which solves the coupled coil circuit of Figure 5.8.1(a) with $i_{L_1}(0) = 5 \text{ A}$ and $i_{L_2}(0) = 0 \text{ A}$. According to the solution expressed by Equation (E5.1.2), the values of the two inductor currents at $t = 0^+$ are

$$\begin{bmatrix} i_1(0^+) \\ i_2(0^+) \end{bmatrix} \stackrel{(E5.1.2)}{=} \begin{bmatrix} e^{-t/10} [\cos(t/5) + 2 \sin(t/5)] \\ e^{-t/10} [-2 \cos(t/5) + \sin(t/5)] \end{bmatrix} \bigg|_{t=0^+} = \begin{bmatrix} 1 \\ -2 \end{bmatrix} \text{ A} \quad (\text{P5.12.1})$$

Noting that this result violates the continuity rule on inductor currents (discussed in Section 3.1.1), apply the principle of flux linkage conservation (summarized in Remark 3.1(2)) to plead for the case. How about the law of energy conservation?

Hint. The total flux linkage and magnetic field energy of a pair of coupled coils are

$$\lambda \stackrel{(5.6a,b), (5.7)}{=} L_1 i_1 \pm M(i_1 + i_2) + L_2 i_2 \quad (\text{P5.12.2})$$

$$W_M \stackrel{(5.12)}{=} \frac{1}{2} L_1 i_1^2 \pm M i_1 i_2 + \frac{1}{2} L_2 i_2^2 \quad (\text{P5.12.3})$$

where L_1 , L_2 , and M are the self-inductances of the two coils and their mutual inductance, respectively, and the sign of the mutual inductance terms is plus or minus depending on whether the flux induced by one coil current and that induced by the other coil current are additive or subtractive.

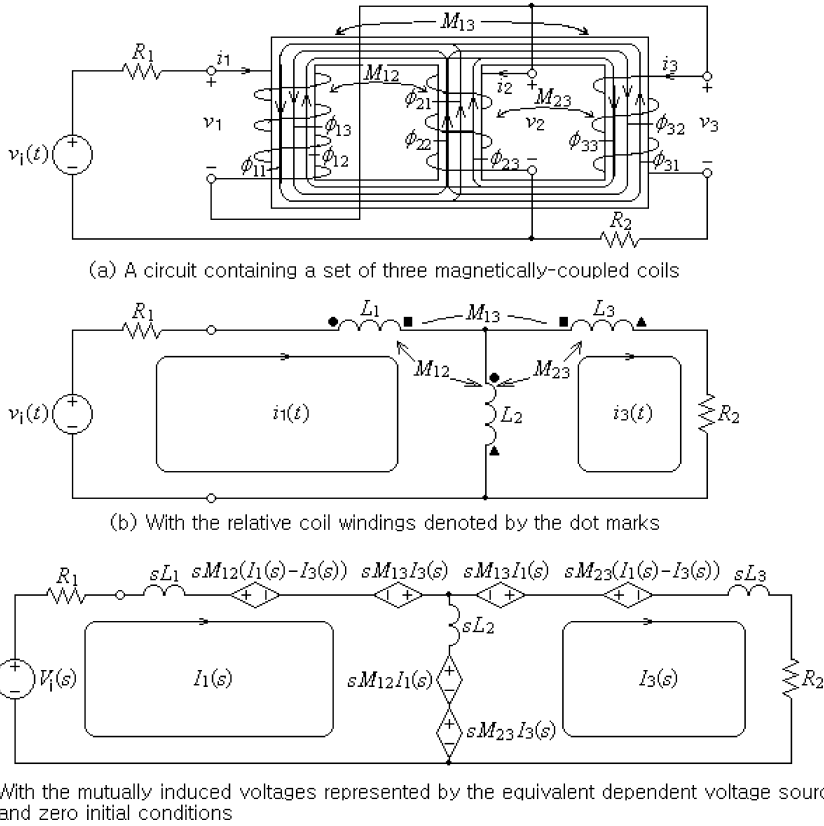


Figure P5.13

- (b) Consider Example 5.1(b), which solves the coupled coil circuit of Figure 5.8.1(b) with $i_{L_1}(0) = 0$ A and $i_{L_2}(0) = 0$ A. According to the solution expressed by Equation (E5.1.4), the values of the two inductor currents at $t = 0^+$ are

$$\begin{bmatrix} i_1(0^+) \\ i_2(0^+) \end{bmatrix} \stackrel{(E5.1.4)}{=} \begin{bmatrix} 10 - e^{-t/10} [2 \cos(t/5) + 4 \sin(t/5)] \\ e^{-t/10} [4 \cos(t/5) - 2 \sin(t/5)] \end{bmatrix} \bigg|_{t=0^+} = \begin{bmatrix} 8 \\ 4 \end{bmatrix} \text{ A} \quad (\text{P5.12.4})$$

Noting that this result violates the continuity rule on inductor currents, apply the principle of flux linkage conservation to plead for the case. How about the law of energy conservation?

5.13 A Circuit Containing a Set of Three Magnetically Coupled Coils

Consider the circuit of Figure P5.13(a), which contains a set of three magnetically coupled coils.

- With the reference directions of the coil currents denoted by the arrows, the mutually induced fluxes and voltages between the (left) coil 1 and the (middle) coil 2 are additive and those between the coil 2 and the (right) coil 3 are also additive. Are those between the two coils 1 and 3 additive or subtractive?
- The relative winding directions of the three coupled coils can be denoted by the dot marks as depicted in Figure P5.13(b). Thus the mutually induced voltages among the three coupled

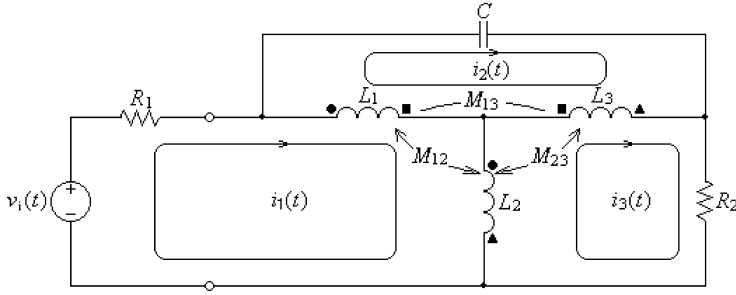


Figure P5.14 A circuit containing a set of three magnetically coupled coils with the relative coil windings denoted by the dot marks

coils can be represented by the dependent sources as shown in the transformed circuits of Figure P5.13(c). Write a set of mesh equations in the two mesh currents $I_1(s)$ and $I_3(s)$.

5.14 A Circuit Containing a Set of Three Magnetically Coupled Coils

Consider the circuit of Figure P5.14, which contains a set of three magnetically coupled coils. Write a set of mesh equations in the three mesh currents $I_1(s)$, $I_2(s)$, and $I_3(s)$.

6

AC Circuits

Having devoted most of the chapters prior to Chapter 5 on the transient response (corresponding to the analysis type of ‘Time domain’ in PSpice), attention is now directed to the AC (alternating current) steady state response (corresponding to the analysis type of ‘AC Sweep’ in PSpice), which is the steady state response of a circuit to a sinusoidal input (of a certain frequency) that survives after all transients have died out. Since the steady state output and the input source in an AC circuit are sinusoidal of the same frequency and with different amplitude/phase (Section 4.6), all the voltages/currents can be represented by complex variables called phasors and, accordingly, it is not the Laplace transform but the phasor transform that is conveniently used for the steady state analysis of AC circuits. As stated in the introductory part of the previous chapter, the electric power that is used every day at home, office, and factories is most efficiently/economically generated, transmitted, and distributed in AC form. Sinusoidal signals are also used in most communication systems as well as in radio/TV broadcasting systems. For these reasons, the AC analysis is very important in the practical aspect.

This chapter introduces the phasor representation (transform) for a sinusoidal function of voltage/current, the concept of AC impedance for passive elements, rms (root mean square) or effective value, instantaneous/active(or average or real)/reactive/complex/apparent power, power factor, and maximum power transfer or impedance matching.

Note. Visit the website (www.sciencejoywagon.com/physicszone/lesson/otherpub/wfendt/generatorengl.htm) to see how an AC voltage is generated.

6.1 Sinusoidal Sources

AC voltages/currents can be represented by cosine and/or sine functions. However, noting that cosine functions are equivalently converted into sine functions and vice versa, basically cosine functions are used to represent AC voltages/currents without loss of generality. Typical representations of an AC voltage and an AC current are

$$v(t) = V_m \cos(\omega t + \theta_v) = V_m \sin\left(\omega t + \theta_v + \frac{\pi}{2}\right) \quad (6.1a)$$

$$i(t) = I_m \cos(\omega t + \theta_i) = I_m \sin\left(\omega t + \theta_i + \frac{\pi}{2}\right) \quad (6.1b)$$

where V_m [V]/ I_m [A], ω [rad/s], and θ_v/θ_i [rad] are the amplitude (or peak or maximum value), the angular (radian) frequency, and the (initial) phase angle of the AC voltage/current, respectively. The angular frequency ω can be expressed in terms of the (ordinary/cyclical) frequency f [Hz] and the period $T = 1/f$ [s] as

$$\omega = 2\pi f = \frac{2\pi}{T} [\text{rad/s}] \quad (6.2)$$

To express the time-varying magnitudes of the AC voltage/current to the same scale as the constant magnitudes of DC voltage/current, the *rms* (root mean square) or *effective value* V/I of the AC voltage/current can be defined as the magnitude of the DC voltage/current that is equivalent in the sense that it can dissipate the same amount of power for a resistance R :

$$\int_T \frac{v^2(t)}{R} dt = \frac{V^2}{R} T, \quad V = \sqrt{\frac{1}{T} \int_T v^2(t) dt} \quad (6.3a)$$

$$\int_T i^2(t) R dt = I^2 RT, \quad I = \sqrt{\frac{1}{T} \int_T i^2(t) dt} \quad (6.3b)$$

These effective values might well be called the rms values because they are just the square roots of the means of the squared voltage/current. Substituting Equations (6.1a) and (6.1b) into Equations (6.3a) and (6.3b) yields the relations between the effective (or rms) values V/I and the peak values V_m/I_m of the sinusoidal voltage/current as

$$V = \sqrt{\frac{1}{T} \int_T V_m^2 \cos^2(\omega t + \theta_v) dt} \stackrel{(F.15)}{=} V_m \sqrt{\frac{1}{2T} \int_T [1 + \cos 2(\omega t + \theta_v)] dt} = \frac{V_m}{\sqrt{2}} \quad (6.4a)$$

$$I = \sqrt{\frac{1}{T} \int_T I_m^2 \cos^2(\omega t + \theta_i) dt} \stackrel{(F.15)}{=} I_m \sqrt{\frac{1}{2T} \int_T [1 + \cos 2(\omega t + \theta_i)] dt} = \frac{I_m}{\sqrt{2}} \quad (6.4b)$$

6.2 Phasor and AC Analysis

In Section 3.4.5, the Laplace transform method was used to solve an RL circuit and an RC circuit excited by a sinusoidal input source. Though much simpler than the conventional time-domain method based on the general/particular solution, it still requires somewhat intensive work. Here, it should be recalled that the steady state response of an LTI (linear time-invariant) circuit excited by a sinusoidal source of frequency ω is simply another sinusoid of the same frequency ω , which differs from the input source only in the magnitude and phase, as mentioned in Section 3.4.5. This gives the hope that an AC circuit can be solved without being involved in the frequency ω . As a substitute for the Laplace transform, the phasor transform comes up to this expectation. The phasor method of analyzing AC circuits is credited to Charles Proteus Steinmetz (1865–1923), a German-American electrical engineer, who was a pioneer in the field of electrical engineering.

The *phasor* and *rms phasor* of a sinusoidal function are complex numbers defined as follows:

(Maximum) phasor $\mathbf{X}_m$ (with the magnitude or modulus X_m and the phase or argument θ):

$$\begin{aligned} x(t) &= X_m \cos(\omega t + \theta) = \operatorname{Re}\{X_m e^{j(\omega t + \theta)}\} = \operatorname{Re}\{X_m e^{j\theta} e^{j\omega t}\} = \operatorname{Re}\{\mathbf{X}_m e^{j\omega t}\} \\ &\xLeftrightarrow[\text{inverse phasor transform}]{\text{phasor transform}} \mathbf{X}_m = \underset{\text{exponential form}}{X_m e^{j\theta}} = \underset{\text{rectangular form}}{X_m (\cos \theta + j \sin \theta)} = \underset{\text{polar form}}{X_m \angle \theta} \end{aligned} \quad (6.5a)$$

RMS phasor $\mathbf{X}$ (with the magnitude or modulus X and the phase or argument θ):

$$\begin{aligned} x(t) &= X_m \cos(\omega t + \theta) = \operatorname{Re}\{\sqrt{2} X e^{j(\omega t + \theta)}\} = \operatorname{Re}\{\sqrt{2} X e^{j\theta} e^{j\omega t}\} = \operatorname{Re}\{\sqrt{2} \mathbf{X} e^{j\omega t}\} \\ \leftrightarrow \mathbf{X} &= X e^{j\theta} = X (\cos \theta + j \sin \theta) = X \angle \theta \quad \text{with} \quad X = X_m / \sqrt{2} \end{aligned} \quad (6.5b)$$

Note. We will omit the subscript m from the phasor notation $\mathbf{X}_m$ if its meaning as the maximum phasor is obvious from the context and so cannot be confused with the rms phasor $\mathbf{X} = \mathbf{X}_m/\sqrt{2}$.

Based on this phasor definition, the sinusoidal voltage/current can be written as the real parts of the phasor voltage/current multiplied by the complex exponential function $e^{j\omega t}$ as

$$v(t) = V_m \cos(\omega t + \theta_v) = \operatorname{Re}\{V_m e^{j(\omega t + \theta_v)}\} = \operatorname{Re}\{V_m e^{j\theta_v} e^{j\omega t}\} = \operatorname{Re}\{\mathbf{V}_m e^{j\omega t}\} \leftrightarrow \mathbf{V}_m \quad (6.6a)$$

$$i(t) = I_m \cos(\omega t + \theta_i) = \operatorname{Re}\{I_m e^{j(\omega t + \theta_i)}\} = \operatorname{Re}\{I_m e^{j\theta_i} e^{j\omega t}\} = \operatorname{Re}\{\mathbf{I}_m e^{j\omega t}\} \leftrightarrow \mathbf{I}_m \quad (6.6b)$$

Since it does not matter whether they are originally given as a cosine function or a sine function, the phasor might have been defined based on the sine function instead of the cosine function.

It is noteworthy that the sum/difference of two sinusoids can easily be expressed and computed by using the phasor representation (transform) as follows:

$$\begin{aligned} v_1(t) \pm v_2(t) &= V_{1m} \cos(\omega t + \theta_1) \pm V_{2m} \cos(\omega t + \theta_2) \\ &\stackrel{(F.6)}{=} V_{1m} (\cos \omega t \cos \theta_1 - \sin \omega t \sin \theta_1) \pm V_{2m} (\cos \omega t \cos \theta_2 - \sin \omega t \sin \theta_2) \\ &= (V_{1m} \cos \theta_1 \pm V_{2m} \cos \theta_2) \cos \omega t - (V_{1m} \sin \theta_1 \pm V_{2m} \sin \theta_2) \sin \omega t \\ &= (V_{1m} \cos \theta_1 \pm V_{2m} \cos \theta_2) \cos \omega t + (V_{1m} \sin \theta_1 \pm V_{2m} \sin \theta_2) \cos\left(\omega t + \frac{\pi}{2}\right) \\ \mathbf{V}_{1m} \pm \mathbf{V}_{2m} &= V_{1m} e^{j\theta_1} \pm V_{2m} e^{j\theta_2} \\ &= V_{1m} (\cos \theta_1 + j \sin \theta_1) \pm V_{2m} (\cos \theta_2 + j \sin \theta_2) \\ &= (V_{1m} \cos \theta_1 \pm V_{2m} \cos \theta_2) + j(V_{1m} \sin \theta_1 \pm V_{2m} \sin \theta_2) \\ &= (V_{1m} \cos \theta_1 \pm V_{2m} \cos \theta_2) e^{j0} + (V_{1m} \sin \theta_1 \pm V_{2m} \sin \theta_2) e^{j\pi/2} \end{aligned}$$

It might be easier to understand the sum of two phasors by taking a look at Figure 6.1, where the two phasors and their sum are represented by two-dimensional vectors on a complex plane, together with the corresponding cosine/sine functions plotted along the ωt -axis.

Now, in order to see the connection between the phasor and the AC circuit analysis, consider an RL circuit excited by a sinusoidal voltage source $v_i(t) = V_m \cos(\omega t + \theta_v)$ (Figure 6.2(a)). KVL can be applied to obtain the mesh equation in the time domain as

$$v_R(t) + v_L(t) = v_i(t); \quad R i(t) + L \frac{di(t)}{dt} = V_m \cos(\omega t + \theta_v) \quad (6.7)$$

Noting that the steady state response $i(t)$ as well as the input source $v_i(t)$ is a sinusoid that can be expressed as the real part of a complex exponential function,

$$i(t) = \operatorname{Re}\{I_m e^{j(\omega t + \theta_i)}\} = \operatorname{Re}\{\mathbf{I}_m e^{j\theta_i}\} \quad \text{and} \quad v_i(t) = \operatorname{Re}\{V_m e^{j(\omega t + \theta_v)}\} = \operatorname{Re}\{\mathbf{V}_m e^{j\theta_v}\} \quad (6.8)$$

is substituted into the above equation (6.7) to rewrite it as

$$\begin{aligned} R \operatorname{Re}\{I_m e^{j(\omega t + \theta_i)}\} + L \frac{d}{dt} \operatorname{Re}\{I_m e^{j(\omega t + \theta_i)}\} &= \operatorname{Re}\{V_m e^{j(\omega t + \theta_v)}\} \\ \operatorname{Re}\{R I_m e^{j(\omega t + \theta_i)}\} + \operatorname{Re}\{j\omega L I_m e^{j(\omega t + \theta_i)}\} &= \operatorname{Re}\{V_m e^{j(\omega t + \theta_v)}\} \\ \operatorname{Re}\{R I_m e^{j\theta_i}\} + \operatorname{Re}\{j\omega L I_m e^{j\theta_i}\} &= \operatorname{Re}\{V_m e^{j\theta_v}\} \\ \text{with } \mathbf{I}_m &= I_m e^{j\theta_i} \quad \text{and} \quad \mathbf{V}_m = V_m e^{j\theta_v} \end{aligned} \quad (6.9)$$

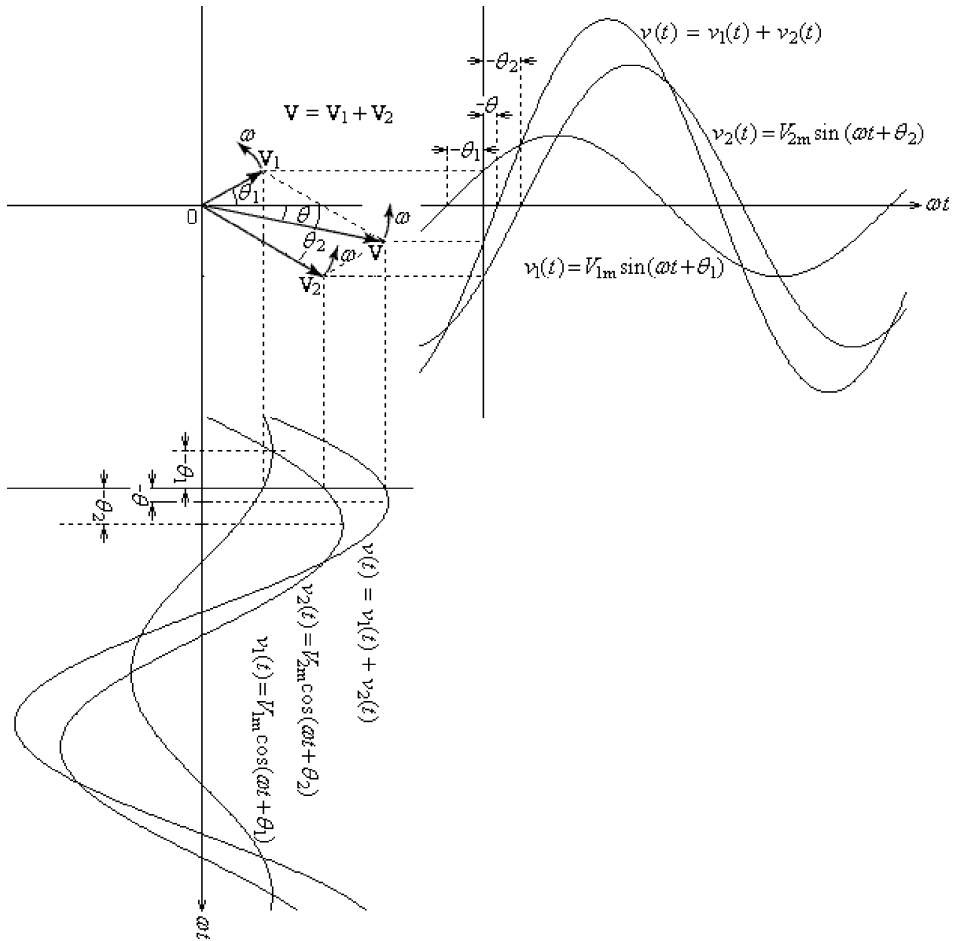


Figure 6.1 Phasor representation of the sum of two sinusoids

Here the fact has been used that the time derivative of the real part of $e^{j(\omega t + \theta)}$ equals the real part of the time derivative of $e^{j(\omega t + \theta)}$, i.e.

$$\begin{aligned} \frac{d}{dt} \operatorname{Re}\{e^{j(\omega t + \theta)}\} &= \frac{d}{dt} \cos(\omega t + \theta) \stackrel{(F.28)}{=} -\omega \sin(\omega t + \theta) \\ \operatorname{Re}\left\{\frac{d}{dt} e^{j(\omega t + \theta)}\right\} &\stackrel{(F.27)}{=} \operatorname{Re}\{j\omega e^{j(\omega t + \theta)}\} \stackrel{(F.20)}{=} \operatorname{Re}\{j\omega [\cos(\omega t + \theta) + j \sin(\omega t + \theta)]\} = -\omega \sin(\omega t + \theta) \end{aligned}$$

Thus we can use the definition of the phasor to rewrite Equation (6.9) as

$$R \mathbf{I}_m + j\omega L \mathbf{I}_m = \mathbf{V}_m \quad (6.10)$$

and solve it for the mesh current phasor as

$$\mathbf{I}_m = \frac{\mathbf{V}_m}{R + j\omega L}$$

$$I_m e^{j\theta_i} = \frac{V_m \angle \theta_v}{\sqrt{R^2 + (\omega L)^2} \angle \tan^{-1}(\omega L/R)} \stackrel{(C.3b)}{=} \frac{V_m}{\sqrt{R^2 + (\omega L)^2}} \angle \left(\theta_v - \tan^{-1} \frac{\omega L}{R} \right) \quad (6.11)$$

Finally the inverse phasor transform of this phasor solution is taken to get the time-domain solution as

$$i(t) = I_m \cos(\omega t + \theta_i) = \frac{V_m}{\sqrt{R^2 + (\omega L)^2}} \cos(\omega t + \theta_v - \theta) \quad \text{with} \quad \theta = \tan^{-1} \frac{\omega L}{R} \quad (6.12)$$

An important implication of this solution procedure is that it would be much simpler to put the phasor transform of the circuit itself into the form of Figure 6.2(b) so that the circuit equation can be directly set up as in Equation (6.10), where the AC impedance of the inductance L is $j\omega L$. Applying this phasor method, the steady state solutions of AC circuits could be obtained in just a few lines, like Equations (6.10) to (6.12).

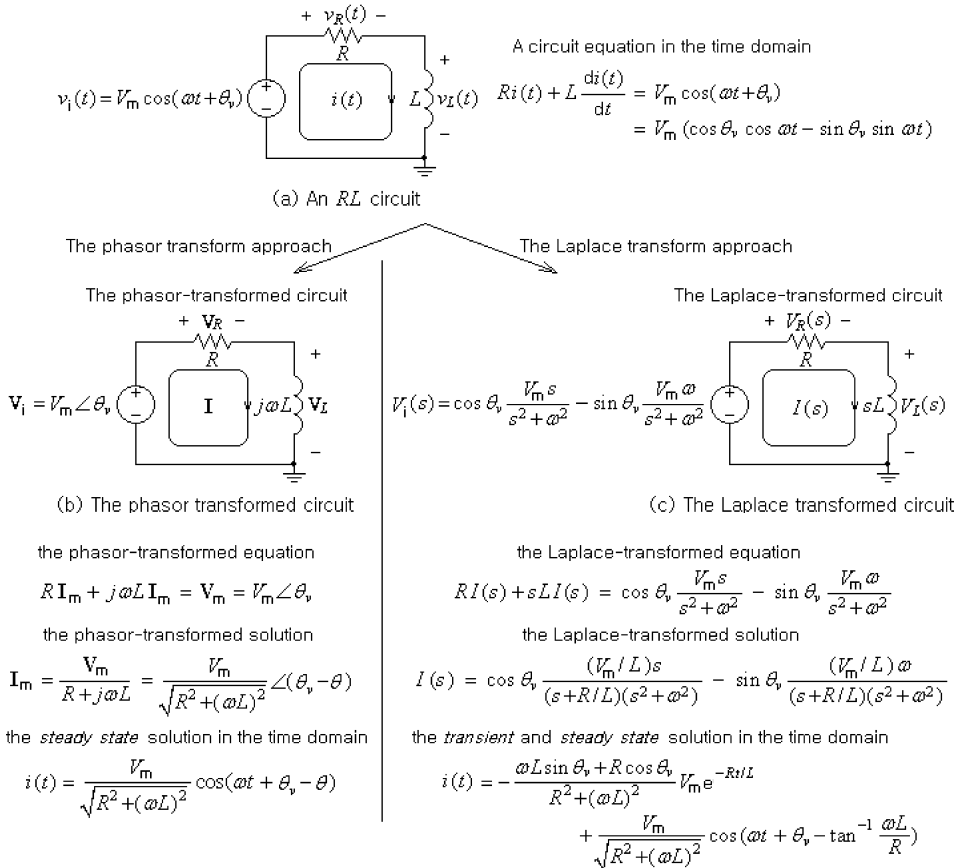


Figure 6.2 An RL circuit and the phasor transform approach versus the Laplace transform approach

This can be compared with the Laplace transform method explained in Section 3.4.5 and is a result of the good work of Mr. Charles Proteus Steinmetz. It might well be called the phasor transform as compared to the Laplace transform of circuits introduced in Section 3.3.2.

Under the impulse of curiosity, you might feel like applying the Laplace transform method to solve Equation (6.7). Let us fall into such an academic temptation. Taking the Laplace transform of the time-domain mesh equation (6.7) or applying KVL directly to the Laplace-transformed circuit in Figure 6.2(c) gives

$$\begin{aligned}
 Ri(t) + L \frac{di(t)}{dt} &= V_m \cos(\omega t + \theta_v) \stackrel{(F.6)}{=} V_m (\cos \theta_v \cos \omega t - \sin \theta_v \sin \omega t) \\
 RI(s) + sLI(s) &\stackrel{\text{Table A.1(8),(7)}}{=} \cos \theta_v \frac{V_m s}{s^2 + \omega^2} - \sin \theta_v \frac{V_m \omega}{s^2 + \omega^2} \\
 I(s) &= \cos \theta_v \frac{(V_m/L)s}{(s + R/L)(s^2 + \omega^2)} - \sin \theta_v \frac{(V_m/L)\omega}{(s + R/L)(s^2 + \omega^2)} \quad (6.13)
 \end{aligned}$$

Even with no initial conditions, this requires us to work twice as much as for Equation (3.49), seducing us into using the symbolic computing function of MATLAB as follows:

```
%cir06_02.m
clear
syms t s theta Vm w R L
Vs= laplace(Vm*cos(w*t+theta)); Is=Vs/(R+s*L) % Eq. (6.13)
it= ilaplace(Is), pretty(simple(it))
```

```
>>cir06_02
```

$$\begin{aligned}
 &\left(-V_m \exp\left(-\frac{Rt}{L}\right) \sin(\theta_v) wL - V_m \exp\left(-\frac{Rt}{L}\right) \cos(\theta_v) R \right. \\
 &\quad \left. + V_m R \cos(\omega t + \theta_v) + V_m L \omega \sin(\omega t + \theta_v) \right) / (R^2 + L^2 \omega^2)
 \end{aligned}$$

This result of running the MATLAB program `cir06_02.m` shows that the solution of Equation (6.7) is

$$\begin{aligned}
 i(t) &= \mathcal{L}^{-1}\{I(s)\} \\
 &= -\frac{\omega L \sin \theta_v + R \cos \theta_v}{R^2 + (\omega L)^2} V_m e^{-Rt/L} + \frac{V_m}{R^2 + (\omega L)^2} [R \cos(\omega t + \theta_v) + \omega L \sin(\omega t + \theta_v)] \\
 &= -\frac{\omega L \sin \theta_v + R \cos \theta_v}{R^2 + (\omega L)^2} V_m e^{-Rt/L} + \frac{V_m}{\sqrt{R^2 + (\omega L)^2}} \cos\left(\omega t + \theta_v - \tan^{-1} \frac{\omega L}{R}\right) \quad (6.14)
 \end{aligned}$$

The steady state part of this solution obtained by using the Laplace transform is identical to Equation (6.12), which was obtained by using the phasor transform. This convinces us that the phasor method suits the steady state solution of AC circuits better than the Laplace transform method. Remark 6.1 compares the phasor transform and the Laplace transform for circuit analysis.

[Remark 6.1] Phasor Transform versus Laplace Transform

1. As illustrated in Section 3.4, the Laplace transform method is quite good at finding the transient and steady state solutions for the circuits excited by DC sources and/or AC sources if only the Laplace transform can be taken of the source functions. Assuming that every source is applied for $t \geq 0$, this

approach treats the initial conditions (i.e. the initial values of capacitor voltages and inductor currents) no differently from the sources.

2. The phasor transform method presents only the steady state solutions for the circuits excited by AC sources, requiring much less effort than the Laplace transform method. This approach assumes that every (sinusoidal) source has been applied since $t = -\infty$ so that the circuit has already reached the steady state. It is not suitable for finding the response of a circuit attributable to DC input sources and initial conditions that cannot be represented by a phasor.

Note. The initial conditions yield only the transient responses that cannot be represented, far from being obtained, by using the phasor transform.

Note. The phasor transform converts a sinusoidal function of ωt into a complex constant carrying only its magnitude and phase so that the term depending on ωt can be put aside until it is needed.

6.3 AC Impedance of Passive Elements

In this section we establish the phasor voltage–current relationship of the passive elements, i.e. the resistor, inductor, and capacitor, where the passive sign convention is respected in all the derivations. For any single two-terminal element or a combination of more than one element in AC circuits excited by a sinusoidal source of a certain frequency, its AC or frequency domain impedance is defined as the ratio of the phasor voltage to the phasor current. The *AC impedance* is also a complex quantity just like the phasor, but should never be referred to as a phasor. Once the AC impedances for the passive elements are established, we can take the phasor transform of linear AC circuits (containing L and/or C) and deal with them as if they consisted of only resistors. It is similar to dealing with the linear circuits containing L and/or C in the same way as the resistor circuits once the Laplace transform of the circuits are taken, as introduced in Sections 3.3 and 3.4.

6.3.1 Resistor

Let the current through a resistor R be

$$i(t) = I_m \cos(\omega t + \theta_i) \leftrightarrow \mathbf{I}_m = I_m e^{j\theta_i}$$

Ohm's law (1.6a) states that the voltage across R is

$$v(t) = R i(t) = R I_m \cos(\omega t + \theta_i) \leftrightarrow \mathbf{V}_m = V_m e^{j\theta_v} = R I_m e^{j\theta_i} = R \mathbf{I}_m \quad (\theta_v = \theta_i)$$

This implies that the AC impedance and admittance of a resistor R defined as its phasor voltage-to-current and current-to-voltage ratios are

$$Z_R = \frac{\mathbf{V}_m}{\mathbf{I}_m} = \frac{\mathbf{V}}{\mathbf{I}} = R \quad \text{and} \quad Y_R = \frac{\mathbf{I}_m}{\mathbf{V}_m} = \frac{\mathbf{I}}{\mathbf{V}} = \frac{1}{R} = G \quad (6.15)$$

respectively.

6.3.2 Inductor

Let the current through an inductor L be

$$i(t) = I_m \cos(\omega t + \theta_i) \leftrightarrow \mathbf{I}_m = I_m e^{j\theta_i}$$

Then, the time-domain voltage–current relation (3.1a) of the inductor states that the voltage across L is

$$\begin{aligned} v(t) &\stackrel{(3.1a)}{=} L \frac{di(t)}{dt} \stackrel{(F.28)}{=} -\omega L I_m \sin(\omega t + \theta_i) \stackrel{(F.2)}{=} \omega L I_m \cos\left(\omega t + \theta_i + \frac{\pi}{2}\right) \\ &\leftrightarrow \mathbf{V}_m = V_m e^{j\theta_v} = \omega L I_m e^{j(\theta_i + \pi/2)} = e^{j\pi/2} \omega L I_m e^{j\theta_i} = j\omega L \mathbf{I}_m \quad \left(\theta_v = \theta_i + \frac{\pi}{2}\right) \end{aligned}$$

This implies that the AC impedance and admittance of an inductor L are

$$Z_L = \frac{\mathbf{V}_m}{\mathbf{I}_m} = \frac{\mathbf{V}}{\mathbf{I}} = j\omega L \quad \text{and} \quad Y_L = \frac{\mathbf{I}_m}{\mathbf{V}_m} = \frac{\mathbf{I}}{\mathbf{V}} = \frac{1}{j\omega L} \quad (6.16)$$

respectively.

6.3.3 Capacitor

Let the voltage across a capacitor C be

$$v(t) = V_m \cos(\omega t + \theta_v) \leftrightarrow \mathbf{V}_m = V_m e^{j\theta_v}$$

Then, the time-domain voltage–current relation (3.4a) of the capacitor states that the voltage across C is

$$\begin{aligned} i(t) &\stackrel{(3.2a)}{=} C \frac{dv(t)}{dt} \stackrel{(F.28)}{=} -\omega C V_m \sin(\omega t + \theta_v) \stackrel{(F.2)}{=} \omega C V_m \cos\left(\omega t + \theta_v + \frac{\pi}{2}\right) \\ &\leftrightarrow \mathbf{I}_m = I_m e^{j\theta_i} = \omega C V_m e^{j(\theta_v + \pi/2)} = e^{j\pi/2} \omega C V_m e^{j\theta_v} = j\omega C \mathbf{V}_m \quad \left(\theta_i = \theta_v + \frac{\pi}{2}\right) \end{aligned}$$

This implies that the AC impedance and admittance of a capacitor C are

$$Z_C = \frac{\mathbf{V}_m}{\mathbf{I}_m} = \frac{\mathbf{V}}{\mathbf{I}} = \frac{1}{j\omega C} \quad \text{and} \quad Y_C = \frac{\mathbf{I}_m}{\mathbf{V}_m} = \frac{\mathbf{I}}{\mathbf{V}} = j\omega C \quad (6.17)$$

respectively.

[Remark 6.2] Impedance/Admittance, Reactance/Susceptance, and Frequency Response

1. As the generalized (extended) concepts of the resistance/conductance, the AC (frequency domain) *impedances/admittances* of a resistor R , an inductor L , and a capacitor C are defined to be their phasor voltage-to-current/current-to-voltage ratios as

$$\begin{aligned} Z_R(j\omega) &= R, & Z_L(j\omega) &= j\omega L, & Z_C(j\omega) &= \frac{1}{j\omega C} \\ Y_R(j\omega) &= \frac{1}{R} = G, & Y_L(j\omega) &= \frac{1}{j\omega L}, & \text{and} & Y_C(j\omega) &= j\omega C \end{aligned}$$

respectively, where ω is the radian frequency of the sinusoidal input source applied to the AC circuit. Note that the AC impedances can be obtained by substituting $s = j\omega$ into their s -domain impedances R , sL , and $1/(sC)$ (see Section 3.3.2).

2. The imaginary part of the impedance/admittance is called the *reactance/susceptance*. If the sign of the reactance/susceptance is positive/negative, it is inductive; otherwise, i.e. if the sign of the reactance/susceptance is negative/positive, it is capacitive.

3. With the AC impedances/admittances, we can take the phasor transform of any linear AC circuits and deal with them as if they consisted of only resistors. It is as if the linear circuits could be dealt with in the same way as the resistor circuits once the Laplace transform of the circuits with the s -domain impedances are taken, as introduced in Sections 3.3 and 3.4.
4. As introduced in Section 4.6, $s = j\omega$ can be substituted in the *transfer function* of a system to obtain the *frequency response*, which is defined as the ratio of the phasor output to the phasor input as a function of the frequency ω . The frequency response will often be used in Chapter 8 when discussing filters.

6.4 AC Circuit Examples

Consider the series RLC circuit of Figure 6.3(a). The phasor mesh current of the phasor-transformed circuit is obtained as

$$\mathbf{I} = \frac{\mathbf{V}_i}{\mathbf{Z}} = \frac{\mathbf{V}_i}{R + j\omega L + 1/(j\omega C)} = \frac{\mathbf{V}_i}{R + j[\omega L - 1/(\omega C)]} \quad (6.18a)$$

$$I \angle \theta_i = \frac{V \angle \theta_v}{|Z| \angle \theta} \stackrel{(C.3b)}{=} \frac{V}{|Z|} \angle (\theta_v - \theta) \quad (6.18b)$$

where

$$\text{Impedance: } Z = R + jX = R + j\left(\omega L - \frac{1}{\omega C}\right) \quad (6.19a)$$

$$\text{Magnitude of impedance: } |Z| \stackrel{(C.4)}{=} \sqrt{R^2 + X^2} = \sqrt{R^2 + \left(\omega L - \frac{1}{\omega C}\right)^2} \quad (6.19b)$$

$$\text{Phase angle of impedance: } \theta \stackrel{(C.4)}{=} \tan^{-1} \frac{X}{R} = \tan^{-1} \frac{\omega L - 1/(\omega C)}{R} = \theta_v - \theta_i \quad (6.19c)$$

(The phase difference between the voltage and the current)

$$\text{Reactance: } X = \omega L - \frac{1}{\omega C} \quad (\text{the imaginary part of the impedance}) \quad (6.19d)$$

Note. The phase angle of a load impedance equals the phase difference between the voltage and the current of the load: $\theta = \theta_v - \theta_i$.

Note. I means the maximum/rms value of the current if V means the maximum/rms value of the voltage.

The *impedance triangle* in Figure 6.3(b) is a graphical representation of an impedance on the complex plane. Since the phase angle of a load impedance equals the phase difference between the voltage and the current of the load, i.e. $\theta = \theta_v - \theta_i$, the following points are implied:

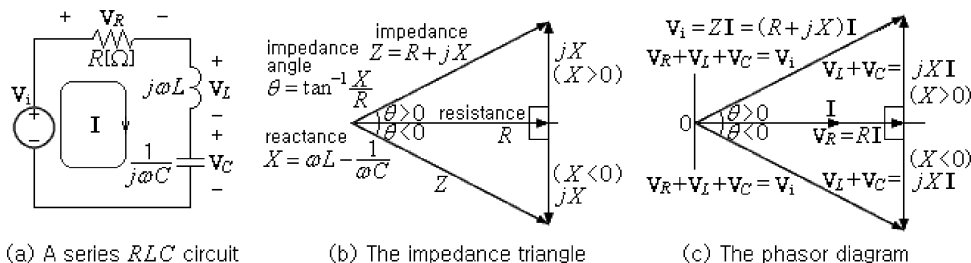


Figure 6.3 The impedance triangle and phasor diagram for a series RLC circuit

1. The *inductive* circuit has

$$X > 0; \omega L \text{ (the magnitude of inductive reactance)} > \frac{1}{\omega C} \text{ (the magnitude of capacitive reactance)}$$

and therefore has the positive impedance phase angle $\theta = \theta_v - \theta_i > 0$ so that the current lags the voltage, i.e. $\theta_i < \theta_v$ (phase lag).

2. The *capacitive* circuit has

$$X < 0; \quad \omega L < \frac{1}{\omega C}$$

and therefore has the negative impedance phase angle $\theta = \theta_v - \theta_i < 0$ so that the current leads the voltage, i.e. $\theta_i > \theta_v$ (phase lead).

3. The purely *resistive* circuit has

$$X = 0; \quad \omega L = \frac{1}{\omega C}$$

and therefore has the zero impedance phase angle $\theta = \theta_v - \theta_i = 0$ so that the current and the voltage have the same phase $\theta_i = \theta_v$ and are said to be 'in phase'.

Note. By contrast, the current and voltage with different phases $\theta_i \neq \theta_v$ are said to be 'out of phase'.

The phasor diagram in Figure 6.3(c) shows the phasor mesh current $\mathbf{I}$, the phasor voltages $\mathbf{V}_i$, $\mathbf{V}_R$, and $(\mathbf{V}_L + \mathbf{V}_C)$ as well as their relationship on the complex plane.

Now consider the parallel RLC circuit of Figure 6.4(a). The overall phasor current of the phasor-transformed circuit is obtained as

$$\mathbf{I} = Y\mathbf{V}_i = \left[\frac{1}{R} + j \left(\omega C - \frac{1}{\omega L} \right) \right] \mathbf{V}_i \quad (6.20a)$$

$$I \angle \theta_i = |Y| \angle \theta \times V_i \angle \theta_v \stackrel{(C.2b)}{=} |Y| V_i \angle (\theta_v + \theta) \quad (6.20b)$$

where

$$\text{Admittance: } Y = G + jB = \frac{1}{R} + j \left(\omega C - \frac{1}{\omega L} \right) \quad (6.21a)$$

$$\text{Magnitude of admittance: } |Y| \stackrel{(C.4)}{=} \sqrt{G^2 + B^2} = \sqrt{\frac{1}{R^2} + \left(\omega C - \frac{1}{\omega L} \right)^2} \quad (6.21b)$$

$$\text{Phase angle of admittance: } \theta \stackrel{(C.4)}{=} \tan^{-1} \frac{B}{G} = \tan^{-1} \frac{\omega C - 1/(\omega L)}{G} = \theta_i - \theta_v \quad (6.21c)$$

(The phase difference between the current and the voltage)

$$\text{Susceptance: } B = \omega C - \frac{1}{\omega L} \quad (\text{the imaginary part of the admittance}) \quad (6.21d)$$

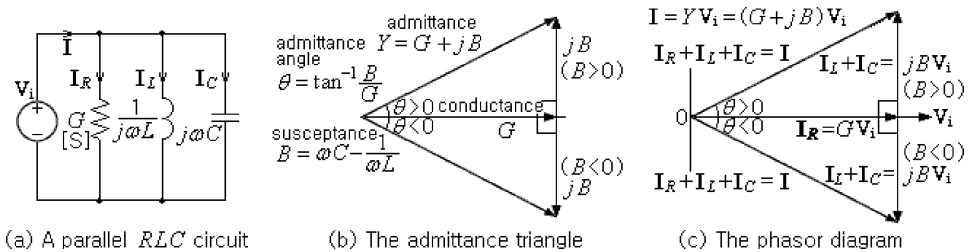


Figure 6.4 The admittance triangle and phasor diagram for a parallel RLC circuit

Note. The phase angle of a load admittance equals the phase difference between the current and the voltage of the load: $\theta = \theta_i - \theta_v$.

The *admittance triangle* in Figure 6.4(b) is a graphical representation of an admittance on the complex plane. Since the phase angle of a load admittance equals the phase difference between the current and the voltage of the load, i.e. $\theta = \theta_i - \theta_v$, the following points are implied:

1. The *capacitive* circuit has

$$B > 0; \quad \omega C (\text{the magnitude of capacitive susceptance}) > \frac{1}{\omega L} (\text{the magnitude of inductive susceptance})$$

and therefore has the positive admittance phase angle $\theta = \theta_i - \theta_v > 0$ so that the current leads the voltage, i.e. $\theta_i > \theta_v$ (phase lead).

2. The *inductive* circuit has

$$B < 0; \quad \omega C < \frac{1}{\omega L}$$

and therefore has the negative admittance phase angle $\theta = \theta_i - \theta_v < 0$ so that the current lags the voltage, i.e. $\theta_i < \theta_v$ (phase lag)

3. The purely *resistive* circuit has

$$B = 0; \quad \omega C = \frac{1}{\omega L}$$

and therefore has the zero admittance phase angle $\theta = \theta_i - \theta_v = 0$ so that the current and the voltage have the same phase $\theta_i = \theta_v$ and are said to be ‘in phase’.

The phasor diagram in Figure 6.4(c) shows the voltage source $\mathbf{V}_i$, the overall phasor current $\mathbf{I}$, the phasor currents $\mathbf{I}_R$, and $(\mathbf{I}_L + \mathbf{I}_C)$ as well as their relationship on the complex plane.

[Remark 6.3] Impedance Triangle and Impedance (Phase) Angle (Power Factor Angle)

The impedance (phase) angle θ defined by Equation (6.19c) and denoted as an acute internal angle of the impedance triangle in Figure 6.3(b) equals the phase difference between the voltage and the current of the load having such an impedance. As will be explained in Section 6.6, it determines the power factor (PF) of the load and so is called the PF angle. The real part of the impedance of a passive element or a circuit consisting of passive elements is its resistance, which is positive, and, accordingly, its impedance (phase) angle θ is between $-\pi/2$ (-90°) and $+\pi/2$ ($+90^\circ$).

Here comes a question. What is the physical meaning of the phase difference θ between the voltage and the current? The answer can be found in Figure 6.5, which shows the current waveforms $i_1(t)$, $i_2(t)$, and $i_3(t)$ of three loads excited by the same voltage $v(t)$, each with $\theta = \theta_v - \theta_i = \pi/6$ (phase lag), $\theta = -\pi/4$ (phase lead), and $\theta = 0$ (in phase). It might be said that the *phase lag* and *phase lead* of θ mean that the current lags behind and leads the voltage by θ on the ωt -axis; in other words, the current reaches its peak/valley θ/ω seconds later and earlier than the voltage reaches its peak/valley along the t axis, respectively:

$$v(t) = V_m \cos(\omega t + \theta_v)$$

$$i(t) = I_m \cos(\omega t + \theta_i) = I_m \cos(\omega t + \theta_v - \theta) = I_m \cos[\omega(t - \theta/\omega) + \theta_v]$$

Now that the AC impedances for the passive elements R , L , and C have been established, all the analysis methods and the related techniques developed for the resistor circuits will be applied in order to deal with the AC circuits containing inductors and/or capacitors as well as

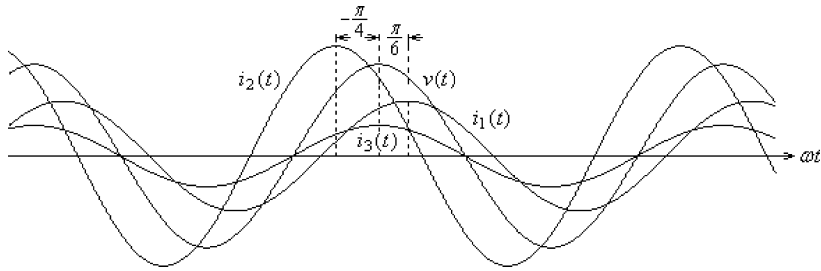


Figure 6.5 The phase difference between the voltages and the currents of the three loads with the impedance phase angle of $\theta = \pi/6$ (phase lag), $-\pi/4$ (phase lead), and 0 (in phase)

resistors. They include the voltage-to-current/current-to-voltage source transformations, the series/parallel combinations of impedances, the Δ -Y/Y- Δ conversions, the mesh/node analyses, Thevenin/Norton equivalent circuits, etc. For instance, the Δ -Y/Y- Δ conversion formulas introduced in Section 2.3 are listed in Table 6.1 and cast into the MATLAB routines `dy_conversion()`/`yd_conversion()`, which will soon be used. Before looking at the following example, it would be good to save the M-files containing the routines together with another M-file `parallel_comb.m` (listed below) in some directory of your computer that can be searched by MATLAB.

```
function[Za,Zb,Zc]=dy_conversion(Zab,Zbc,Zca)
temp = Zab+Zbc+Zca;
Za=Zca*Zab/temp; Zb=Zab*Zbc/temp; Zc=Zbc*Zca/temp; % Eq.s (6.22a-c)

function[Zab,Zbc,Zca]=yd_conversion(Za,Zb,Zc)
temp = Za*Zb+Zb*Zc+Zc*Za;
Zab=temp/Zc; Zbc=temp/Za; Zca=temp/Zb; % Eq.s (6.23a-c)

function Zp=parallel_comb(Zs)
Zp = 1/sum(1./Zs); % The reciprocal of Eq. (2.2)
```

Table 6.1 Δ -Y and Y- Δ conversion formulas

Δ -Y conversion formulas	Y- Δ conversion formulas
$Z_a = \frac{Z_{ca}Z_{ab}}{Z_{ab} + Z_{bc} + Z_{ca}} \quad (6.22a)$	$Z_{ab} = \frac{Z_aZ_b + Z_bZ_c + Z_cZ_a}{Z_c} \quad (6.23a)$
$Z_b = \frac{Z_{ab}Z_{bc}}{Z_{ab} + Z_{bc} + Z_{ca}} \quad (6.22b)$	$Z_{bc} = \frac{Z_aZ_b + Z_bZ_c + Z_cZ_a}{Z_a} \quad (6.23b)$
$Z_c = \frac{Z_{bc}Z_{ca}}{Z_{ab} + Z_{bc} + Z_{ca}} \quad (6.22c)$	$Z_{ca} = \frac{Z_aZ_b + Z_bZ_c + Z_cZ_a}{Z_b} \quad (6.23c)$

(Example 6.1) A Bridge Circuit

The current through the 40Ω -resistor in the bridge circuit of Figure 6.6(a) will be found by using the four different methods, i.e. the mesh analysis, the Δ -Y conversion, the Thevenin equivalent, and the node analysis. Besides, the PSpice simulation will be performed to see the current waveform.

- (a) To apply the mesh analysis, the mesh equation is set up in the three mesh currents $\mathbf{I}_1$, $\mathbf{I}_{ac}$, and $\mathbf{I}_{cd}$ and solved as follows:

$$\begin{bmatrix} 40 - j80 & j100 & -40 - j20 \\ j100 & 94 - j82 & -40 \\ -40 - j20 & -40 & 80 - j160 \end{bmatrix} \begin{bmatrix} \mathbf{I}_1 \\ \mathbf{I}_{ac} \\ \mathbf{I}_{cd} \end{bmatrix} = \begin{bmatrix} 450 \\ 0 \\ 0 \end{bmatrix} \quad (\text{E6.1.1})$$

$$\begin{bmatrix} \mathbf{I}_1 \\ \mathbf{I}_{ac} \\ \mathbf{I}_{cd} \end{bmatrix} = \begin{bmatrix} 4 + j2 \\ 3 - j \\ j1.5 \end{bmatrix}; \quad \mathbf{I}_{bc} = \mathbf{I}_{cd} - \mathbf{I}_{ac} = j1.5 - (3 - j) = -3 + j2.5 \quad (\text{E6.1.2})$$

This result can be obtained by typing the following statements into the MATLAB command window:

```
>> Z=[ 40-80i 100i -40-20i; 100i 94-82i -40; -40-20i -40 80-160i]
Z = 1.0e002 * 0.4000 - 0.8000i 0 + 1.0000i - 0.4000 - 0.2000i
0 + 1.0000i 0.9400 - 0.8200i - 0.4000
-0.4000 - 0.2000i - 0.4000 0.8000 - 1.6000i
>> I=Z^-1*[ 450 0 0] .' % I=Z\ [ 450 0 0] .' %.' for transpose
I = 4.0000 + 2.0000i
3.0000 - 1.0000i
0.0000 + 1.5000i
>> I(3) - I(2) % Eq. (E6.1.2)
ans = -3.0000 + 2.5000i
```

- (b) The Δ -Y conversion is applied for the lower ∇ part of the circuit in Figure 6.6(a) to get an equivalent, as depicted in Figure 6.6(b):

$$Z_b = \frac{Z_{db}Z_{bc}}{Z_{bc} + Z_{cd} + Z_{db}} = \frac{(40 + j20) \times 40}{40 - j180 + (40 + j20)} = \frac{800(2 + j)}{80(1 - j2)} = j10 \quad (\text{E6.1.3})$$

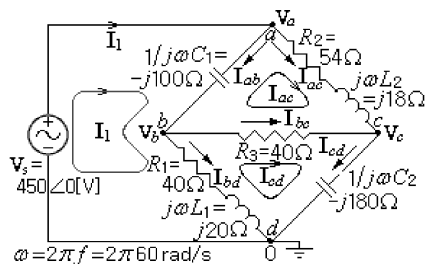
$$Z_c = \frac{Z_{bc}Z_{cd}}{Z_{bc} + Z_{cd} + Z_{db}} = \frac{40 \times (-j180)}{80(1 - j2)} = \frac{-j90}{1 - j2} = 36 - j18 \quad (\text{E6.1.4})$$

$$Z_d = \frac{Z_{cd}Z_{db}}{Z_{bc} + Z_{cd} + Z_{db}} = \frac{(-j180) \times (40 + j20)}{80(1 - j2)} = \frac{45(1 - j2)}{1 - j2} = 45 \quad (\text{E6.1.5})$$

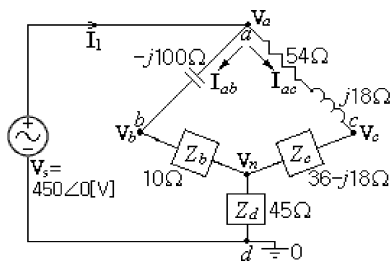
Then we can apply the series/parallel combination formula of impedances to get the (total) impedance seen from terminals a - d and find the (total) current $\mathbf{I}_1$ as

$$\begin{aligned} Z_{ad} &= [(Z_{ab} + Z_b) \parallel (Z_{ac} + Z_c)] + Z_d \\ &= (-j100 + j10) \parallel [(54 + j18) + (36 - j18)] + 45 \\ &= \frac{(-j90) \times 90}{(-j90) + 90} + 45 = 45(1 - j) + 45 = 90 - j45 \end{aligned} \quad (\text{E6.1.6})$$

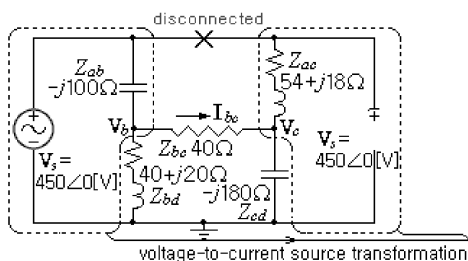
$$\mathbf{I}_1 = \frac{\mathbf{V}_s}{Z_{ad}} = \frac{450}{45(2 - j)} = 4 + j2 \quad (\text{E6.1.7})$$



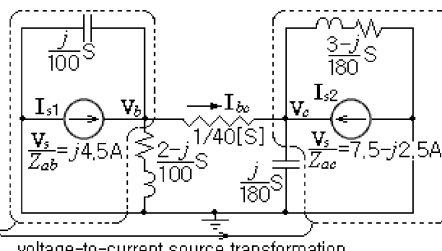
(a) A bridge circuit



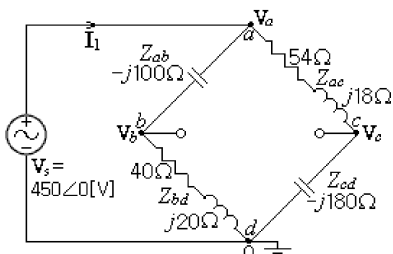
(b) The equivalent circuit with Δ -Y conversion



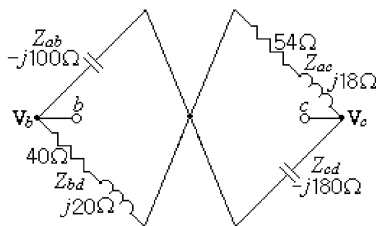
(c1) With duplicated voltage sources for the voltage-to-current source transformation



(c2) With voltage-to-current source transformation



(d1) With the terminals b - c opened to find the Thevenin equivalent voltage source



(d2) With the independent voltage source shorted to find the Thevenin equivalent impedance

$$\begin{aligned} V_{Th} &= V_b - V_c \\ &= \left(\frac{40 + j20}{-j100 + (40 + j20)} - \frac{-j180}{(54 + j18) - j180} \right) 450 \\ &= -450 + j375 \end{aligned}$$

$$Z_{Th} = (-j100) \parallel (40 + j20) + (54 + j18) \parallel (-j180) = 110$$

(d3) The Thevenin equivalent circuit (at terminals b - c) connected to the 40Ω -load

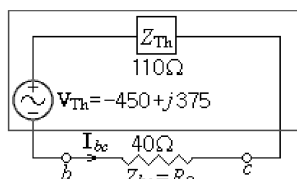


Figure 6.6 The circuit diagrams for Example 6.1

Successively, the current divider rule is used to get I_{ab} and I_{ac} as

$$I_{ab} = \frac{Z_{acn}}{Z_{abn} + Z_{acn}} I_1 = \frac{90}{90 - j90} (4 + j2) = \frac{4 + j2}{1 - j} = 1 + j3 \quad (E6.1.8)$$

$$I_{ac} = \frac{Z_{abn}}{Z_{abn} + Z_{acn}} I_1 = \frac{-j90}{90 - j90} (4 + j2) = \frac{-j(4 + j2)}{1 - j} = 3 - j \quad (E6.1.9)$$

and find the voltages $\mathbf{V}_b$ and $\mathbf{V}_c$ at the two nodes b and c as

$$\mathbf{V}_b = \mathbf{V}_s - Z_{ab}\mathbf{I}_{ab} = 450 - (-j100)(1 + j3) = 150 + j100 \quad (\text{E6.1.10})$$

$$\mathbf{V}_c = \mathbf{V}_s - Z_{ac}\mathbf{I}_{ac} = 450 - (54 + j18)(3 - j) = 270 \quad (\text{E6.1.11})$$

Finally, the phasor current through the $R_3 = 40\Omega$ resistor is found as

$$\mathbf{I}_{bc} = \frac{\mathbf{V}_b - \mathbf{V}_c}{Z_{bc} = R_3} = \frac{150 + j100 - 270}{40} = -3 + j2.5 \quad (\text{E6.1.12})$$

These results can be obtained by typing the following statements into the MATLAB command window:

```
>> Zab=-100i; Zac=54+18i; Zbc=40; Zbd=40+20i; Zcd=-180i;
>> [ Zb,Zc,Zd]=dy_conversion(Zbc,Zcd,Zbd) % Eq. (E6.1.3,4,5)
    Zb=    0+10.0000i    Zc= 36.0000 - 18.0000i    Zd=45
>> Zad=parallel_comb([ Zab+Zb  Zac+Zc ])+Zd % Eq. (E6.1.6)
    Zad= 90.0000 - 45.0000i
>> Vs=450; I1=Vs/Zad % Eq. (E6.1.7)
    I1= 4.0000 + 2.0000i
>> Zabn=Zab+Zb; Zacn=Zac+Zc;
>> Iab=Zacn/(Zabn+Zacn)*I1, Iac=Zabn/(Zabn+Zacn)*I1 % Eq. (E6.1.8,9)
    Iab= 1.0000 + 3.0000i
    Iac= 3.0000 - 1.0000i
>> Vb=Vs-Zab*Iab, Vc=Vs-Zac*Iac % Eq. (E6.1.10,11)
    Vb= 1.50000e+002 + 1.0000e+002i    Vc= 270
>> Ibc=(Vb-Vc)/Zbc % Eq. (E6.1.12)
    Ibc= -3.0000 + 2.5000i
```

- (c) To apply the node analysis, the voltage source is duplicated as depicted in Figure 6.6(c1), the two voltage sources are separated, and then each one is associated with the series element to transform them into current sources as depicted in Figure 6.6(c2). Then the node equation is set up in the two node voltages $\mathbf{V}_b$ and $\mathbf{V}_c$ and then solved as

$$\begin{bmatrix} \frac{j}{100} + \frac{2-j}{100} + \frac{1}{40} & -\frac{1}{40} \\ -\frac{1}{40} & \frac{3-j}{180} + \frac{j}{180} + \frac{1}{40} \end{bmatrix} \begin{bmatrix} \mathbf{V}_b \\ \mathbf{V}_c \end{bmatrix} = \begin{bmatrix} j4.5 \\ 7.5 - j2.5 \end{bmatrix} \quad (\text{E6.1.13})$$

$$\begin{bmatrix} 9 & -5 \\ -9 & 15 \end{bmatrix} \begin{bmatrix} \mathbf{V}_b \\ \mathbf{V}_c \end{bmatrix} = \begin{bmatrix} j900 \\ 2700 - j900 \end{bmatrix}$$

$$\begin{bmatrix} \mathbf{V}_b \\ \mathbf{V}_c \end{bmatrix} = \frac{1}{90} \begin{bmatrix} 15 & 5 \\ 9 & 9 \end{bmatrix} \begin{bmatrix} j900 \\ 2700 - j900 \end{bmatrix} = \begin{bmatrix} 150 + j100 \\ 270 \end{bmatrix} \quad (\text{E6.1.14})$$

$$\mathbf{I}_{bc} = \frac{\mathbf{V}_b - \mathbf{V}_c}{Z_{bc} = R_3} = \frac{150 + j100 - 270}{40} = -3 + j2.5 \quad (\text{E6.1.15})$$

This result can be obtained by typing the following statements into the MATLAB command window:

```
>> Y=[ 1/Zab+1/Zbd+1/Zbc -1/Zbc; -1/Zbc 1/Zac+1/Zcd+1/Zbc ];
>> V=Y\[ Vs/Zab; Vs/Zac] % Eq. (E6.1.14)
    V= 1.0e+002 * 1.5000 + 1.0000i
        2.7000 - 0.0000i
```

```
>> Ibc=(Vb-Vc)/Zbc % Eq. (E6.1.15)
Ibc = -3.0000 + 2.5000i
```

(d) To obtain the Thevenin equivalent seen at terminals b - c , the following steps are taken:

- Open the terminals b - c by disconnecting the load impedance R_3 as depicted in Figure 6.6(d1).
- As for the Thevenin equivalent voltage source, find the voltage difference between the terminals b - c :

$$\mathbf{V}_{Th} = \left(\frac{40 + j20}{-j100 + (40 + j20)} - \frac{-j180}{(54 + j18) - j180} \right) 450 = -450 + j375 \quad (\text{E6.1.16})$$

- As for the Thevenin equivalent impedance, remove the independent (voltage) source by short-circuiting it as depicted in Figure 6.6(d2) and find the equivalent impedance seen from terminals b - c by using the parallel-series combinations:

$$Z_{Th} = (-j100) \parallel (40 + j20) + (54 + j18) \parallel (-j180) = 110 \quad (\text{E6.1.17})$$

Now that we have the Thevenin equivalent together with the load impedance $R_3 = 40 \Omega$ as depicted in Figure 6.6(d3), the current through R_3 can be obtained as

$$\mathbf{I}_{bc} = \frac{\mathbf{V}_{Th}}{Z_{Th} + R_3} = \frac{-450 + j375}{110 + 40} = -3 + j2.5 \stackrel{(\text{C.4})}{=} \sqrt{(-3)^2 + 2.5^2} \angle \tan^{-1} \frac{2.5}{-3} = 3.9 \angle 140^\circ \quad (\text{E6.1.18})$$

Note that this result means that

$$i_{bc}(t) = \begin{cases} 3.9 \sin(2\pi 60t + 140^\circ) & \text{if } \mathbf{V}_s = 450 \angle 0^\circ \text{ means } v_s(t) = 450 \sin(2\pi 60t) \\ 3.9\sqrt{2} \cos(2\pi 60t + 140^\circ) & \text{if } \mathbf{V}_s = 450 \angle 0^\circ \text{ (rms phasor) means } v_s(t) = 450\sqrt{2} \cos(2\pi 60t) \end{cases}$$

Note: The above results imply that it does not matter whether the (maximum) phasor (defined by Equation (6.5a)) or the rms phasor (defined by Equation (6.5b)) is used and nor whether the cosine function or the sine function is adopted as the basic function of the phasor definition.

```
>> Vth=(Zbd/(Zab+Zbd)-Zcd/(Zac+Zcd))*Vs % Eq. (E6.1.16)
Vth = -4.5000e+002 + 3.7500e+002i
>> Zth=parallel_comb([Zab Zbd])+parallel_comb([Zac Zcd]) % Eq. (E6.1.17)
Zth = 1.1000e+002 -3.1225e-015i % 110
>> R3=40; Ibc=Vth/(Zth+R3) % Eq. (E6.1.18)
Ibc = -3.0000 + 2.5000i
>> Ibc_mag=abs(Ibc), Ibc_angle=angle(Ibc)*180/pi % magnitude & phase
Ibc_mag = 3.9051 Ibc_angle = 140.1944
```

(e) To perform the PSpice simulation, the schematic (Figure 6.7(a)) is drawn in the Schematic Editor window and the following steps are taken:

- Set the values of the passive elements as

$$R_1 = 40 \Omega, \quad R_2 = 54 \Omega, \quad R_3 = 40 \Omega$$

$$L_1 = \frac{X_{L1}}{\omega} = \frac{20}{2\pi 60} = 53.1 \text{ mH}, \quad L_2 = \frac{X_{L2}}{\omega} = \frac{18}{2\pi 60} = 47.7 \text{ mH}$$

$$C_1 = \frac{1/|X_{C1}|}{\omega} = \frac{1/100}{2\pi 60} = 26.526 \mu\text{F}, \quad C_2 = \frac{1/|X_{C2}|}{\omega} = \frac{1/180}{2\pi 60} = 14.737 \mu\text{F}$$

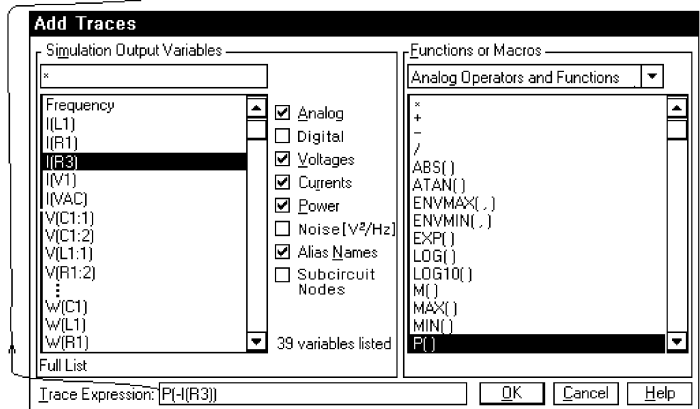
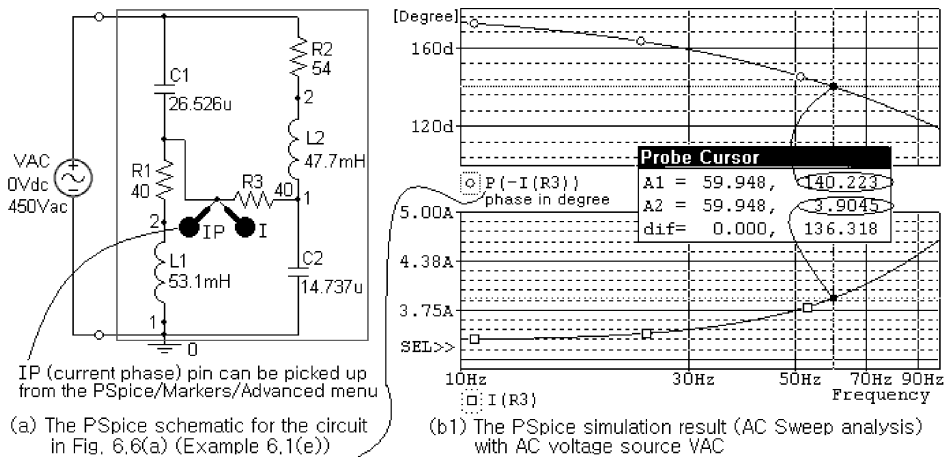
- In order to see $\mathbf{I}_{R_3}(\omega)$ in the frequency domain, set the values of the AC voltage source VAC as

$$\text{ACMAG} = 450, \quad \text{DC} = 0 \quad (\text{with ACPHASE} = 0 \text{ by default})$$

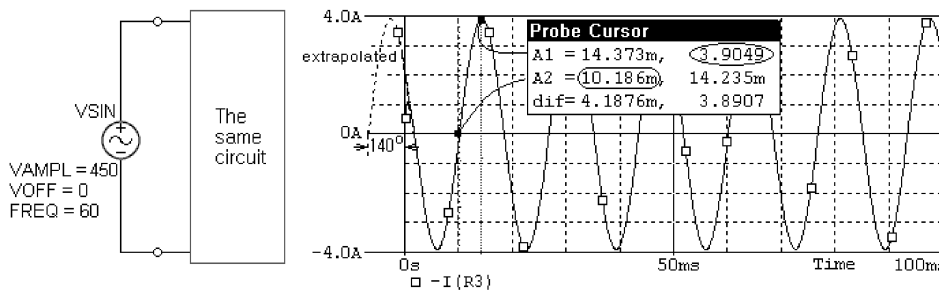
- Click the New Simulation Profile button (on the toolbar) and name it, say AC_sweep.
- In the Simulation Settings dialog box opened by clicking the Edit Simulation Settings button (on the toolbar), set the Analysis type to ‘AC Sweep/Noise’ and the AC Sweep type to ‘Logarithmic/Decade’. Also set the Start, End Frequency, and Points/Decade to 10, 100 Hz and 400 Hz, respectively, so that the source frequency of 60 Hz is contained in the frequency interval.
- Click the Current Marker button on the toolbar to place a current probe pin at a terminal of R_3 .
- Click the Run button (on the toolbar) and see the graph of $|\mathbf{I}_{R_3}|$ for $10 \leq f \leq 100$ Hz on the PSpice A/D (Probe) window (Figure 6.7(b1)).
- Click the Toggle Cursor button on the toolbar in the Probe window to call the cross-type cursor, use the left mouse pointer and/or the left/right arrow key to locate the cursor near the source frequency of 60 Hz and then read the value of $|\mathbf{I}_{R_3}|$ from the Probe Cursor box. Is it close to 3.9, which is analytically obtained in Equation (E6.1.18)?
- In order to get the phase of $\mathbf{I}_{R_3}$, click the PSpice/Markers/Advanced/Phase_of_Current menu to pick up a current phase (IP) probe pin and place it at a terminal of R_3 . Alternatively, click Plot/Add_Plot_to_Window on the menu bar of the Probe window and click the Add Trace button on the toolbar to open the Add Traces dialog box, in which you select ‘P()’ in the Functions or Macros box on the right-hand side and then select ‘I(R3)’ in the Simulation Output Variable box on the left-hand side so that ‘P(I(R3))’ will appear on the Trace Expression field at the bottom part. Optionally, put the minus sign before ‘I(R3)’, which is not indispensable. You might directly type ‘P(I(R3))’ or ‘P(-I(R3))’ (without the single quotation marks) into the Trace Expression field (see Figure 6.7(b2)).
- Click OK to close the Add Traces dialog box and click the Toggle Cursor button (on the toolbar in the Probe window) to call the cross-type cursor, use the right mouse pointer and/or the shift+left/right arrow key to locate the second cursor near the source frequency of 60 Hz and read the value of $\angle \mathbf{I}_{R_3}$ from the Probe Cursor box (Figure 6.7(b1)). Is it close to 140° , which is analytically obtained in Equation (E6.1.18)?
- Now, in order to see the current $i_{R_3}(t)$ in the time domain, click the Place Part button on the tool palette in the Schematic Editor window to place the VSIN part in place of the VAC part (Figure 6.7(c)) and set its parameters as

$$\text{VAMPL} = 450, \quad \text{VOFF} = 0, \quad \text{and} \quad \text{FREQ} = 60$$

- Click the New Simulation Profile button (on the toolbar) and name it, say, tran.
- In the Simulation Settings dialog box opened by clicking the Edit Simulation Settings button (on the toolbar), set the Analysis type to ‘Time Domain (Transient)’, Run_to_time to 100 ms, and Maximum step to 100 μ s, respectively.
- Click the Current Marker button on the toolbar to place a current probe pin at a terminal of R_3 .
- Click Run (on the toolbar) and see the graph of $i_{R_3}(t)$ for $0 \leq t \leq 100$ ms on the Probe window (Figure 6.7(d)).
- Click the Toggle Cursor button (on the toolbar in the Probe window) to call the cross-type cursor, use the left mouse pointer and/or the left/right arrow key, or just click the Cursor Peak button to locate the cursor at the peak point and read the value of $i_{R_3, \text{peak}} = |\mathbf{I}_{R_3}|$ from the Probe Cursor box. Is it close to 3.9, which is analytically obtained in Equation (E6.1.18)?
- Use the right mouse pointer and/or the shift+left/right arrow key to locate the cursor near the zero-crossing point and read the zero-crossing time from the Probe Cursor box. From the zero-crossing time of 10.186 ms, it is possible to find that the initial phase of $i_{R_3}(t)$ is rather $360^\circ - 220^\circ = 140^\circ$ than $-\omega t_0 = -2\pi \times 60 \times 10.186 \text{ ms} = -3.84 \text{ rad} = -3.84 \times 180/\pi = -220^\circ$. Since the current waveform can be expressed as $i_{R_3}(t) = 3.9 \sin(2\pi ft - 220^\circ) = 3.9 \sin(2\pi ft + 140^\circ)$ and the voltage source waveform generated by the VSIN part is $v_s(t) = 450 \sin(2\pi ft)$, $i_{R_3}(t)$ is said to lead $v_s(t)$ by $140^\circ (< 180^\circ)$ rather than to lag behind $v_s(t)$ by $220^\circ (> 180^\circ)$.
- The two PSpice simulation results obtained from the AC steady state analysis (with VAC) and the transient analysis (with VSIN) agree with the analytical result (E6.1.18).



(b2) The Add Traces dialog box opened by clicking the Add Traces icon on the tool bar of the PSpice A/D (Probe) window



(c) With the voltage source VSIN (d) The PSpice simulation result (Time Domain - Transient analysis) with the sine-wave voltage source VSIN for the current through R3

Figure 6.7 PSpice simulation for Example 6.1

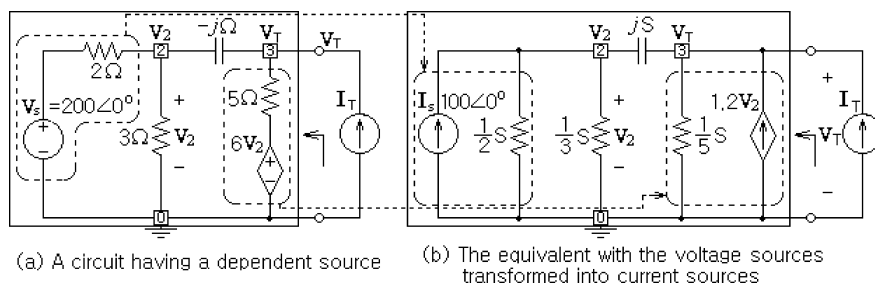


Figure 6.8 (for Example 6.2)

(Example 6.2) The Thevenin Equivalent of a Circuit Having a Dependent Source

Find the Thevenin equivalent of the circuit in Figure 6.8(a) seen from the right side (terminals 3-0). While applying a test voltage source makes a set of three mesh equations, applying a test current source makes a set of two node equations, which is easier to solve. Therefore, in the teeth of two voltage-to-current source transformations, as depicted in Figure 6.8(b), a test current source is applied, the node equation is set up, and then it is solved to express the voltage V_T across the test current source in terms of I_T as follows:

$$\begin{bmatrix} 1/2 + 1/3 + j & -j \\ -j & 1/5 + j \end{bmatrix} \begin{bmatrix} V_2 \\ V_T \end{bmatrix} = \begin{bmatrix} 100 \\ I_T + 1.2V_2 \end{bmatrix} \quad (\text{E6.2.1})$$

$$\begin{bmatrix} 5 + j6 & -j6 \\ -6 - j5 & 1 + j5 \end{bmatrix} \begin{bmatrix} V_2 \\ V_T \end{bmatrix} = \begin{bmatrix} 600 \\ 5I_T \end{bmatrix} \quad (\text{E6.2.2})$$

$$\begin{bmatrix} V_2 \\ V_T \end{bmatrix} = \frac{1}{5 - j5} \begin{bmatrix} 1 + j5 & j6 \\ 6 + j5 & 5 + j6 \end{bmatrix} \begin{bmatrix} 600 \\ 5I_T \end{bmatrix} = \frac{1}{1 - j} \begin{bmatrix} 120(1 + j5) + j6I_T \\ 120(6 + j5) + (5 + j6)I_T \end{bmatrix} \quad (\text{E6.2.3})$$

$$V_T = (-0.5 + j5.5)I_T + 60(1 + j11) \quad (\text{E6.2.4})$$

Matching this expression with Equation (2.14) yields the Thevenin equivalent voltage source and impedance of the circuit depicted in Figure 6.8(a):

$$V_{Th} = 60(1 + j11), \quad Z_{Th} = -0.5 + j5.5 \quad (\text{E6.2.5})$$

This result can be obtained by typing the following statements into the MATLAB command window:

```
>> syms IT; Y=[ 5+6i -6i; -6-5i 1+5i]; V=Y\[ 600; 5*IT]; % Eq. (E6.2.2)
>> V(2) % Eq. (E6.2.4)
ans = 60+660*i -1/2*IT+11/2*i*IT
>> Vth=subs(V(2),'IT',0) % Open-circuit voltage Eq. (E6.2.5)
Vth = 6.0000e+001 +6.6000e+002i
>> Zth=subs(V(2)-Vth,'IT',1) % Thevenin equivalent impedance Eq. (E6.2.5)
Zth = -0.5000 + 5.5000i
```

(Example 6.3) The Bridge Balance Condition

Figure 6.9 shows a bridge circuit, which can be used to measure an unknown inductance/capacitance as well as an unknown resistance based on the bridge balance condition. The bridge is balanced when

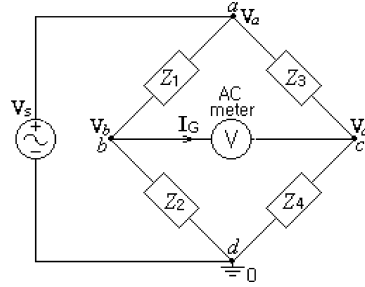


Figure 6.9 A bridge circuit (for Example 6.3)

no current flows through the meter, corresponding to $V_b = V_c$. This *bridge balance condition* can be written as

$$V_b = \frac{Z_2}{Z_1 + Z_2} V_s = \frac{Z_4}{Z_3 + Z_4} V_s = V_c; \quad \frac{Z_2}{Z_1 + Z_2} = \frac{Z_4}{Z_3 + Z_4} \quad (\text{E6.3.1})$$

$$\frac{Z_1}{Z_2} = \frac{Z_3}{Z_4}; \quad Z_1 Z_4 = Z_2 Z_3 \quad (\text{E6.24})$$

(Example 6.4) A Circuit Containing Magnetically Coupled Coils

Figure 6.10(a) shows a circuit containing two magnetically coupled coils. Replace the coupled coils by the equivalent model (Figure 5.6.1(b)) with dependent sources as depicted in Figure 6.10(b), where the minus sign of the CCVS (current-controlled voltage source) stems from the fact that one coil current I_1 enters the dotted end of the coil L_1 while the other coil current I_2 enters the undotted end of the coil L_2 . Then the mesh equation can be set up and solved as

$$\begin{bmatrix} 1-j & -(-j2) \\ -(-j2) & 1+j \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} 3+jI_2 \\ jI_1 \end{bmatrix}; \quad \begin{bmatrix} 1-j & j \\ j & 1+j \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} 3 \\ 0 \end{bmatrix} \quad (\text{E6.4.1})$$

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} 1-j & j \\ j & 1+j \end{bmatrix}^{-1} \begin{bmatrix} 3 \\ 0 \end{bmatrix} = \frac{1}{3} \begin{bmatrix} 1+j & -j \\ -j & 1-j \end{bmatrix} \begin{bmatrix} 3 \\ 0 \end{bmatrix} = \begin{bmatrix} 1+j \\ -j \end{bmatrix} = \begin{bmatrix} \sqrt{2}/\pi/4 \\ 1/\pi/2 \end{bmatrix} = \begin{bmatrix} \sqrt{2}/45^\circ \\ 1/\pi/2 \end{bmatrix} \quad (\text{E6.4.2})$$

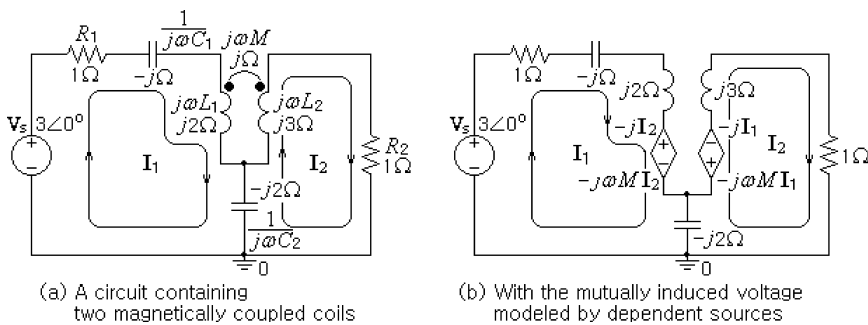


Figure 6.10 (for Example 6.4)

```
>> I=[ 1-i i; i 1+i]\[ 3; 0]; % Eq. (E6.4.2)
>> [ I abs(I) angle(I)*180/pi] %with their magnitude & phase (in degree)
    1.0000 + 1.0000i    1.4142    45
    0 - 1.0000i    1.0000   -90
```

6.5 Instantaneous, Active, Reactive, and Complex Power

In case of DC, the power supplied or dissipated by an element or a circuit having the terminal voltage V and current I (with the voltage polarity and current direction conforming to the passive sign convention (Section 1.2.4)) is simply $P = VI$ (Equation (1.4b)), since V and I are constant. In contrast, AC circuits have time-varying voltages and currents. Accordingly, Equation (1.4a) is used to write the instantaneous (AC) power of an element or a circuit having the terminal voltage $v(t)$ and current $i(t)$ as

$$p(t) \stackrel{(1.4a)}{=} v(t)i(t) = V_m \cos(\omega t + \theta_v) I_m \cos(\omega t + \theta_i) \quad (6.25a)$$

$$\stackrel{(F.11)}{=} \sqrt{2}V\sqrt{2}I\frac{1}{2}[\cos(\theta_v - \theta_i) + \cos(2\omega t + \theta_v + \theta_i)]$$

$$\stackrel{(F.6)}{=} VI[\cos(\theta_v - \theta_i) + \cos(\theta_v - \theta_i)\cos(2\omega t + 2\theta_i) - \sin(\theta_v - \theta_i)\sin(2\omega t + 2\theta_i)]$$

$$= P[1 + \cos(2\omega t + 2\theta_i)] - Q \sin(2\omega t + 2\theta_i) \quad (6.25b)$$

$$\stackrel{(F.15),(F.16)}{=} 2P \cos^2(\omega t + \theta_i) - 2Q \sin(\omega t + \theta_i) \cos(\omega t + \theta_i) \quad (6.25c)$$

where

$$P = VI \cos(\theta_v - \theta_i) = VI \cos \theta \text{ [W]: the active or average or real power} \quad (6.26)$$

$$Q = VI \sin(\theta_v - \theta_i) = VI \sin \theta \text{ [VAR or VA Reactive]: the reactive power} \quad (6.27)$$

with the voltage–current phase difference $\theta = \theta_v - \theta_i$ (equal to the load impedance angle)

Noting that the positive/negative power means the power absorbed/supplied by the load, let us think about the concepts of the active power P and the reactive power Q . The *active* (or *average*) power P is simply the time average of the instantaneous power $p(t)$ over one period T , i.e.

$$\frac{1}{T} \int_{t_0}^{t_0+T} p(\tau) d\tau = \frac{1}{\omega T} \int_{\omega t_0}^{\omega(t_0+T)} p(\omega\tau) d(\omega\tau)$$

$$\stackrel{\omega t = \theta, \omega T = 2\pi}{(6.25b)} = \frac{1}{2\pi} \int_{\theta_0}^{\theta_0+2\pi} \{P[1 + \cos(2\theta + 2\theta_i)] - Q \sin(2\theta + 2\theta_i)\} d\theta = P$$

This explains why it is called ‘the average power’. What about the reactive power Q ? To figure out its physical meaning, let us see the active and reactive powers together with the instantaneous power for four different types of load impedance (phase) angle θ :

1. For a purely resistive load with $\theta = \theta_v - \theta_i = 0$,

$$P = VI \cos \theta = VI, \quad Q = VI \sin \theta = 0$$

and

$$p(t) = 2P \cos^2(\omega t + \theta_i) \stackrel{(F.15)}{=} P[1 + \cos(2\omega t + 2\theta_i)] \geq 0 \quad \forall t$$

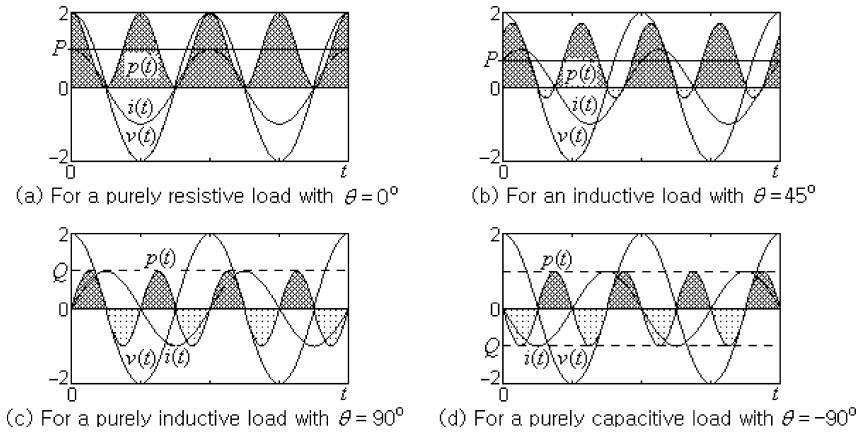


Figure 6.11 The instantaneous power depending on the load impedance phase angle

which is depicted in Figure 6.11(a).

2. For an inductive load with $\theta = \theta_v - \theta_i = \pi/4$,

$$P = VI \cos \theta = \frac{1}{\sqrt{2}} VI, \quad Q = VI \sin \theta = \frac{1}{\sqrt{2}} VI$$

and

$$p(t) = 2P \cos^2(\omega t + \theta_i) - 2Q \sin(\omega t + \theta_i) \cos(\omega t + \theta_i)$$

which is depicted in Figure 6.11(b).

3. For a purely inductive load with $\theta = \theta_v - \theta_i = \pi/2$,

$$P = VI \cos \theta = 0, \quad Q = VI \sin \theta = VI, \quad \text{and} \quad p(t) = -Q \sin(2\omega t + 2\theta_i)$$

which is depicted in Figure 6.11(c).

4. For a purely capacitive load with $\theta = \theta_v - \theta_i = -\pi/2$,

$$P = VI \cos \theta = 0, \quad Q = VI \sin \theta = -VI, \quad \text{and} \quad p(t) = -Q \sin(2\omega t + 2\theta_i)$$

which is depicted in Figure 6.11(d).

It is implied that purely inductive/capacitive loads do not dissipate the electric energy, but just store (absorb) some electric energy (measured as Q) in the magnetic/electric field and return it back, consuming no energy. In contrast, purely resistive loads always consume some electric energy, never returning any part of it. Therefore, the *reactive power* Q can be interpreted as a measure of the AC power being exchanged between the inductive/capacitive loads and the source driving them. Note that the sign of the reactive power is positive/negative for inductive/capacitive loads.

Now that the concepts of the active and reactive powers are established, one might wonder how to compute them from the phasor voltage/current having the magnitude and phase of the AC

voltage/current. The answer is the *complex power* that can be obtained from the multiplication of the rms phasor voltage by the conjugate of the rms phasor current as

$$S = \mathbf{V} \mathbf{I}^* = V \angle \theta_v \times I \angle -\theta_i = VI \angle (\theta_v - \theta_i) = VI \angle \theta = VI (\cos \theta + j \sin \theta) = P + jQ \quad (6.28)$$

where the real part (in-phase component) and the imaginary part (quadrature component) of the complex power are the active and reactive powers, respectively. The magnitude of the complex power, VI [VA], is referred to as the *apparent power*:

$$|S| = \sqrt{P^2 + Q^2} = VI \text{ [VA]} \quad (6.29)$$

For AC devices, the requirement on the apparent power can be more critical than that on the active power since it specifies the voltage–current capacity in terms of the magnitudes of AC voltage and current. The units VA (volt-ampere), watt, and VAR (volt-ampere reactive) are used for the apparent power, the active power, and the reactive power, respectively.

$\mathbf{V} = \mathbf{Z} \mathbf{I} = (R + jX) \mathbf{I}$ or $\mathbf{I} = \mathbf{Y} \mathbf{V} = (G + jB) \mathbf{V}$ can be substituted into Equation (6.28) to obtain

$$S = \mathbf{V} \mathbf{I}^* = \mathbf{Z} \mathbf{I} \mathbf{I}^* = \mathbf{Z} I^2 = (R + jX) I^2; \quad P + jQ = R I^2 + jX I^2$$

$$P = R I^2, \quad Q = X I^2 \quad (6.30a)$$

$$S = \mathbf{V} \mathbf{I}^* = \mathbf{V} \mathbf{Y}^* \mathbf{V}^* = \mathbf{Y}^* V^2 = (G - jB) V^2; \quad P + jQ = G V^2 + j(-B) V^2$$

$$P = G V^2, \quad Q = (-B) V^2 \quad (6.30b)$$

which are alternate expressions for the active and reactive powers, respectively.

The complex, active, and reactive powers and their relations are best described by the *power triangle* in Figure 6.12, where its hypotenuse, horizontal side, vertical side, and the length of the hypotenuse represent the complex, active, reactive, and apparent powers, respectively.

Note. In fact, the power triangle of $S = Z I^2$ has the same shape as the impedance angle of Z (Figure 6.3(b)), since the power triangle can be obtained from the impedance triangle by multiplying each side by I^2 .

The following is the law of AC power conservation stated in different forms.

[Remark 6.4] Power Conservation Law

The algebraic sums of the instantaneous, complex, active, and reactive powers of all the elements in a circuit are zero.

$$\sum_k p_k = \sum_k v_k i_k \stackrel{\text{(P1.6.5)}}{=} 0, \quad \sum_{\text{every element } k} S_k = \sum_k \mathbf{V}_k \mathbf{I}_k^* = 0, \quad \sum_k P_k = 0, \quad \sum_k Q_k = 0 \quad (6.31)$$

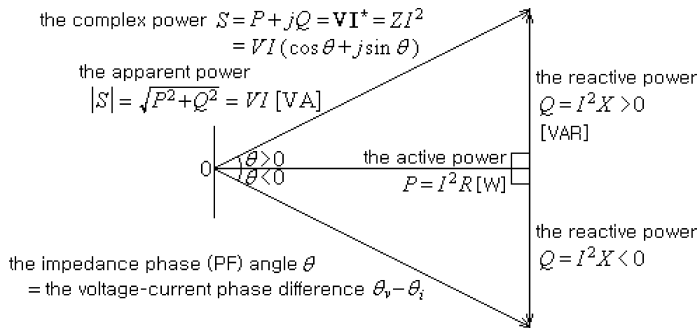


Figure 6.12 The power triangle

It can equivalently be said that the sums of the instantaneous, complex, active, and reactive powers delivered by all AC sources equal the sums of the instantaneous, complex, active (or average or real), and reactive powers of all passive elements in a circuit, respectively.

[Remark 6.5] Magnitude of Load Impedance and Power

A load impedance is said to be large/small if its power is large/small. Since the impedances connected in series have the current in common, their powers are proportional to their impedances: $S_k = Z_k I^2$. In contrast, since the impedances connected in parallel have the voltage in common, their powers are proportional to their admittances: $S_k = Y_k V^2$. That is why a series RLC circuit is inductive/capacitive depending on whether the inductive reactance ωL (the magnitude of the impedance of L) is greater/smaller than the capacitive reactance $1/(\omega C)$ (the magnitude of the impedance of C), while a parallel RLC circuit is inductive/capacitive depending on whether the inductive susceptance $1/(\omega L)$ (the magnitude of the admittance of L) is greater/smaller than the capacitive susceptance ωC (the magnitude of the admittance of C), as discussed in Section 6.4.

6.6 Power Factor

As a measure of how well an AC power is being used, the *power factor* (PF) is defined to be the ratio of the active power to the apparent power, i.e.

$$\text{PF (power factor)} = \frac{P}{VI} = \frac{P}{\sqrt{P^2 + Q^2}} = \cos \theta \quad (6.32)$$

where $\theta = \theta_v - \theta_i$ is the phase angle of the load impedance and is referred to as the *power factor angle*. Noting that the power factors are the same for an inductive load with $\theta = \theta_L > 0$, i.e. $\theta_i < \theta_v$, and a capacitive load with $\theta = -\theta_L < 0$, i.e. $\theta_i > \theta_v$, they are differentiated by referring to the former as a *lagging* PF and the latter as a *leading* PF, where the two terms, lagging/leading, refer to the phase of the current w.r.t. the voltage.

What is the significance of a high PF (close to 1) or a low PF (close to 0)? To figure this out, let us examine the two power triangles having the same active power P_L as depicted in Figure 6.13, one with a low PF $\cos \theta$ and the other with a high PF $\cos \theta_c$. What is their difference? They differ in the apparent power ($VI_L > VI_{L,c}$) and the reactive power ($Q_L > Q_{L,c}$). This implies that a load with a low PF needs more load current ($I_L > I_{L,c}$) than one with a high PF to consume the same active power at the same load voltage V . More load current in a transmission line causes more power loss ($I_L^2 R_l > I_{L,c}^2 R_l$; R_l : the resistance in the transmission line) and a larger voltage drop across the transmission line; i.e. a lower PF of a load requires the generator to produce more power with higher voltage in order to supply the load

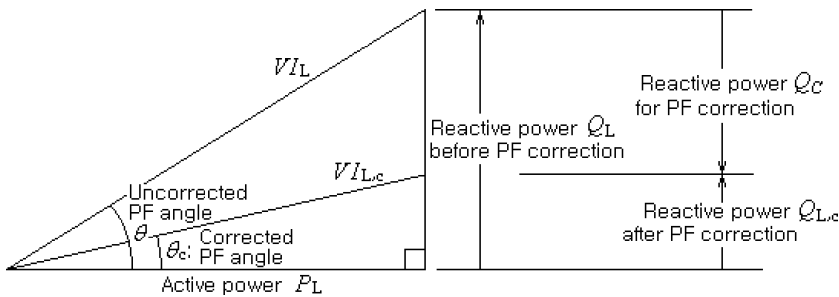


Figure 6.13 Power factor (PF) correction and power triangle

with the same active power at the same voltage level. This results in a waste of energy as well as an increased production cost of power and high electric fee, which is undesirable for both the power company and the consumers. That is why the power company encourages large industrial users to raise their PFs by charging lower rates for consumers with higher PFs and/or by imposing a penalty on consumers with a PF lower than, say, 0.9 lagging.

Now is the time to consider how to improve (raise) the PF, which is called the *power factor correction* or *compensation*. Most industrial loads requiring a large amount of power consist of electric motors and so are normally inductive, having lagging power factors. Hence, a higher PF can be achieved by placing a capacitor of fixed/adjustable capacitance or the apparatuses having capacitive reactive power (like a static VAR compensator or a synchronous condenser) in parallel with the load.

Note. You can visit the following website for more information about PF correction techniques including a synchronous condenser: <http://www.pserc.org/ecow/get/generalinf/presentati/presentati/heydt_synchronous_mach_sep03.pdf>.

Note. A capacitive load would need to be connected with an inductor for power factor correction.

In the light of the two power triangles depicted in Figure 6.13, the capacitive reactive power needed to improve the PF from PF to PF_c is

$$Q_C = Q_{L,c} - Q_L = P_L (\tan \theta_c - \tan \theta) = P_L [\tan (\cos^{-1} PF_c) - \tan (\cos^{-1} PF)] \quad (6.33)$$

The PF is improved by the capacitive reactive power $Q_C (< 0)$ as

$$PF = \cos \theta = \frac{P_L}{\sqrt{P_L^2 + Q_L^2}} \rightarrow PF_c = \cos \theta_c = \frac{P_L}{\sqrt{P_L^2 + (Q_L + Q_C)^2}} \quad (6.34)$$

The capacitive reactance $X_C = -1/(\omega C)$ to be installed for the capacitive reactive power Q_C is

$$X_C = \frac{V^2}{Q_C} (< 0) \quad (6.35)$$

since the complex power of the PF correcting capacitor with a capacitive reactance $X_C = -1/(\omega C)$ at the phasor voltage $\mathbf{V}$ is

$$S_C \stackrel{(6.28)}{=} \mathbf{V} \mathbf{I}_C^* = \mathbf{V} \left[\frac{\mathbf{V}}{1/(j\omega C)} \right]^* = \mathbf{V} \mathbf{V}^* \left(\frac{1}{jX_C} \right)^* = j \frac{V^2}{X_C} = jQ_C \text{ [VAR]} \quad (6.36)$$

The following example illustrates a simple method for PF compensation, which is to install a capacitor in parallel with the load, where we find the transmission power loss, the generator voltage, and the voltage regulation affected by the power factor.

(Example 6.5) Power Factor Compensation or Correction

Consider the power system of Figure 6.14(a) in which two loads Z_1 and Z_2 are fed through a transmission line with an impedance $Z_l = 0.3 + j0.4 \Omega$. At the rated voltage of 250 V, the capacitive load Z_1 consumes an active power of 8 kW at a leading PF of 0.8 and the inductive load Z_2 consumes an apparent power of 20 kVA at a lagging PF of 0.6.

(a) Find the overall complex power and power factor:

$$\begin{aligned} S_1 &= P_1 + jQ_1 = P_1 \left(1 - j \frac{\sin \theta_1}{\cos \theta_1} \right) = P_1 \left(1 - j \frac{\sqrt{1 - \cos^2 \theta_1}}{\cos \theta_1} \right) \\ &= 8000 \left(1 - j \frac{0.6}{0.8} \right) = 8000 - j6000 \end{aligned} \quad (\text{E6.5.1})$$

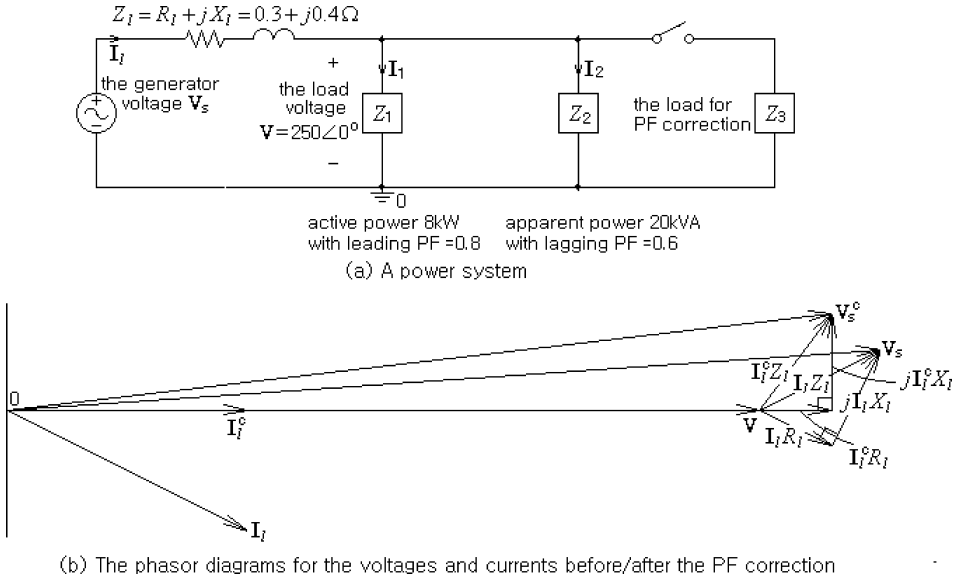


Figure 6.14 (for Example 6.5)

$$S_2 = P_2 + jQ_2 = V I_2 (\cos \theta_2 + j \sqrt{1 - \cos^2 \theta_2}) = 20\,000 (0.6 + j0.8) = 12\,000 + j16\,000 \quad (\text{E6.5.2})$$

$$S_L = S_1 + S_2 = 20\,000 [\text{W}] + j10\,000 [\text{VAR}] = P + jQ \quad (\text{E6.5.3})$$

$$PF = \cos \theta \stackrel{(6.32)}{=} \frac{P}{\sqrt{P^2 + Q^2}} = \frac{20\,000}{\sqrt{20\,000^2 + 10\,000^2}} = 0.8944, \quad \text{lagging} \quad (\text{E6.5.4})$$

Note. Why is the overall PF lagging? Because the total reactive power Q is positive.

- (b) Find the currents through the two impedances and the transmission line, the transmission loss P_l , the generator voltage $\mathbf{V}_s$, and the load voltage regulation $|\mathbf{V}_s - \mathbf{V}|/\mathbf{V}$:

$$\mathbf{I}_1^* \stackrel{(6.28)}{=} S_1/\mathbf{V} \stackrel{(\text{E6.5.1})}{=} (8000 - j6000)/250 = 32 - j24; \quad \mathbf{I}_1 = 32 + j24 \quad (\text{E6.5.5})$$

$$\mathbf{I}_2^* \stackrel{(6.28)}{=} S_2/\mathbf{V} \stackrel{(\text{E6.5.2})}{=} (12\,000 + j16\,000)/250 = 48 + j64; \quad \mathbf{I}_2 = 48 - j64 \quad (\text{E6.5.6})$$

$$\mathbf{I}_l = \mathbf{I}_1 + \mathbf{I}_2 = (S_L/\mathbf{V})^* = 80 - j40 [\text{A}] \quad (\text{E6.5.7})$$

$$P_l \stackrel{(6.30a)}{=} I_l^2 R_l = |\mathbf{I}_l|^2 R_l = [80^2 + (-40)^2] \times 0.3 = 2400 [\text{W}] \quad (\text{E6.5.8})$$

$$\mathbf{V}_s = \mathbf{V} + Z_l \mathbf{I}_l = 250 + (0.3 + j0.4)(80 - j40) = 290 + j20 [\text{V}] \quad (\text{E6.5.9})$$

$$V_s = |\mathbf{V}_s| = \sqrt{290^2 + 20^2} \simeq 291 [\text{V}] \quad (\text{E6.5.10})$$

This implies that the generator must maintain its output voltage level at 291 V to keep the load voltage at 250 V and generate the average power of $P_L + P_l = 20\,000 + 2400 = 22\,400 [\text{W}]$ including the transmission line loss to meet the average power 20 000 W of the load.

As a measure of how severely the load voltage fluctuates depending on the load, the *load voltage regulation* is

$$\frac{|\text{Load voltage with no load} - \text{load voltage with full load}|}{\text{Load voltage with full load}} = \frac{|\mathbf{V}_s - \mathbf{V}|}{|\mathbf{V}|} = \frac{|290 + j20 - 250|}{250} \times 100 = 18\% \quad (\text{E6.5.11})$$

(c) Check whether the power conservation law expressed by Equation (6.31) is satisfied.

$$\text{Load : } S_L = S_1 + S_2 = 20\,000 [\text{W}] + j10\,000 [\text{VAR}]$$

$$\text{Line : } S_l = \mathbf{V}_l \mathbf{I}_l^* = Z_l \mathbf{I}_l \mathbf{I}_l^* = Z_l |\mathbf{I}_l|^2 = (0.3 + j0.4)(80^2 + 40^2) = 2400 [\text{W}] + j3200 [\text{VAR}]$$

$$\text{Generator : } S_s = \mathbf{V}_s \mathbf{I}_s^* = \mathbf{V}_s (-\mathbf{I}_l^*) = (290 + j20)(-80 - j40) = -22\,400 [\text{W}] - j13\,200 [\text{VAR}]$$

Sum of the complex powers of all the elements:

$$S_L + S_l + S_s = 20\,000 + j10\,000 + 2400 + j3200 - 22\,400 - j13\,200 = 0 \quad (\text{E6.5.12})$$

(d) Noting that $S_L = S_1 + S_2 = 20\,000 [\text{W}] + j10\,000 [\text{VAR}]$, find the reactance of the capacitive load that is to be installed in parallel with the existing load to compensate for the reactive power by $Q_C = -10\,000 [\text{VAR}]$ so that the PF can be corrected to 1:

$$X_C \left(= -\frac{1}{\omega C} \right) \stackrel{(6.35)}{=} \frac{V^2}{Q_C} = \frac{250^2}{-10\,000} = -6.25 \, \Omega \quad (\text{E6.5.13})$$

(e) In order to see the effect of power factor correction, find the line current $\mathbf{I}_l^c$, the transmission loss P_l^c , the generator voltage $\mathbf{V}_s^c$, and the load voltage regulation $|\mathbf{V}_s^c - \mathbf{V}|/|\mathbf{V}|$ of the power system with the PF corrected to 1:

$$\mathbf{I}_l^c \stackrel{(6.28)}{=} (S_L^c / \mathbf{V})^* = [(S_L + jQ_C) / \mathbf{V}]^* = (20\,000 / 250)^* = 80 \, \text{A} \quad (\text{E6.5.14})$$

$$P_l^c \stackrel{(6.30a)}{=} (\mathbf{I}_l^c)^2 R_l = |\mathbf{I}_l^c|^2 R_l = 80^2 \times 0.3 = 1920 \, \text{W} \quad (\text{E6.5.15})$$

$$\mathbf{V}_s^c = \mathbf{V} + Z_l \mathbf{I}_l^c = 250 + (0.3 + j0.4)80 = 274 + j32 \, \text{V} \quad (\text{E6.5.16})$$

$$V_s^c = |\mathbf{V}_s^c| = \sqrt{274^2 + 32^2} \simeq 276 \, \text{V} \quad (\text{E6.5.17})$$

It turns out that the generator voltage is lowered from 291 V to 276 V and the transmission loss decreases from 2400 W to 1920 W. The *load voltage regulation* also decreases from 18 % to

$$\frac{|\mathbf{V}_s^c - \mathbf{V}|}{|\mathbf{V}|} = \frac{|274 + j32 - 250|}{250} \times 100 = 16\% \quad (\text{E6.5.18})$$

The phasor diagram in Figure 6.14(b) shows the phasor currents and voltages before/after the power factor compensation so that it is possible to see how the line current, the voltage drop across the transmission line, and the generator voltage are changed by the PF compensation, while the load voltage is kept at the same level.

Given the desired PF (Pfc), the complex power (SL) of the load, the transmission line impedance (Zl), and the rated load voltage (V), the above routine `PF_correction()` computes the reactance (Xc) of the PF compensating load, the reactive power (Qc) to be added for the PF correction, the transmission losses (Ploss_c/Ploss), the voltage regulations (Vreg_c/Vreg), and the generator voltages (Vs_c/Vs) of the power system after/before the PF correction. Interested readers are invited to run the program `do_PF_correction.m` to get the power factor correction results and compare them with the results obtained above.

```

function [Xc,Qc,Ploss_c,Vreg_c,Vs_c,Ploss,Vreg,Vs]=...
    PF_correction(PFc,SL,Zl,V)
%Input: PFc = desired (corrected) Power Factor
%      SL = Complex Power of loads (PL+j*QL)
%      Zl = the line impedance (Rl+j*Xl)
%      V = Rated rms Load Voltage at the receiving end
%Output: Xc = Reactance of the PF compensating load
%      Qc = Reactive power to be added for PF correction
%      Ploss_c = Power loss after PF correction
%      Vreg_c = Voltage regulation after PF correction
%      Vs_c = rms Generator voltage after PF correction
%      Ploss = Power loss before PF correction
%      Vreg = Voltage regulation before PF correction
%      Vs = rms Generator voltage before PF correction
%      Copyleft: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only theta=
angle(sum(SL)); %PF angle
PF= sign(theta)*cos(theta); %overall PF without compensation
% +/- for inductive/capacitive load(lagging/leading)
Il= conj(SL/V); Rl= real(Zl); % line current and line resistance
Ploss= abs(Il)^2*Rl; % Transmission loss
Vs= V + Zl*Il; % The generator voltage
Vreg= abs((Vs-V)/V); % The voltage regulation
Vs= abs(Vs); % The rms amplitude of generator voltage
% Reactive power and capacitive reactance to be added for PF compensation
[Qc,Xc]= Qc_for_PF_correction(SL,PFc,V);
% Qc= real(SL)*(tan(acos(PFc))-tan(acos(PF))); %Reactive power to be added
% Xc= V*conj(V)/Qc; % Capacitive reactance for PF correction
SL_c= SL + j*Qc; % The complex power of the load after PF correction
Il_c= conj(SL_c/V); % line current after PF correction
Ploss_c= abs(Il_c)^2*Rl; % Transmission loss after PF correction
Vs_c= V+Zl*Il_c; % The generator voltage after PF correction
Vreg_c= abs((Vs_c-V)/V); % The voltage regulation after PF correction
Vs_c=abs(Vs_c); %The rms amplitude of Vs after PF correction

```

```

function [Qc,Xc,C]=Qc_for_PF_correction(SL,PFc,V,w)
% Finds the reactive power Qc to be added
% for correcting the PF of a given complex power (SL) to lagging PFc
% and computes Xc and C if V (voltage) and w (radian frequency) are given
Qc= real(SL)*tan(acos(PFc))-imag(SL); % Eq. (6.33)
if nargin>2, Xc= V*V./Qc; % -1/wC Capacitive reactance for PF correction
    if nargin>3, C= -1./Xc/w; end % Capacitance for PF correction
end

```

```

function wC=wC_for_PF_correction(ZL,PFd)
% Finds the capacitive susceptance wC to be connected in parallel
% for correcting the PF of a given inductive impedance ZL to lagging PFd
YL= 1./ZL; BL= -imag(YL); wC= BL-real(YL)*tan(acos(PFd));

```

```

%do_PF_correction.m
clear, clf
PFc=1; S1=8000-6000j; S2=12000+16000j; Zl=0.3+0.4j; V=250; f=60; w=2*pi*f;
[Xc,Qc,Ploss_c,Vreg_c,Vs_c,Ploss,Vreg,Vs]= PF_correction(PFc,S1+S2,Zl,V)
if Qc<0 % if reactive power added for PF compensation is negative
    C= -1/Xc/w % Capacitance
end

```

6.7 Maximum Power Transfer – Impedance Matching

In this section the condition for maximum average power transfer in an AC circuit will be derived, which has a wide range of applications in electronic circuits for communication and audio systems. Consider the circuit of Figure 6.15 in which an AC voltage source $\mathbf{V}_s$ with a source impedance $Z_s = R_s + jX_s$ in series drives a load impedance $Z_L = R_L + jX_L$. In fact, $\mathbf{V}_s$ and Z_s can be regarded as the Thevenin equivalent of a general AC source network.

In order to find the maximum power transfer condition, the average power delivered to the load is written as

$$\mathbf{I} = \frac{\mathbf{V}_s}{Z_s + Z_L} = \frac{\mathbf{V}_s}{(R_s + R_L) + j(X_s + X_L)}$$

$$P \stackrel{(6.30a)}{=} |\mathbf{I}|^2 R_L \stackrel{(C.4)}{=} \frac{V_s^2 R_L}{(R_s + R_L)^2 + (X_s + X_L)^2} \quad \text{with } V_s = |\mathbf{V}_s| \quad (6.37)$$

With only X_L free to be adjusted, this average power is maximized for

$$X_L = -X_s$$

which minimizes the denominator. With X_L fixed as this value and only R_L free to be adjusted, the derivative of the average power expression (6.37) is taken w.r.t. R_L and set to zero to find the maximizing value of R_L as

$$\frac{\partial}{\partial R_L} P \stackrel{(6.37)}{=} \frac{\partial}{\partial R_L} \left[\frac{V_s^2 R_L}{(R_s + R_L)^2} \right] \stackrel{(F.31)}{=} V_s^2 \frac{(R_s + R_L)^2 - 2R_L(R_s + R_L)}{(R_s + R_L)^4} = V_s^2 \frac{R_s - R_L}{(R_s + R_L)^3} = 0$$

$$R_L = R_s$$

Combining these two results yields the *maximum power transfer* or *impedance matching* condition

$$Z_L = R_L + jX_L \equiv R_s - jX_s = Z_s^* \quad (6.38)$$

This implies that the AC power transferred from a source network is maximized when the load impedance is the complex conjugate of the source impedance, i.e. the output impedance of the source network seen from the terminals of the load. When this condition is met, the load impedance is said to be matched to the source impedance and the two matched impedances form a resonant circuit.

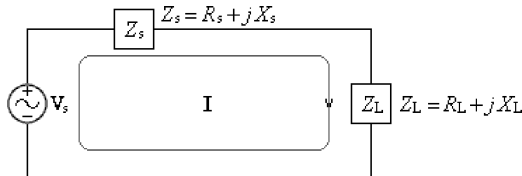


Figure 6.15 A circuit for maximum power transfer

Consider the case where the range of variable load impedance $Z_L = R_L + jX_L$ is limited. For the maximum power transfer in this case, the load reactance X_L can be adjusted near to $-X_s$ and then the load resistance R_L can be adjusted close to

$$R_L = \sqrt{R_s^2 + (X_s + X_L)^2} \quad (6.39)$$

at which

$$\frac{\partial}{\partial R_L} P \stackrel{(6.37)}{=} \frac{\partial}{\partial R_L} \left[\frac{V_s^2 R_L}{(R_s + R_L)^2 + (X_s + X_L)^2} \right] \stackrel{(F.31)}{=} V_s^2 \frac{(R_s + R_L)^2 + (X_s + X_L)^2 - 2R_L(R_s + R_L)}{[(R_s + R_L)^2 + (X_s + X_L)^2]^2} = 0$$

Now consider the case where only the magnitude $|Z_L|$ of the load impedance

$$Z_L = |Z_L| \angle \theta \stackrel{(C.4)}{=} |Z_L| \cos \theta + j|Z_L| \sin \theta$$

can be adjusted with the load impedance phase angle θ fixed. For this case, the derivative of the average power expression (6.37) is taken w.r.t. $|Z_L|$ and set to zero to find the maximizing value of $|Z_L|$ as

$$\begin{aligned} \frac{\partial}{\partial |Z_L|} P &\stackrel{(6.37)}{=} \frac{\partial}{\partial |Z_L|} \left[\frac{V_s^2 R_L}{R_s^2 + X_s^2 + R_L^2 + X_L^2 + 2(R_s R_L + X_s X_L)} \right] \\ &= \frac{\partial}{\partial |Z_L|} \left[\frac{V_s^2 |Z_L| \cos \theta_L}{|Z_s|^2 + |Z_L|^2 + 2|Z_s||Z_L|(\cos \theta_s \cos \theta_L + \sin \theta_s \sin \theta_L)} \right] \\ &\stackrel{(F.6), (F.31)}{=} \frac{V_s^2 \cos \theta_L (|Z_s|^2 + |Z_L|^2 - 2|Z_L|^2)}{[|Z_s|^2 + |Z_L|^2 + 2|Z_s||Z_L| \cos(\theta_s - \theta_L)]^2} = 0 \\ &|Z_s| = |Z_L| \end{aligned} \quad (6.40)$$

A question arises: ‘How do we meet this condition?’ The answer is to insert a transformer, hopefully an ideal one with no loss, between the source and the load, whose turns ratio is

$$\frac{N_2}{N_1} = \sqrt{\frac{|Z_L|}{|Z_s|}} \quad (6.41)$$

so that the (secondary) load impedance reflected to the (primary) source side (Equation (5.23)) equals the source impedance as

$$|Z_s| = \left(\frac{N_1}{N_2} \right)^2 |Z_L| \quad (6.42)$$

[Remark 6.6] Maximum Power Transfer – Impedance Matching

Maximum power transfer or impedance matching can find few applications in a power system, which is built and operated on the principle of supplying consumers with the power they need, not as much as possible. In fact, when the power dissipated in the load impedance is maximized, the power dissipated in the source impedance is also maximized. Maximum power transfer or impedance matching can find applications in audio or communication systems where the output device or the receiver is interested in extracting the maximum power without having to pay attention to the supplier.

6.8 Load Flow Calculation

Figure 6.16 shows an AC power system consisting of two nodes (buses) 1 and 2 that are connected via a line with the admittance of $y_{12} = g_{12} + jb_{12}$, where the two node voltages are $\mathbf{V}_1 = V_1 \angle \theta_1$ and $\mathbf{V}_2 = V_2 \angle \theta_2$, respectively. The complex power delivered from node 1 to node 2 is computed as

$$\begin{aligned} S_{12} &= \mathbf{V}_2 \mathbf{I}_{12}^* = \mathbf{V}_2 [(g_{12} + jb_{12})(\mathbf{V}_1 - \mathbf{V}_2)]^* = V_2 \angle \theta_2 (g_{12} - jb_{12})(V_1 \angle -\theta_1 - V_2 \angle -\theta_2) \\ &= V_2 (g_{12} V_1 \angle (\theta_2 - \theta_1) - g_{12} V_2 - jb_{12} V_1 \angle (\theta_2 - \theta_1) + jb_{12} V_2) \end{aligned} \quad (6.43)$$

If the complex power of the load at node 2 is given as $S_L = P_L + jQ_L$, the delivered power S_{12} and the load power S_L can be equated to formulate the load flow equations as

$$\begin{aligned} V_2 (V_1 [g_{12} \cos(\theta_2 - \theta_1) + b_{12} \sin(\theta_2 - \theta_1)] - V_2 g_{12}) &= P_L \\ V_2 (V_1 [g_{12} \sin(\theta_2 - \theta_1) - b_{12} \cos(\theta_2 - \theta_1)] + V_2 b_{12}) &= Q_L \end{aligned} \quad (6.44)$$

Given the values of $\mathbf{V}_1 = V_1 \angle \theta_1$ and $S_L = P_L + jQ_L$, this set of nonlinear equations can be solved for $\mathbf{V}_2 = V_2 \angle \theta_2$, where $\mathbf{V}_1 = V_1 \angle \theta_1$ is often set to $V_1 \angle 0$ as the voltage of the swing bus (node 1). Consider the following example.

(Example 6.6) Load Flow Problem for a Two-Bus Power System

Consider the power system of Figure 6.16 in which the admittance of the line is $y_{12} = g_{12} + jb_{12} = 1/z_{12} = 1/(0.1 + j)[S]$ and the complex power absorbed at node 2 is $S_L = P_L + jQ_L = 1090 \text{ W} + j218 \text{ VAR}$. Let the voltage at node 1 be fixed as $\mathbf{V}_1 = V_1 \angle \theta_1 = 110 \angle 0$. Equation (6.44) can be solved for this system by saving an M-file `f_cir06e06.m` defining Equation (6.44) and running the following MATLAB program `cir06e06.m`, which uses the nonlinear equation solver `fsolve()`. This yields $\mathbf{V}_2 = V_2 \angle \theta_2 = 106.5 \angle -5.233^\circ$.

```
%cir06e06.m
V1=110; z1=0.1+j; y1=1/z1; SL=1090+218i;
x0=[80 -0.1]; % The initial guesses of V1 and theta1
[xo,fxo]=fsolve('f_cir06e06',x0,[],V1,y1,SL); [xo(1) xo(2)*180/pi]

function f=f_cir06e06(x,V1,y1,SL)
V2=x(1); thd=x(2)-angle(V1); % V2 and the phase difference th2-th1
PL= real(SL); QL= imag(SL); g12=real(y1); b12=imag(y1);
Y=[g12 b12; -b12 g12];
f= V2*V1*Y* [cos(thd); sin(thd)] + V2^2* [-g12; b12] - [PL;QL]; % Eq. (6.44)
```

```
>>cir06e06
1.0647e+002 -5.2331e+000
```

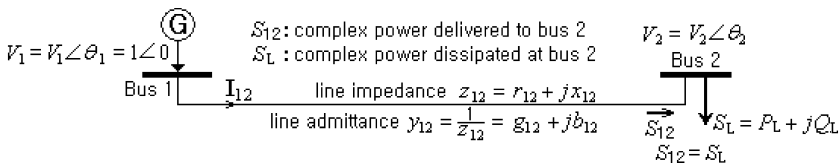


Figure 6.16 A two-bus power system

6.9 Design and Simulation for Maximum Power Transfer

The transformer discussed in the previous section is an example of the interfaces that can be inserted for maximum power transfer or impedance matching between the AC source and the load. A CL circuit and an LC circuit can also be used for the same purpose. Consider the following example.

(Example 6.7) Design of an Interface for Maximum Power Transfer – Impedance Matching

Consider the network of Figure 6.17 in which a CL circuit is inserted between the AC source $\mathbf{V}_s$ (of a radian frequency ω [rad/s]) with the source impedance R_s and the load $R_L > R_s$.

- (a) Find the values of C and L such that the maximum power transfer (impedance matching) condition (6.38) is satisfied for the series combination of R_s and C versus the parallel combination of L and R_L :

$$\frac{R_L \times j\omega L}{R_L + j\omega L} = \frac{j\omega R_L L (R_L - j\omega L)}{R_L^2 + \omega^2 L^2} = \frac{\omega^2 L^2 R_L}{R_L^2 + \omega^2 L^2} + j \frac{\omega L R_L^2}{R_L^2 + \omega^2 L^2} \equiv R_s - \frac{1}{j\omega C} \quad (\text{E6.7.1})$$

$$\text{Real part: } \frac{R_L}{R_s} \equiv \frac{R_L^2 + \omega^2 L^2}{\omega^2 L^2} = 1 + \frac{R_L^2}{\omega^2 L^2}, \quad L = \frac{R_L}{\omega \sqrt{(R_L - R_s)/R_s}} \quad (\text{E6.7.2})$$

$$\text{Imaginary part: } C = \frac{R_L^2 + \omega^2 L^2}{\omega^2 L R_L^2} = \frac{R_L L}{R_s R_L^2} = \frac{L}{R_s R_L} \stackrel{(\text{E6.7.2})}{=} \frac{1}{\omega \sqrt{(R_L - R_s)R_s}} \quad (\text{E6.7.3})$$

The following statements may be typed into the MATLAB command window to get the same result:

```
>>syms w Rs RL
>>CL= solve('w^2*L^2*RL/(RL^2+w^2*L^2)=Rs',...
            'w*L*RL^2/(RL^2+w^2*L^2)=-1/w/C', 'C,L');
>>pretty(CL.C), pretty(CL.L)

[      1      ]      [      1/2      ]
[  -----  ]      [  ( (RL - Rs) Rs)  RL  ]
[      1/2    ]      [  -----  ]
[  w ( (RL - Rs) Rs)  ]      [  (RL - Rs) w  ]
```

Note. Since Equations (E6.7.2) and (E6.7.3) are not valid for $R_L < R_s$, the CL circuit can be used only for $R_L > R_s$. What about the impedance matching of the network with $R_L < R_s$? See Problem 6.16.

- (b) With the values of L and C determined by Equations (E6.7.2) and (E6.7.3), the average power delivered to the load resistor R_L is found as

$$P_L = |\mathbf{I}|^2 R_L = \left| \frac{\mathbf{V}_s}{R_s + R_s} \frac{j\omega L}{R_L + j\omega L} \right|^2 R_L = \frac{|\mathbf{V}_s|^2 R_L}{(R_s + R_s)^2} \frac{R_s R_L^2 / (R_L - R_s)}{R_L^2 + R_s R_L^2 / (R_L - R_s)} = \frac{|\mathbf{V}_s|^2}{4 R_s} \quad (\text{E6.7.4})$$

where $\mathbf{I}$ and $\mathbf{V}_s$ are the load current and source voltage expressed in rms phasor.

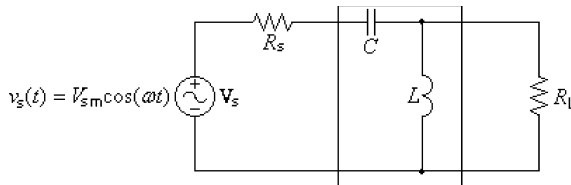


Figure 6.17 An interface for maximum power transfer – impedance matching

- (c) Let the source impedance, the load impedance, and the AC voltage source be $R_s = 8\Omega$, $R_L = 16\Omega$, and $v_s(t) = \sqrt{2}\cos(\omega t)[V]$ ($V_{\text{rms}} = 1$, $\omega = 2\pi \times 60[\text{rad/s}]$), respectively. Then the values of L and C and the average power are found as

$$L = \frac{R_L}{\omega\sqrt{(R_L - R_s)/R_s}} = 0.0424 \text{ H}, \quad C = \frac{1}{\omega\sqrt{(R_L - R_s)R_s}} = 331.57 \times 10^{-6} \text{ F} \quad (\text{E6.7.5})$$

$$P_L = \frac{|V_s|^2}{4R_s} = \frac{V_{\text{rms}}^2}{4R_s} = \frac{1}{4 \times 8} = 31.25 \text{ mW} \quad (\text{E6.7.6})$$

Alternatively, the MATLAB optimization routine `fminsearch()` can be used to find these values, which maximizes the average power (6.37) of R_L . Interested readers are invited to save the function `power_of_RL()` (presented below) in an M-file named `power_of_RL.m` and type the following statements into the MATLAB command window:

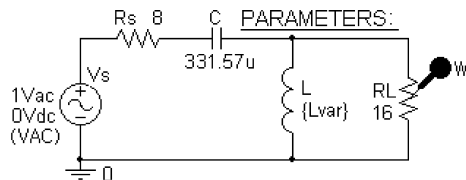
```
>>Vs=1; w=2*pi*60; Rs=8; RL=16; LC0=[1 1]; %initial values of L=1 & C=1
>>options= optimset('TolX',1e-6); % the tolerance of error in x
>>[LC,Pmin]= fminsearch('power_of_RL',LC0,options,Vs,w,Rs,RL)
```

Note. The MATLAB function `power_of_RL` computes the negative average power because the minimization routine is going to be used for maximizing the power.

```
function P_negative=power_of_RL(LC,Vs,w,Rs,RL)
L=LC(1); C=LC(2); jw=j*w; jwL=jw*L; jwC=jw*C;
Z_total=Rs+1/jwC+parallel_comb([RL jwL]);
P_negative=-abs(Vs/Z_total*jwL/(RL+jwL)).^2*RL;
```

- (d) To check whether the average power dissipated in the load resistor $R_L = 16\Omega$ is maximized by the CL interface circuit with $L = 0.0424 \text{ H}$ and $C = 331.57 \mu\text{F}$ as designed above, the PSpice simulation will be performed for the network with four values of the inductance $L = 0.01, 0.0424, 0.07$, and 0.1 H where the other parameters are fixed as $v_s(t) = \sqrt{2}\cos(\omega t)$, $R_s = 8\Omega$, $R_L = 16\Omega$, and $C = 331.57 \mu\text{F}$. Then the following steps are taken:

- Draw the schematic as depicted in Figure 6.18(a), where the value of L is set to $\{Lvar\}$ and a special part `PARAMETERS` with the part name `'PARAM'` (stored in the library `'special.olb'`) is picked up and placed.
- Double-click the part `PARAMETERS` to open its Property Editor spreadsheet (Figure 6.18(b)), in which you click the `New Column` button to open the `Add New Column` dialog box (Figure 6.18(c)), type in the `Name/Value` of a new column as `Lvar/0.01`, click `OK` to close the dialog box, and then click `x` to close the Property Editor spreadsheet.
- Double-click the part `PARAMETERS` to reopen its Property Editor spreadsheet and check whether a new column with `Lvar/0.01` has been created (see Figure 6.18(d)).
- Click the `New Simulation Profile` button to create a simulation profile with the name of, say, `AC_sweep_par` and click the `Create` button.
- Click the `Edit Simulation Settings` button to open the `Simulation Settings` dialog box (Figure 6.18(e)), in which you set the `Analysis` type to `AC Sweep/Noise` and then the `Start/End Frequencies` and `Points/Decade` to, say, `10 Hz/100 Hz` and `1000`, respectively. Then check the square box for `Parametric Sweep`, the circle for `Global parameter`, and type `'Lvar'` into the `Parameter` field. Also check the circle for `Value list` and put `[0.01 0.0424 0.07 0.1]` into the corresponding field. Then click `OK` to close the `Simulation Settings` dialog box.



(a) The PSpice schematic for Example 6.7

Property Editor						
New Column...	Apply	Display...	Delete Property	Filter	< Current properties >	Help
	Color	Primitive	Reference	Source Library	Source Part	Value
1 SCHEMATIC1:PAGE1:1	Default	DEFAULT	1	C:/ORCAD/...	PARAM: Normal	PARAM

(b) The Property Editor spreadsheet for PARAM – Parametric Sweep

Add New Column

Name:
Lvar

Value:
0.01

(c) Add New Column dialog box

	Color	Primitive	Reference	Lvar	Source Library	Source Part	Value
1 SCHEMATIC1:PAGE1:1	Default	DEFAULT	1	0.01	C:/ORCAD/...	PARAM: Normal	PARAM

(d) The Property Editor spreadsheet changed by adding a new column

Simulation Settings - ac_sweep

General
Analysis
Configuration Files
Options
Data Collection
Probe Window

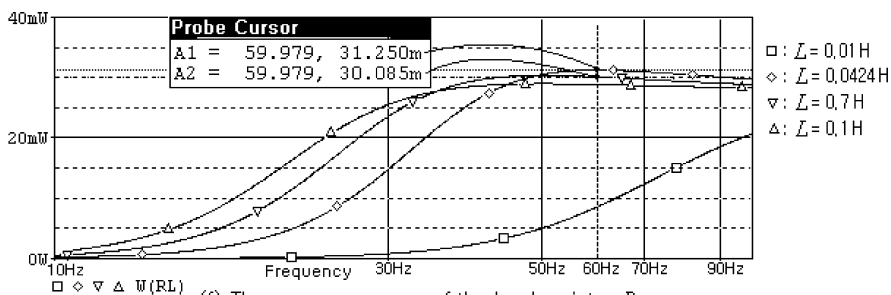
Analysis type:
Time Domain (Transient)

Options:
☒ General Settings
☐ Monte Carlo/Worst Case
☒ Parametric Sweep
☐ Temperature (Sweep)
☐ Save Bias Point
☐ Load Bias Point

Sweep variable
☐ Voltage source
☐ Current source
☒ Global parameter
☐ Model parameter
☐ Temperature
Name:
Model type:
Model:
Parameter: Lvar

Sweep type
☐ Linear
☐ Logarithmic
☒ Value list
Decade
Start value: 0.01
End value: 1.072
Increment: 0.0324
[0.01 0.0424 0.07 0.1]

(e) The Simulation Settings dialog box



(f) The average powers of the load resistor R_L for the four values of the inductance L

Figure 6.18 From the PSpice schematic to the simulation result for Example 6.7

- Click the Power Dissipation (W) Marker button on the toolbar of the Schematic Editor window and put the power probe pin into the resistor R_L .
- Click the Run button on the toolbar to make the PSpice A/D (Probe) window appear on the screen. Then you will see the graph of the four (average) power waveforms for $L = 0.01, 0.0424, 0.07, \text{ and } 0.1 \text{ H}$, as depicted in Figure 6.18(f). If the graph does not show up in the Probe window, then click the Add Trace button on the toolbar of the window to find the Add Traces dialog box (Figure H.8(a) in Appendix H), select W(RL) (the power of R_L) from the left side menu box, and click OK.
- On the Probe window having the power graphs (Figure 6.18(f)), click the Toggle Cursor button and the graphic symbol ($\diamond$) corresponding to the second one in the value list of $L = [0.01 \ 0.0424 \ 0.07 \ 0.1]$. Then click around the position of 60 Hz, press the Arrow key to move the cursor as close as possible to 60 Hz, and read the value of the power from the Probe Cursor box, which will be 31.25 mW and the maximum among the four power values. This supports the above analytical results that the average power of R_L is maximized at $L = 0.0424 \text{ H}$ and the maximized value is 31.25 mW, as obtained in Equation (E6.7.6).

Note. If you would like to see the instantaneous power waveform, you should replace the VAC part by the VSIN part (with VOFF=0, VAMPL=1.414, and FREQ=60) in the schematic, set the Analysis type to Time Domain (Transient) and the Run_to_time to, say, 50 ms in the Simulation Settings dialog box. Then click the Power (W) Marker button on the Schematic Editor window, put the power probe pin into the resistor R_L , and click Run to obtain the four instantaneous power waveforms in the Probe window.

Problems

6.1 Equivalent Impedance for Parallel-Series Combination

Consider the circuit of Figure P6.1. Find the values of the inductive/capacitive reactances X_L and X_C such that its equivalent impedance Z_{Th} is purely resistive as exactly a times the resistance R . Express them in terms of a and R .

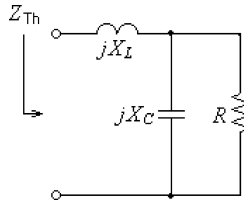


Figure P6.1

6.2 Node Analysis

Consider the AC circuit of Figure P6.2, which is driven by a sinusoidal voltage source of $v_s(t) = 10 \cos(100t) [\text{V}]$. Find the voltage $v_3(t)$ across the capacitor C_3 .

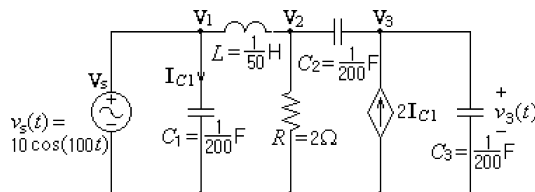


Figure P6.2

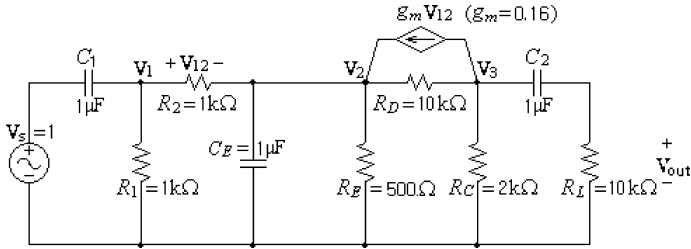


Figure P6.3

6.3 Node Analysis

Consider the circuit of Figure P6.3.

- Set up the node equation in the unknown node voltages V_1 , V_2 , and V_3 without substituting the values of the elements.
- Solve the node equation for V_1 , V_2 , and V_3 as a function of the complex variable $s = j\omega$.
- Find the output voltage $V_{out}(j\omega)$ across the load resistor R_L as a function of the complex variable $s = j\omega$.
- Plot $V_{out}(j\omega)$ for the frequency points between 10^1 and 10^3 Hz obtained by the MATLAB statements:

```
>> f=logspace(1,3,401);w=2*pi*f;
```

- Perform the PSpice simulation of the circuit to get its frequency response and compare it with that obtained in (d). Do they look similar?

Note. Could you solve this problem without using MATLAB or PSpice? This illustrates that the softwares are indispensable for engineers to survive this era of technology.

6.4 The Frequency Responses of OP Amp Circuits

- Find the frequency response $G(j\omega) = V_o/V_i(j\omega)$ of the OP Amp circuit in Figure P6.4(a).
- Find the frequency response $G(j\omega) = V_o/V_i(j\omega)$ of the OP Amp circuit in Figure P6.4(b).

6.5 The Maximum Output (Saturation or Swing) Voltage of an OP Amp Circuit

Consider the OP Amp circuit of Figure P6.5.

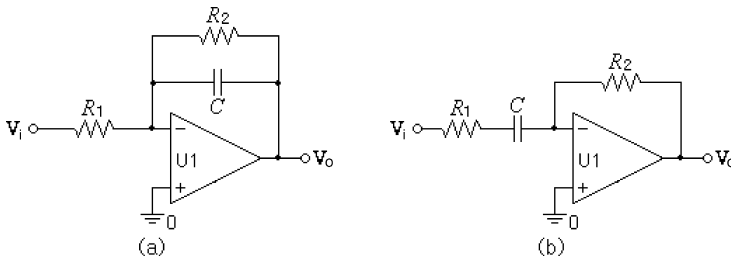


Figure P6.4

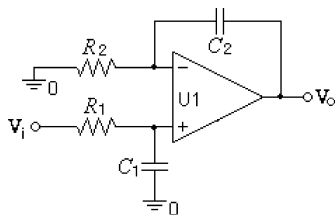


Figure P6.5

- (a) Find the expression of the transformed output voltage $V_o(s)$ in terms of the transformed input voltage $V_i(s)$.
- (b) Let $R_1 = 700\Omega$, $R_2 = 100\Omega$, $C_2 = 1\mu\text{F}$, and $v_i(t) = 73\cos(10^4t)[\text{V}]$. Verify that the amplitude of the output voltage is

$$V_{o,\max} = |V_o| = \frac{73\sqrt{2}}{\sqrt{1 + 10^8 \times 700^2 C_1^2}} \quad (\text{P6.5.1})$$

- (c) Noting that the output voltage $v_o(t)$ will be distorted from the pure sinusoidal form when its amplitude exceeds the upper/lower saturation limit voltage $\pm V_{\text{om}} = \pm 14.6\text{ V}$, find the range of capacitance C_1 such that the distortion may not take place.

6.6 Phase Difference between Currents through Impedances in Parallel

Figure P6.6 shows the main/auxiliary field coils of an electric motor driven by an AC voltage source of $v_s(t) = 120\sqrt{2}\cos(\omega t)$ with $\omega = 2\pi \times 60 = 377\text{ rad/s}$. Find the capacitance C which is to be installed in series with the auxiliary field coil L_2 in order for the current $\mathbf{I}_2$ through L_2 to lead the current $\mathbf{I}_1$ through the main field coil L_1 by $\pi/2\text{ rad} = 90^\circ$. Note that the impedances of the two field coils are

$$R_1 + j\omega L_1 = 3 + j4\Omega \quad \text{and} \quad R_2 + j\omega L_2 = 40 + j50\Omega \quad (\text{P6.6.1})$$

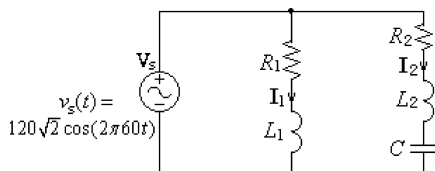


Figure P6.6

6.7 Mesh Analysis and Node Analysis

Consider the circuit of Figure P6.7.

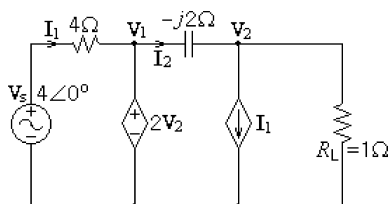


Figure P6.7

- (a) Associate the CCCS-type dependent current source with R_L and transform it into a (dependent) voltage source. Then set up the mesh equation and solve it for $\mathbf{I}_1$ and $\mathbf{I}_2$. Also find the voltage $\mathbf{V}_2$ across R_L .
- (b) Set up the node equation and solve it for $\mathbf{V}_1$ and $\mathbf{V}_2$.

6.8 Mesh Analysis of a Circuit containing Magnetically Coupled Coils

Find the current $\mathbf{I} = \mathbf{I}_1 - \mathbf{I}_2$ through the capacitor in the circuit of Figure P6.8.

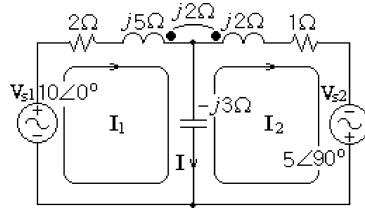


Figure P6.8

6.9 Thevenin Equivalent

- (a) For the circuit of Figure P6.9(a), find its Thevenin equivalent seen at terminals a - b .
- (b) For the circuit of Figure P6.9(b), find its Thevenin equivalent seen at terminals a - b .

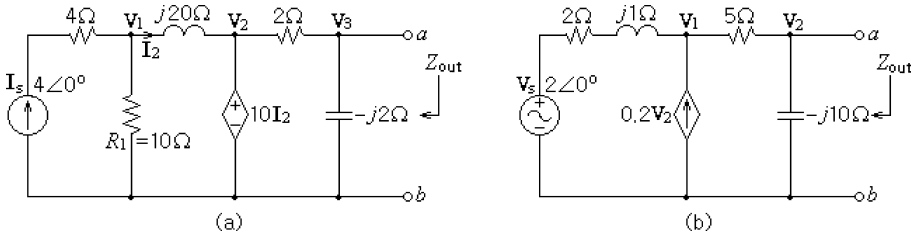


Figure P6.9

6.10 Bridge Circuits

- (a) For the bridge circuit in Figure P6.10(a), verify that the balance condition yields the following values of the unknown resistance R_x and inductance L_x :

$$R_x = \frac{R_2 R_3}{R_1}, \quad L_x = R_2 R_3 C_1 \quad (\text{P6.10.1})$$

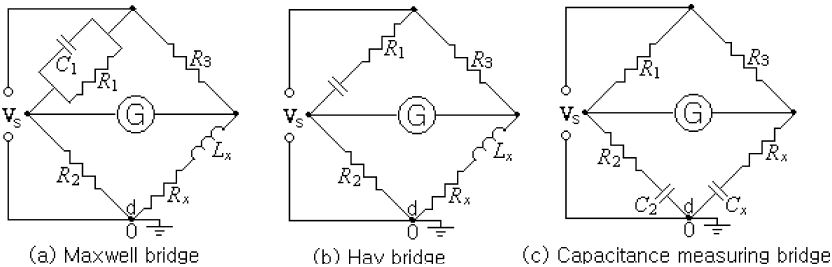


Figure P6.10

- (b) For the bridge circuit in Figure P6.10(b), verify that the balance condition yields the following values of the unknown resistance R_x and inductance L_x :

$$R_x = \frac{\omega^2 R_1 R_2 R_3 C_1^2}{1 + (\omega R_1 C_1)^2}, \quad L_x = \frac{R_2 R_3 C_1}{1 + (\omega R_1 C_1)^2} \quad (\text{P6.10.2})$$

- (c) For the bridge circuit in Figure P6.10(c), verify that the balance condition yields the following values of the unknown resistance R_x and capacitance C_x :

$$R_x = \frac{R_2 R_3}{R_1}, \quad C_x = \frac{R_1 C_2}{R_3} \quad (\text{P6.10.3})$$

6.11 Phasor Diagram for a Phase Shifter

Consider the circuit of Figure P6.11(a), which is driven by an AC voltage source.

- (a) Verify that the phasor output voltage is

$$\begin{aligned} \mathbf{V}_o &= \left[\frac{1}{2} - \frac{R_x}{R_x + 1/(j\omega C)} \right] \mathbf{V}_s = \left\{ \frac{1}{2} - \frac{R_x [R_x - 1/(j\omega C)]}{R_x^2 + [1/(\omega C)]^2} \right\} V_s \angle 0 \\ &= \frac{[1/(\omega C)]^2 - R_x^2 - j2R_x/(\omega C)}{2\{R_x^2 + [1/(\omega C)]^2\}} V_s = X + jY \end{aligned} \quad (\text{P6.11.1})$$

- (b) Verify that the real and imaginary parts of the phasor output voltage are related as

$$X^2 + Y^2 = \left(\frac{V_s}{2} \right)^2 \quad (\text{P6.11.2})$$

Note. This implies that the amplitude of the output voltage does not depend on the resistance R_x .

- (c) Verify that the sum of the phasor voltages $\mathbf{V}_{R_x}$ (across the variable resistor) and $\mathbf{V}_C$ (across the capacitor)

$$\mathbf{V}_{R_x} + \mathbf{V}_C = R_x \mathbf{I} + \frac{1}{j\omega C} \mathbf{I} = \mathbf{V}_s = V_s \angle 0 \quad (\text{P6.11.3})$$

and the phasor voltage $\mathbf{V}_a$ at node a are described by the phasor diagram in Figure P6.11(b).

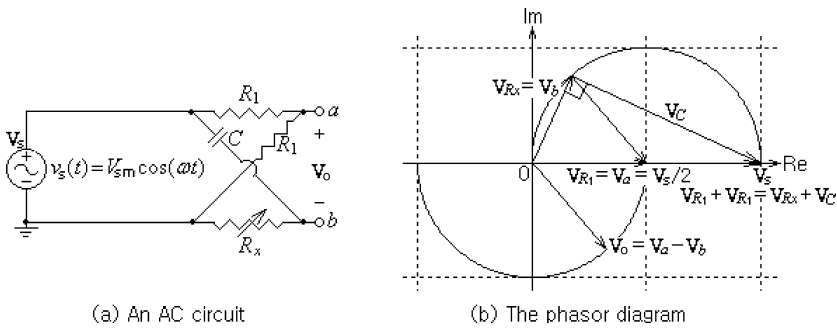


Figure P6.11

Note. Compared with the analytical description (P6.11.1) of the phasor output voltage $\mathbf{V}_o$, this phasor diagram for $\mathbf{V}_o = \mathbf{V}_a - \mathbf{V}_b = \mathbf{V}_s/2 - \mathbf{V}_{R_c}$ visually shows that the amplitude of $\mathbf{V}_o$ remains constant as the radius of the circle, $V_s/2$, but the phase of $\mathbf{V}_o$ varies from 0° to -180° as R_x varies from 0 to ∞ . In addition to the phasor diagram in Figure 6.14 for Example 6.5, this is another example showing the usefulness of the phasor diagram as a clearer description of the qualitative properties of circuits than the analytical or numerical expressions.

6.12 Open/Short-Circuit Tests to Obtain the Equivalent of a Transformer

Figure P6.12 shows an equivalent circuit model for the nonideal transformer consisting of an ideal transformer together with the series/parallel resistors and inductors. The series resistance R_1 represents the resistance of the copper winding. The series inductance L_1 , the parallel resistance R_c , and the parallel (magnetizing) inductance L_c account for the flux leakage, the core loss, and the finite permeability of the magnetic core material, respectively.

- (a) With the secondary port open, the rms voltage, the rms current, and the power are measured as $V = 120$ V, $I = 0.5$ A, and $P = 30$ W, respectively, at the primary port of the transformer, as depicted in Figure P6.12(a). Find the parallel resistance R_c and inductance L_c , assuming that the series resistance R_1 and inductance L_1 are negligibly small compared with R_c and L_c and that $\mathbf{I}_1 = \mathbf{I}_2 = 0$ A.

Hint. Noting that the (active) power P [W] is dissipated in R_c across which the AC voltage of the rms value V is applied,

$$P = \frac{V^2}{R_c} \quad (\text{P6.12.1})$$

from which R_c can be obtained. Since the apparent power VI [VA] and the active power P are known, the power triangle can be used to get the reactive power Q . Noting that the reactive power is due to L_c across which the AC voltage of the rms value V is applied, the complex power of L_c can be written as

$$jQ \stackrel{(6.28)}{=} \mathbf{V} \mathbf{I}_{L_c}^* = \mathbf{V} \left(\frac{\mathbf{V}}{j\omega L_c} \right)^* = \frac{jV^2}{\omega L_c} \quad (\text{P6.12.2})$$

which yields L_c . The magnitude of the admittance of the parallel R_c and L_c can also be used:

$$|Y_c| = \left| \frac{1}{R_c} + \frac{1}{j\omega L_c} \right| = \sqrt{\frac{1}{R_c^2} + \frac{1}{\omega^2 L_c^2}} \equiv \frac{I}{V} \quad (\text{P6.12.3})$$

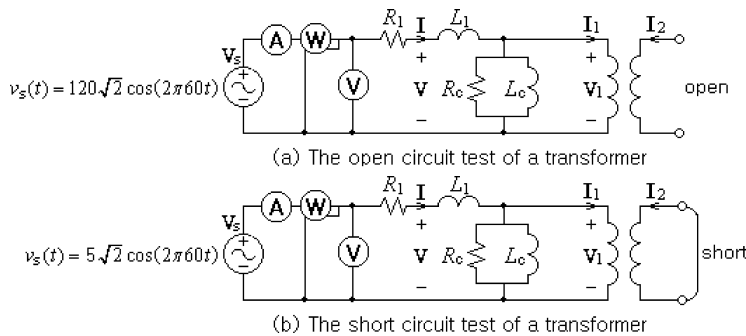


Figure P6.12

- (b) With the secondary port shorted, the rms voltage, the rms current, and the power are measured as $V = 5 \text{ V}$, $I = 5 \text{ A}$, and $P = 20 \text{ W}$, respectively, at the primary port of the transformer, as depicted in Figure P6.12(b). Find the series resistance R_1 and inductance L_1 , assuming that $V_1 = V_2 = 0 \text{ V}$.

Hint. Noting that the (active) power $P[\text{W}]$ is dissipated in R_1 through which the AC current of rms value I flows,

$$P = R_1 I^2 \quad (\text{P6.12.4})$$

from which R_1 can be obtained. Since the apparent power $VI[\text{VA}]$ and the active power P are known, the power triangle can be used to get the reactive power Q . Noting that the reactive power is due to L_1 through which the AC current of rms value I flows, the complex power of L_1 can be written as

$$jQ \stackrel{(6.28)}{=} \mathbf{V}_{L_1} \mathbf{I}^* = j\omega L_1 \mathbf{I}^* = j\omega L_1 I^2 \quad (\text{P6.12.5})$$

which yields L_1 . The magnitude of the impedance of the series R_1 and L_1 can also be used:

$$|Z_1| = |R_1 + j\omega L_1| = \sqrt{R_1^2 + (\omega L_1)^2} \equiv \frac{V}{I} \quad (\text{P6.12.6})$$

6.13 PF (Power Factor) Correction of a Power System

With reference to Example 6.5, consider the power system in Figure P6.13, where the load voltage is maintained at $\mathbf{V} = 100\angle 0^\circ$.

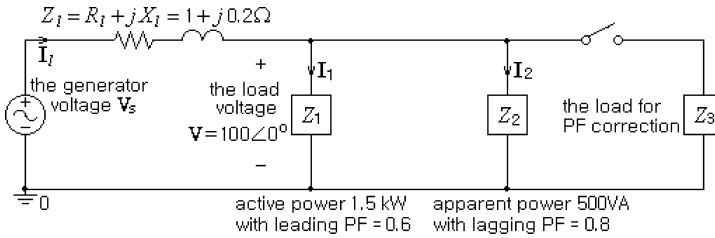


Figure P6.13

- Find the overall PF of the two load impedances Z_1 and Z_2 in parallel.
- Find the transmission power loss, the generator voltage, and the voltage regulation before the PF correction.
- Find the reactance of the third load to be connected for improving the PF to 0.95 .
- Compare the transmission power losses, the required generator voltages, and the voltage regulations before and after the PF correction.

Note. Use the MATLAB routine `PF_correction()` introduced in Section 6.6.

6.14 High Voltage Transmission and Transmission Loss

Figure P6.14 shows a power system that generates an AC voltage of 220 V , steps the voltage up to 6600 V at the transmitting site, and steps it down to 220 V at the receiving site. Assuming that the step-up/down transformers are ideal ones with no power losses, find the ratio between the magnitude of the transmission line current $\mathbf{I}_l$ and that of the load current $\mathbf{I}_L$. How do you compare the transmission losses that are obtained with and without the voltage step-up/down on the assumption that $R_l \ll R_L$ and $L_l \ll L_L$?

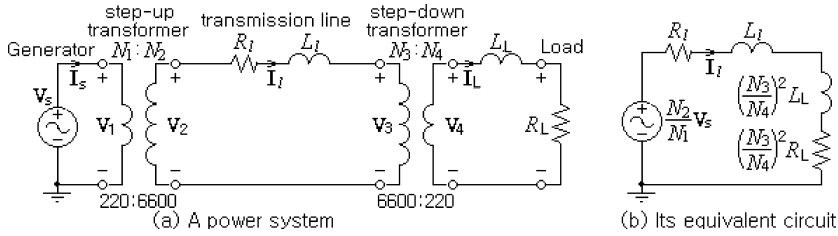


Figure P6.14

6.15 Maximum Power Transfer

- For the circuit of Figure P6.15(a), find the line impedance $R_s + j\omega L$ such that the power transferred from the source to the load R_L is maximized.
- For the circuit of Figure P6.15(b), find the capacitance C such that the power transferred from the source to the load R_L is maximized.
- For the circuit of Figure P6.15(c), find the gain K of the CCCS (current-controlled current source) such that the power transferred from the source to the load R_L is maximized.

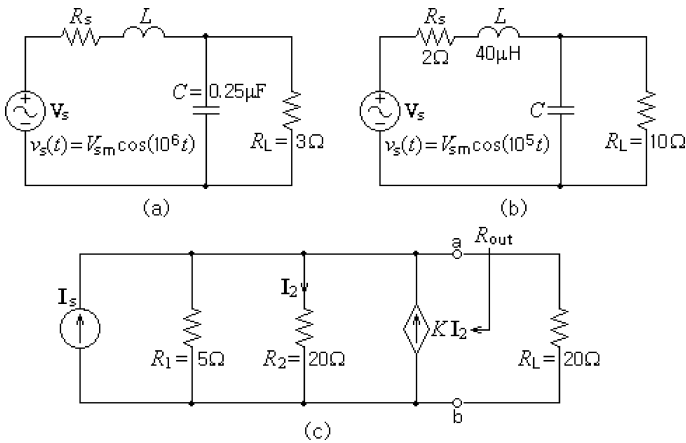


Figure P6.15

6.16 Design of a CL Interface Circuit for Impedance Matching

Consider the circuit of Figure P6.16 in which the source resistance is $R_s [\Omega]$ and the load resistance is $R_L < R_s [\Omega]$. Find the values of C and L of the CL interface such that the power transferred from the source to the load resistor R_L is maximized.

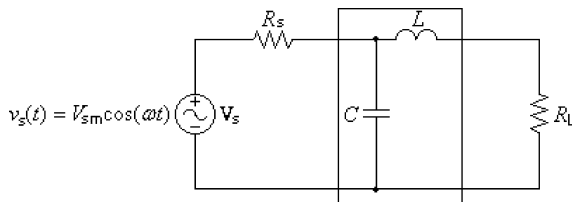


Figure P6.16

6.17 Equivalent Impedance and Reflected Impedance of a Transformer

The fourth term on the right-hand side of Equation (5.26) is the *reflected impedance*, defined to be the equivalent impedance of the secondary coil and the load reflected to the primary side of the transformer.

- (a) Verify that the AC reflected impedance of the secondary coil and the load reflected to the primary side with $s = j\omega$ and $Z_L(j\omega) = R_L + jX_L$ is

$$\begin{aligned} Z_{12}(j\omega) &= \frac{\omega^2 M^2}{R_2 + R_L + j(\omega L_2 + X_L)} \\ &= \frac{\omega^2 M^2 (R_2 + R_L)}{(R_2 + R_L)^2 + (\omega L_2 + X_L)^2} - j \frac{\omega^2 M^2 (\omega L_2 + X_L)}{(R_2 + R_L)^2 + (\omega L_2 + X_L)^2} \end{aligned} \quad (\text{P6.17.1})$$

- (b) Adding the impedance of the primary coil to the above reflected impedance makes the equivalent impedance at the terminals of the source as

$$\begin{aligned} R_1 + j\omega L_1 + Z_{12}(j\omega) &= R_1 + \frac{\omega^2 M^2 (R_2 + R_L)}{(R_2 + R_L)^2 + (\omega L_2 + X_L)^2} \\ &\quad + j\omega \left[L_1 - \frac{\omega M^2 (\omega L_2 + X_L)}{(R_2 + R_L)^2 + (\omega L_2 + X_L)^2} \right] \end{aligned} \quad (\text{P6.17.2})$$

Verify that for an ideal transformer with $L_1 L_2 = M^2$, $L_2 = \infty$, $L_1 = L_2 (N_1^2 / N_2^2)$, and $R_1 = R_2 = 0$, this AC impedance of the transformer and load seen from the primary side will become

$$\frac{\omega^2 M^2 R_L}{\omega^2 L_2^2} + j\omega \left(L_1 - \frac{\omega M^2}{\omega L_2 + X_L} \right) \simeq \frac{L_1}{L_2} (R_L + jX_L) = \frac{N_1^2}{N_2^2} Z_L \quad (\text{P6.17.3})$$

Note. This, coinciding with Equation (5.28), implies that the ideal transformer may be used to scale up or down the load impedance.

6.18 A Circuit Using a Transformer

Consider the circuit of Figure P6.18(a) in which the source parameters are $f = 60$ Hz, $R_s = 5 \Omega$, and $L_s = 2.65$ mH, the parameters of the transformer are $R_1 = 2 \Omega$, $L_1 = 95.5$ mH, $L_2 = 42.4$ mH, $R_2 = 1 \Omega$, and $k = 0.5$, and the load parameters are $R_L = 8 \Omega$ and $C = 0.106$ mF. Note that

$$j\omega L_1 \simeq j36, \quad j\omega L_2 \simeq j16, \quad j\omega M = j\omega k \sqrt{L_1 L_2} \simeq j12, \quad j\omega L_s \simeq j1, \quad -j/(\omega C) \simeq -j25 \quad (\text{P6.18.1})$$

- (a) Noting that the current $\mathbf{I}_1$ enters the primary coil from the dotted terminal while the current $\mathbf{I}_2$ enters the secondary coil from the undotted terminal, it is possible to tell whether the polarity of the voltage drop across one coil caused by its current is opposite to or the same as that of the voltage drop induced by the other coil current. Are the polarities of the mutual electromotive forces (emfs) described properly by the CCVSs (current-controlled voltage sources) in Figure P6.18(b)?

- (b) Verify that the mesh equation for this circuit is

$$\begin{bmatrix} R_s + R_1 + j\omega(L_s + L_1) & -j\omega M \\ -j\omega M & R_2 + R_L + j[\omega L_2 - 1/(\omega C)] \end{bmatrix} \begin{bmatrix} \mathbf{I}_1 \\ \mathbf{I}_2 \end{bmatrix} = \begin{bmatrix} \mathbf{V}_s \\ 0 \end{bmatrix} \quad (\text{P6.18.2})$$

- (c) Solve the mesh equation with the given values of the parameters for $\mathbf{I}_1$ and $\mathbf{I}_2$.

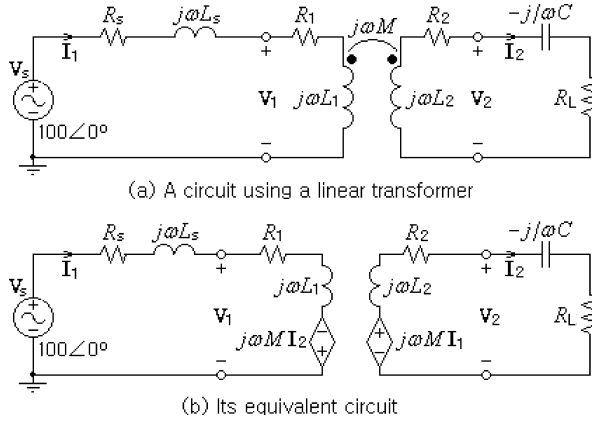


Figure P6.18

- (d) Using Equation (5.26) with $s = j\omega$ or Equation (P6.17.1) for the reflected impedance, i.e. the equivalent impedance of the secondary coil and the load reflected to the primary side of the transformer, the primary coil current can be expressed as

$$I_1 = \frac{V_s}{R_s + j\omega L_s + R_1 + j\omega L_1 + Z_{12}(j\omega)} \quad (\text{P6.18.3})$$

$$\text{with } Z_{12}(j\omega) \stackrel{(\text{P6.17.1})}{=} \frac{\omega^2 M^2}{R_2 + R_L + j(\omega L_2 + X_L)} = \frac{\omega^2 M^2}{R_2 + R_L + j[\omega L_2 - 1/(\omega C)]} \quad (\text{P6.18.4})$$

Find the primary coil current I_1 using this equation and see if it agrees with that obtained in (c).

- (e) Find the power, P_t , delivered to the transformer from the source and that, P_L , dissipated in the load. Also find the *power efficiency*, P_L/P_t , of the transformer.
- (f) Perform the PSpice simulation to find the magnitude and phase of the secondary coil current I_2 and see if it agrees with those obtained in (c). To which value do you set the coefficient of (magnetic) coupling, $k = 0.5$ or $k = -0.5$?

Three-Phase AC Circuits

While the previous chapter dealt with a single-phase (1- ϕ) AC that is transmitted through a transmission line (consisting of a pair of wires) to a load, attention now turns to a *three-phase* (3- ϕ) AC power system, in which three AC sources operate at the same frequency but with different phases. A 3- ϕ AC power system has the following advantages over a 1- ϕ AC power system:

1. The instantaneous power delivered to a load fluctuates much less in a polyphase AC power system than in a single-phase AC power system. Especially when it is used in rotating machinery like motors, the torque on the rotor pulsates much less than in a single-phase AC power system.
2. It can deliver the same power with appreciably less conductors and components than a single-phase AC power system. That is why almost all electric power in the world is generated, transmitted, and distributed in the form of three-phase AC (at 50 or 60 Hz) throughout the world.

In one example a 3- ϕ AC power system will be solved by using MATLAB and PSpice.

7.1 Balanced Three-Phase Voltages

It may be helpful in understanding three-phase AC circuits to see the rough structure and principle of a three-phase AC generator such as the ones illustrated in Figure 7.1 or Reference [W-9]. Both of the two three-phase generators consist of a rotor, a stator, and three separate armature coils with terminals a - a' , b - b' , and c - c' that are placed 120° apart around the rotor (Figure 7.1(a)) or the stator (Figure 7.1(b)). Since each armature coil has a flux linkage of $\lambda(t) = N\phi_m \sin(\omega t + \theta)$ (N = the number of windings in an armature coil, ϕ_m = the flux produced by the magnet, ω = the angular velocity of the rotor, t = the time, and θ = the initial angular position of the rotor) depending on its angular position relative to the stator, the induced voltage between its terminals is

$$v(t) = \frac{d}{dt}\lambda(t) = \frac{d}{dt}[N\phi_m \sin(\omega t + \theta)] = N\phi_m \omega \cos(\omega t + \theta) = V_m \cos(\omega t + \theta)$$

Thus, depending on the relative position of the three coils and the rotating direction of the rotor, the three induced voltages across the armature coils between terminals a - a' , b - b' , and c - c' can be written as follows:

Positive (abc) sequence

$$v_a(t) = V_m \cos(\omega t), \quad v_b(t) = V_m \cos(\omega t - 120^\circ), \quad v_c(t) = V_m \cos(\omega t + 120^\circ) \quad (7.1a)$$

$$\begin{aligned} \mathbf{V}_a &= V_m \angle 0^\circ, & \mathbf{V}_b &= V_m \angle -120^\circ, & \mathbf{V}_c &= V_m \angle +120^\circ \\ (a\text{-phase voltage}) & & (b\text{-phase voltage}) & & (c\text{-phase voltage}) \end{aligned} \quad (7.1b)$$

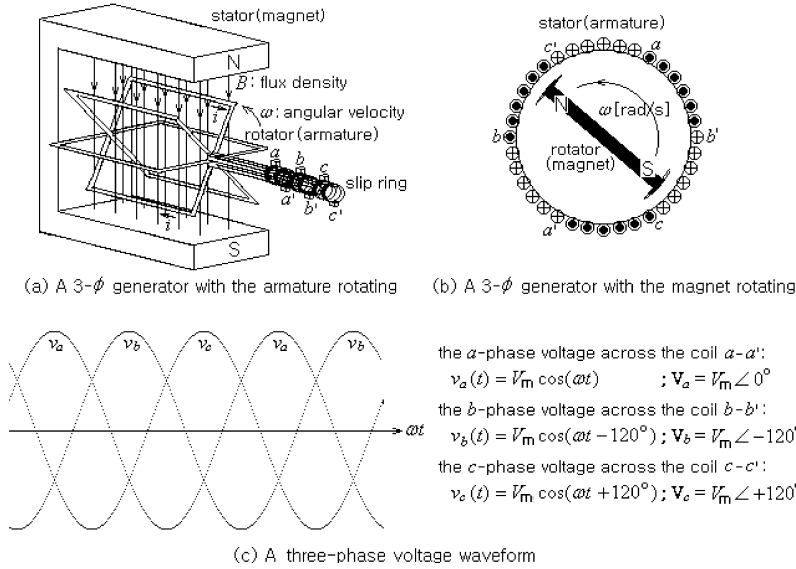


Figure 7.1 Basic structure and principle of an AC generator

Negative (acb) sequence

$$v_a(t) = V_m \cos(\omega t), \quad v_b(t) = V_m \cos(\omega t + 120^\circ), \quad v_c(t) = V_m \cos(\omega t - 120^\circ) \quad (7.2a)$$

$$\mathbf{V}_a = V_m \angle 0^\circ, \quad \mathbf{V}_b = V_m \angle +120^\circ, \quad \mathbf{V}_c = V_m \angle -120^\circ \quad (7.2b)$$

Figure 7.1(c) shows a typical set of three-phase voltage waveforms in the positive (abc) sequence, which can be produced by a three-phase AC generator.

In order to operate more than one three-phase AC generators in parallel, their phase sequences should be the same, positive (abc) or negative (acb). Such a set of three AC voltages as these is said to be *balanced* because they all have the same frequency and magnitude but are out of phase with each other by 120° .

An important feature of balanced three-phase voltages is that their sum is zero:

$$\mathbf{V}_a + \mathbf{V}_b + \mathbf{V}_c = 0 \quad (7.3)$$

This zero-sum property can also be shown in the time domain:

$$v_a(t) + v_b(t) + v_c(t) = 0 \quad (7.4)$$

$$\begin{aligned} & V_m \cos(\omega t) + V_m \cos(\omega t - 120^\circ) + V_m \cos(\omega t + 120^\circ) \\ & \stackrel{(F.6)}{=} V_m \cos(\omega t) + V_m [\cos(\omega t) \cos(120^\circ) + \sin(\omega t) \sin(120^\circ)] \\ & \quad + V_m [\cos(\omega t) \cos(120^\circ) - \sin(\omega t) \sin(120^\circ)] \\ & = V_m \cos(\omega t) [1 + 2 \cos(120^\circ)] = V_m \cos(\omega t) [1 + 2 \times (-1/2)] = 0 \end{aligned}$$

The output voltage/current level of a three-phase generator depends on which connection of the three armature coils is made, Y (*wye*)-*connection* or Δ (*delta*)-*connection*. The two connection diagrams of a

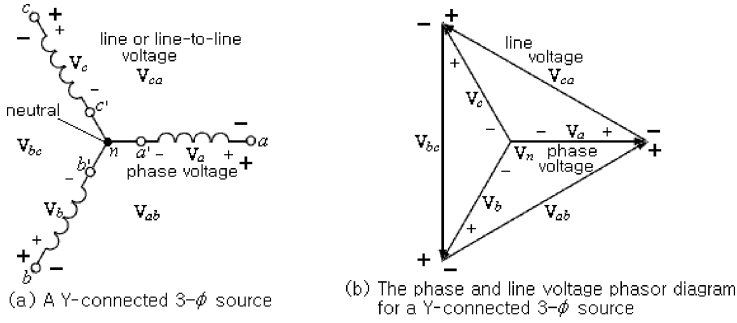


Figure 7.2 A Y-connected 3- ϕ source and its voltage phasor diagram

three-phase source together with the corresponding voltage/current phasor diagrams are depicted in Figures 7.2 and 7.3, respectively. Especially the Y-connected three-phase source (Figure 7.2(a)) has a common terminal of the three coils, labeled n , which is called the *neutral* (node or terminal) of the source.

Figure 7.2(b) shows the voltage phasor diagram of a Y-connected balanced three-phase source, where the a -phase voltage $\mathbf{V}_a$ is regarded as the reference with the phase angle of 0° . From this phasor diagram, the relationship between the *phase* (or *line-to-neutral*) voltages and the *line* (or *line-to-line*) voltages can be written as follows:

$$\mathbf{V}_{ab} = \mathbf{V}_a - \mathbf{V}_b = \sqrt{3} V_Y \angle 30^\circ \quad (7.5a)$$

$$\mathbf{V}_{bc} = \mathbf{V}_b - \mathbf{V}_c = \sqrt{3} V_Y \angle -90^\circ \quad (7.5b)$$

$$\mathbf{V}_{ca} = \mathbf{V}_c - \mathbf{V}_a = \sqrt{3} V_Y \angle +150^\circ \quad (7.5c)$$

Note that for a Y-connected three-phase source, the amplitudes of line voltages are $\sqrt{3}$ times that of phase voltages and the line current is the same as the phase current:

$$V_l = \sqrt{3} V_Y \quad \text{and} \quad \mathbf{I}_l = \mathbf{I}_Y \quad (7.5d)$$

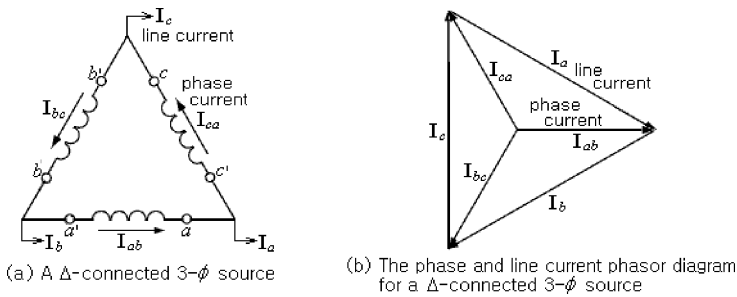


Figure 7.3 A Δ -connected 3- ϕ source and its current phasor diagram

Figure 7.3(b) shows the current phasor diagram of a Δ -connected balanced three-phase source, where the a -phase current ($\mathbf{I}_{ab}$) is regarded as the reference with the phase angle of 0° . From this phasor diagram, the relationship between the *phase currents* and the *line currents* can be written as follows:

$$\mathbf{I}_a = \mathbf{I}_{ab} - \mathbf{I}_{ca} = \sqrt{3} I_{\Delta} \angle -30^\circ \quad (7.6a)$$

$$\mathbf{I}_b = \mathbf{I}_{bc} - \mathbf{I}_{ab} = \sqrt{3} I_{\Delta} \angle -150^\circ \quad (7.6b)$$

$$\mathbf{I}_c = \mathbf{I}_{ca} - \mathbf{I}_{bc} = \sqrt{3} I_{\Delta} \angle +90^\circ \quad (7.6c)$$

Note that for a Δ -connected three-phase source, the amplitudes of line currents are $\sqrt{3}$ times that of phase currents and the line voltage is the same as the phase voltage:

$$I_l = \sqrt{3} I_{\Delta} \quad \text{and} \quad \mathbf{V}_l = \mathbf{V}_{\Delta} \quad (7.6d)$$

Since not only three-phase sources but also three-phase loads can be either Y-connected or Δ -connected, there are four possible configurations of a three-phase power system: Y-Y, Y- Δ , Δ -Y, and Δ - Δ . However, Δ -connected three-phase sources are seldom used in practice because they make a loop of voltage sources, which may have a large *circulating current* if the three-phase voltages are not exactly balanced and therefore do not sum to zero. For example, a slightly unbalanced three-phase voltage source of

$$\mathbf{V}_{ab} = 207.9 \angle 29^\circ, \quad \mathbf{V}_{bc} = 208.1 \angle -90^\circ, \quad \text{and} \quad \mathbf{V}_{ca} = 208 \angle +151^\circ$$

has an emf (electromotive force) around the Δ -loop inside the generator as

$$\begin{aligned} \mathbf{V}_{ab} + \mathbf{V}_{bc} + \mathbf{V}_{ca} &= 207.9(0.8746 + j0.4848) - j208.1 + 208(-0.8746 + j0.4848) \\ &= -0.0875 - j6.47 \text{ V} \end{aligned}$$

Since the impedance of the windings (coils) of a generator is very small, even this low emf may result in a large circulating current that will heat the Δ -connected generator, making its efficiency and life suffer.

7.2 Power of Balanced Three-Phase Loads

A three-phase load is said to be *balanced* if the three impedance legs are all the same. In this section we will find the power of a balanced three-phase load to which a balanced three-phase voltage source with an rms line voltage V_l and an rms line current I_l is applied:

Power of a Y-connected balanced three-phase load

$$\text{Active power:} \quad P_{Y,\text{Total}} = 3P_Y = 3 V_Y I_Y \cos \theta \stackrel{(7.5d)}{=} 3 \frac{V_l}{\sqrt{3}} I_l \cos \theta = \sqrt{3} V_l I_l \cos \theta \quad (7.7a)$$

$$\text{Reactive power:} \quad Q_{Y,\text{Total}} = 3Q_Y = 3 V_Y I_Y \sin \theta = \sqrt{3} V_l I_l \sin \theta \quad (7.7b)$$

$$\begin{aligned} \text{Complex power:} \quad S_{Y,\text{Total}} &= 3S_Y = 3 \mathbf{V}_Y \mathbf{I}_Y^* = \sqrt{3} V_l I_l \angle \theta = P_{Y,\text{Total}} + jQ_{Y,\text{Total}} \\ &\text{with } \theta = \text{phase angle or power factor angle of the load} \end{aligned} \quad (7.7c)$$

Power of a Δ -connected balanced three-phase load

$$\text{Active power:} \quad P_{\Delta,\text{Total}} = 3P_{\Delta} = 3 V_{\Delta} I_{\Delta} \cos \theta \stackrel{(7.6d)}{=} 3 V_l \frac{I_l}{\sqrt{3}} \cos \theta = \sqrt{3} V_l I_l \cos \theta \quad (7.8a)$$

$$\text{Reactive power:} \quad Q_{\Delta,\text{Total}} = 3Q_{\Delta} = 3 V_{\Delta} I_{\Delta} \sin \theta = \sqrt{3} V_l I_l \sin \theta \quad (7.8b)$$

$$\text{Complex power:} \quad S_{\Delta,\text{Total}} = 3S_{\Delta} = 3 \mathbf{V}_{\Delta} \mathbf{I}_{\Delta}^* = \sqrt{3} V_l I_l \angle \theta = P_{Y,\text{Total}} + jQ_{Y,\text{Total}} \quad (7.8c)$$

Note. It is implied that regardless of whether a balanced three-phase source/load is Y-connected or Δ -connected, the power transferred from the source to the load is determined by the line (or line-to-line) voltage and the line current.

[Remark 7.1] Instantaneous Power of a 1- ϕ System and a 3- ϕ System

Referring to Section 6.5, the instantaneous power of a load (with the impedance angle of θ) supplied by a single-phase AC source (with the rms values of its terminal voltage and current given as V and I , respectively) varies with time:

$$p(t) \stackrel{(6.25a)}{=} VI [\cos \theta + \cos(2\omega t + 2\theta_v - \theta)] \quad (7.9)$$

By contrast, the instantaneous power of a balanced three-phase load supplied by a balanced three-phase AC source (with the rms values of its phase voltage and current given as V_ϕ and I_ϕ , respectively) does not vary with time, producing a constant torque for a three-phase AC motor:

$$\begin{aligned} p_{3\phi}(t) &= v_a(t)i_a(t) + v_b(t)i_b(t) + v_c(t)i_c(t) \\ &= V_{\phi m} \cos(\omega t) I_{\phi m} \cos(\omega t - \theta) + V_{\phi m} \cos(\omega t - 120^\circ) I_{\phi m} \cos(\omega t - 120^\circ - \theta) \\ &\quad + V_{\phi m} \cos(\omega t + 120^\circ) I_{\phi m} \cos(\omega t + 120^\circ - \theta) \\ &= V_\phi I_\phi [\cos \theta + \cos(2\omega t - \theta)] + V_\phi I_\phi [\cos \theta + \cos(2\omega t - 240^\circ - \theta)] \\ &\quad + V_\phi I_\phi [\cos \theta + \cos(2\omega t + 240^\circ - \theta)] \\ &= 3V_\phi I_\phi \cos \theta : \text{constant} \end{aligned} \quad (7.10)$$

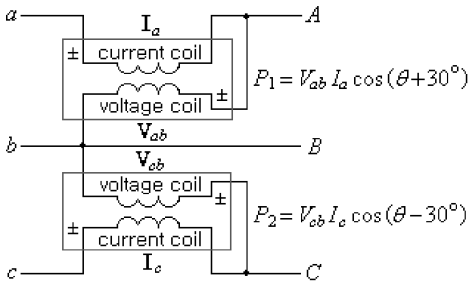
7.3 Measurement of Three-Phase Power

It is no wonder that three wattmeters are used to measure a three-phase power. However, it may be surprising that a three-phase power can be measured using just two wattmeters. Figure 7.4(a) shows how to connect two wattmeters for measuring a three-phase power, where their readings are written as

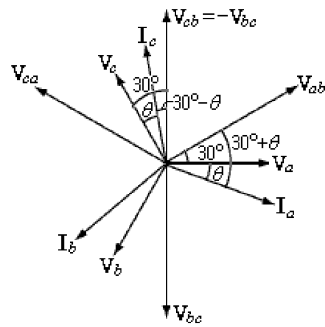
$$\begin{aligned} P_1 &= V_{ab} I_a \cos(\theta + 30^\circ) \\ P_2 &= V_{cb} I_c \cos(\theta - 30^\circ) \end{aligned} \quad (7.11)$$

from the phasor diagram in Figure 7.4(b). The sum of these two readings yields the three-phase active power:

$$\begin{aligned} P_1 + P_2 &= V_{ab} I_a \cos(\theta + 30^\circ) + V_{cb} I_c \cos(\theta - 30^\circ) \\ &= V_L I_L \cos(\theta + 30^\circ) + V_L I_L \cos(\theta - 30^\circ) \\ &\stackrel{(F.6)}{=} 2V_L I_L \cos \theta \cos 30^\circ = \sqrt{3} V_L I_L \cos \theta \stackrel{(7.7a), (7.8a)}{=} P_{\text{Total}} \end{aligned} \quad (7.12)$$



(a) Two-wattmeter connection to measure a three-phase AC power



(b) The phasor diagrams for voltages/currents

Figure 7.4 A circuit and its phasor diagram for the two-wattmeter method to measure a 3- ϕ power

On the other hand, the difference between these two readings turns out to be

$$\begin{aligned} P_2 - P_1 &= V_l I_l \cos(\theta - 30^\circ) - V_l I_l \cos(\theta + 30^\circ) \\ &\stackrel{(F.6)}{=} 2V_l I_l \sin \theta \sin 30^\circ = V_l I_l \sin \theta \stackrel{(7.7b),(7.8b)}{=} Q_{\text{Total}} / \sqrt{3} \end{aligned} \quad (7.13)$$

Thus this can be multiplied by $\sqrt{3}$ to get the three-phase reactive power Q_{Total} and, furthermore, use can be made of the ratio of the reactive power to the active power to obtain the power factor as

$$PF = \cos \theta \stackrel{\text{Fig. 6.12}}{=} \cos \left(\tan^{-1} \frac{Q_{\text{Total}}}{P_{\text{Total}}} \right) \stackrel{(7.12),(7.13)}{=} \cos \left[\tan^{-1} \frac{\sqrt{3}(P_2 - P_1)}{P_1 + P_2} \right] \quad (7.14)$$

7.4 Three-Phase Power System

As discussed in Section 7.1, there are four possible configurations (Y-Y, Y- Δ , Δ -Y, and Δ - Δ) of three-phase power systems. In this section, only the Y-Y connection will be examined since Δ -connected three-phase sources are seldom used in practice due to the circulating current problem and Δ -connected loads can easily be converted into their equivalent Y-connected ones (see Table 6.1 in Section 6.4).

Figure 7.5 shows a Y-Y three-phase power system, where the neutral line connecting the two neutrals of the source and the load is shown as a dotted line to denote that it is dispensable in principle for a balanced system because no current will flow through it. The three-phase power system with no neutral line is denoted by ‘3 ϕ -3w’ and one having a neutral line by ‘3 ϕ -4w’. To analyze this power system, the source-side neutral is set as the reference node (having zero potential) and KCL is applied to the load-side neutral N to write the node equation as

$$\frac{\mathbf{V}_a - \mathbf{V}_N}{Z_A} + \frac{\mathbf{V}_b - \mathbf{V}_N}{Z_B} + \frac{\mathbf{V}_c - \mathbf{V}_N}{Z_C} = \frac{\mathbf{V}_N}{Z_{nl}} \quad (7.15)$$

where $Z_A = Z_{al} + Z_{AL}$, $Z_B = Z_{bl} + Z_{BL}$, and $Z_C = Z_{cl} + Z_{CL}$ are the sums of each line impedance and load impedance per phase. This equation can be solved to obtain the load side neutral voltage as

$$\mathbf{V}_N = \frac{\mathbf{V}_a/Z_A + \mathbf{V}_b/Z_B + \mathbf{V}_c/Z_C}{1/Z_A + 1/Z_B + 1/Z_C + 1/Z_{nl}} \quad (7.16)$$

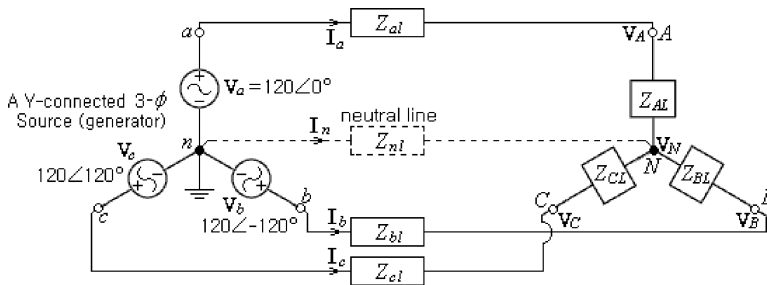


Figure 7.5 The Y-Y configuration of a three-phase power system

and, furthermore, the line currents (identical to the phase currents for a Y-connection) and the load-side (receiving end) voltages as

$$\mathbf{I}_a = \frac{\mathbf{V}_a - \mathbf{V}_N}{Z_A}, \quad \mathbf{I}_b = \frac{\mathbf{V}_b - \mathbf{V}_N}{Z_B}, \quad \mathbf{I}_c = \frac{\mathbf{V}_c - \mathbf{V}_N}{Z_C}, \quad \mathbf{I}_n = -\frac{\mathbf{V}_N}{Z_{nl}} \quad (7.17)$$

$$\mathbf{V}_A = \mathbf{V}_a - Z_{al}\mathbf{I}_a, \quad \mathbf{V}_B = \mathbf{V}_b - Z_{bl}\mathbf{I}_b, \quad \mathbf{V}_C = \mathbf{V}_c - Z_{cl}\mathbf{I}_c \quad (7.18)$$

Note. With the mesh analysis, a set of two or three equations would have to be solved for $\mathbf{I}_a$, $\mathbf{I}_b$, and $\mathbf{I}_c$.

If the three-phase system is balanced with identical line impedances and load impedances

$$Z_A = Z_B = Z_C = Z \quad (7.19)$$

Equation (7.16) for the load-side neutral voltage becomes

$$\mathbf{V}_N = \frac{\mathbf{V}_a/Z + \mathbf{V}_b/Z + \mathbf{V}_c/Z}{1/Z + 1/Z + 1/Z + 1/Z_{nl}} = \frac{\mathbf{V}_a + \mathbf{V}_b + \mathbf{V}_c}{3 + Z/Z_{nl}} \stackrel{(7.3)}{=} 0 \quad (7.20)$$

so that Equations (7.17) and (7.18) become

$$\mathbf{I}_a = \frac{\mathbf{V}_a}{Z}, \quad \mathbf{I}_b = \frac{\mathbf{V}_b}{Z} = \mathbf{I}_a \angle -120^\circ, \quad \mathbf{I}_c = \frac{\mathbf{V}_c}{Z} = \mathbf{I}_a \angle +120^\circ, \quad \mathbf{I}_n = -\frac{\mathbf{V}_N}{Z_{nl}} = 0 \quad (7.21)$$

$$\mathbf{V}_A = Z_L \mathbf{I}_a, \quad \mathbf{V}_B = Z_L \mathbf{I}_b = \mathbf{V}_A \angle -120^\circ, \quad \mathbf{V}_C = Z_L \mathbf{I}_c = \mathbf{V}_A \angle +120^\circ \quad (7.22)$$

```
function [VN,VABC,Iabc,SABC]=y_y(Vabc,ZABCL,Zabcl)
% To solve a Y-Y connected 3-phase AC system (stored in an M-file "y_y.m")
%Input: Vabc=[Va Vb Vc]: the three phase voltage sources
%       ZABCL=[ZAL ZBL ZCL]: the three phase load impedances
%       Zabcl=[Zal Zbl Zcl Znl]: the three or four line impedances
%       optionally with Znl= the neutral line impedance
%Output: VN= the load side neutral voltage
%       VABC=[VA;VB;VC]= the three load-side voltages
%       Iabc=[Ia;Ib;Ic]= the three line currents
%       SABC=[SA;SB;SC]= the 3-phase complex powers
% Copyleft: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only
Vabc=Vabc(:); ZABCL=ZABCL(:); Zabcl=Zabcl(:); ZABCN=Zabcl(1:3)+ZABCL;
tmp = sum(1./ZABCN);
if length(Zabcl)>3, tmp=tmp+1/Zabcl(4); end % with a neutral line if any
VN = sum(Vabc./ZABCN)/tmp; % Load side neutral voltage - Eq. (7.16)
Iabc = (Vabc-VN)./ZABCN; % 3 Line currents - Eq. (7.17)
VABC = Vabc - Zabcl(1:3).*Iabc; % 3 Load end voltages - Eq. (7.18)
% Complex power S=VI* based on the rms phasor voltages/currents
SABC = (VABC-VN).*conj(Iabc); % Complex power of load - Eq. (6.28):
disp('Neutral Voltage (Mag&Phase) at Load side=')
disp([abs(VN) angle(VN)*180/pi])
disp('Load end voltages (Mag&Phase) Line currents (Mag&Phase) Complex powers')
disp([abs(VABCN) angle(VABCN)*180/pi abs(Iabc) angle(Iabc)*180/pi SABC])
```

This result implies that the voltages/currents for the three phases have the same magnitude, but differing from each other in phase angle by 120° and, consequently, the three-phase system needs to be solved only for one phase in the same way as for a single-phase system. Another implication is that in the case of a balanced three-phase system, it makes no difference whether the neutral line exists or not since no current will flow through it. Even in the case of a (slightly) unbalanced three-phase system, the current through the neutral line is expected to be much smaller than the hot-line current. That is why a thinner and cheaper wire is used as the neutral line.

A three-phase system is efficient in the sense that it requires fewer conductors to handle the same power as three separate single-phase systems. However, the solution formulas (7.16) to (7.18) are difficult to compute by hand. That is why the following MATLAB routine `y_y ( )` is introduced, which can be used to solve a Y-Y connected three-phase system. A user is supposed to put the impedance of the neutral line as the fourth element of the third-input argument (`Zabcl`) only when it exists, as in the case of the three-phase four-wire (3ϕ -4w) power system.

Figure 7.6 shows a Y- Δ /Y configuration of 3ϕ -4w three-phase power system, where the load consists of a Y-connected one and a Δ -connected one. If there is no neutral line connected between the source-side neutral n and the load-side neutral N , virtually for a 3ϕ -3w system, the following steps could be taken to convert the power system into a Y-Y configuration:

1. Make the Y- Δ conversion of the Y-connected load to get the equivalent Δ -connection.
2. Make the parallel combination of the equivalent Δ -connection and the original Δ -connection.
3. Make the Δ -Y conversion of the composite Δ -connection to obtain the composite Y-connected load.

Note. Why not make a straight Δ -Y conversion of the Δ -connected one and then make a parallel combination of the two Y-connected loads? The neutral of the originally Y-connected one and the resulting neutral of the Y-connected one converted from the Δ -connected one do not generally match each other in the case of unbalanced Y-connected and Δ -connected loads. However, either will do in the case of balanced Y-connected and Δ -connected loads.

4. Apply the Y-Y system analysis implemented by the MATLAB routine `y_y ( )` as if there were only a Y-connected load.

This approach is, however, not applicable for a 3ϕ -4w system with a neutral line, for which a set of node equations should be written in the four unknown voltage variables $\mathbf{V}_A$, $\mathbf{V}_B$, $\mathbf{V}_C$, and $\mathbf{V}_N$:

$$\begin{bmatrix} Y_{al} + Y_{AN} + Y_{AB} + Y_{CA} & -Y_{AB} & -Y_{CA} & -Y_{AN} \\ -Y_{AB} & Y_{bl} + Y_{BN} + Y_{AB} + Y_{BC} & -Y_{BC} & -Y_{BN} \\ -Y_{CA} & -Y_{BC} & Y_{cl} + Y_{CN} + Y_{CA} + Y_{BC} & -Y_{CN} \\ -Y_{AN} & -Y_{BN} & -Y_{CN} & Y_{AN} + Y_{BN} + Y_{CN} + Y_{nl} \end{bmatrix} \times \begin{bmatrix} \mathbf{V}_A \\ \mathbf{V}_B \\ \mathbf{V}_C \\ \mathbf{V}_N \end{bmatrix} = \begin{bmatrix} Y_{al} \mathbf{V}_a \\ Y_{bl} \mathbf{V}_b \\ Y_{cl} \mathbf{V}_c \\ 0 \end{bmatrix} \quad (7.23)$$

After solving this set of equations for $\mathbf{V}_A$, $\mathbf{V}_B$, $\mathbf{V}_C$, and $\mathbf{V}_N$, the line currents can be obtained as

$$\mathbf{I}_a = \frac{\mathbf{V}_a - \mathbf{V}_A}{Z_{al}}, \quad \mathbf{I}_b = \frac{\mathbf{V}_b - \mathbf{V}_B}{Z_{bl}}, \quad \mathbf{I}_c = \frac{\mathbf{V}_c - \mathbf{V}_C}{Z_{cl}}, \quad \mathbf{I}_n = -\frac{\mathbf{V}_N}{Z_{nl}} \quad (7.24)$$

Note. With the mesh analysis, a set of six equations would need to be solved.

However, Equation (7.23) is formidable to compute by hand and thus the following MATLAB routine `y_dy ( )` is introduced, which can be used to solve a Y- Δ /Y-connected three-phase system like the one depicted in Figure 7.6.

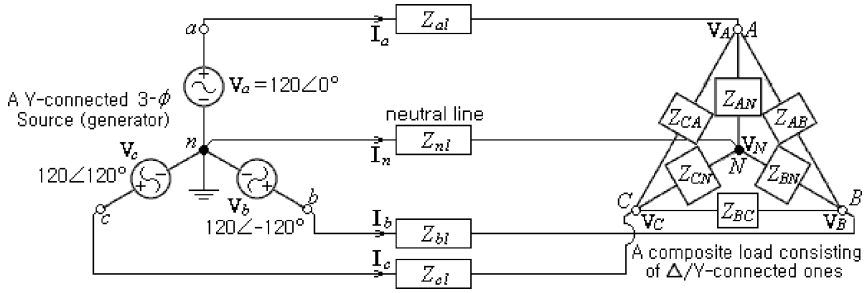


Figure 7.6 The Y-Δ/Y configuration of a three-phase power system

```
function[VN,VABC,Iabc,S_total]=y_dy(Vabc,ZABCN,ZABC,Zabcl)
% To solve a 3p-4w system with Delta/Y-connected loads
%Input: Vabc=[Va Vb Vc]: the three phase voltage sources
%       ZABCN=[ZAN ZBN ZCN]: the Y-connected three phase load impedances
%       ZABC=[ZAB ZBC ZCA]: the Delta-connected three phase load impedances
%       Zabcl=[Zal Zbl Zcl Znl]: the three or four line impedances
%       optionally with Znl= the neutral line impedance
%Output: VN= the load side neutral voltage
%        VABC=[VA;VB;VC]= the load-side end voltages
%        Iabc=[Ia;Ib;Ic]= the three line currents
%        S_total= the total 3-phase complex power
% Copyleft: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only
YABCN=1./ZABCN; YABC=1./ZABC; Yabcl=1./Zabcl;
if length(Zabcl)>3, Ynl=Yabcl(4); Yabcl=Yabcl(1:3); else Ynl=0; end
Va=Vabc(1); Vb=Vabc(2); Vc=Vabc(3); % Voltages at the sending end
YAN=YABCN(1); YBN=YABCN(2); YCN=YABCN(3); % each of Y-connected admittances
YAB=YABC(1); YBC=YABC(2); YCA=YABC(3); % each of Delta-connected admittances
Yal=Yabcl(1); Ybl=Yabcl(2); Ycl=Yabcl(3); % each line admittance
Y=[Yal+YAN+YAB+YCA -YAB -YCA -YAN; -YAB Ybl+YBN+YAB+YBC -YBC -YBN;
   -YCA -YBC Ycl+YCN+YBC+YCA -YCN; -YAN -YBN -YCN YAN+YBN+YCN+Ynl];
VABCN=Y\Yal*Va; Ybl*Vb; Ycl*Vc; 0]; % Eq. (7.23)
VABC=VABCN(1:3); VN=VABCN(4); Iabc=Yabc(:).*(Vabc(:)-VABC); % Eq. (7.24)
% Complex power S=VI* based on the rms phasor voltages/currents
S_Y=((VABC-VN)./ZABCN(:))'* (VABC-VN) % Complex power of Y-load: Eq. (6.28)
V=[VABC(1)-VABC(2) VABC(2)-VABC(3) VABC(3)-VABC(1)]; % Line-to-line voltages
S_D=V*(V./ZABC)' % Complex power of Delta-load
% The sum of complex powers for the Y-connected and Delta-connected loads
disp('Total complex power'), S_total=S_Y+S_D
disp('Neutral Voltage (Mag&Phase) at Load side=')
disp([abs(VN) angle(VN)*180/pi])
disp('Phase voltages (Mag&Phase) Line currents (Mag&Phase)')
disp([abs(VABC) angle(VABC)*180/pi abs(Iabc) angle(Iabc)*180/pi])
```

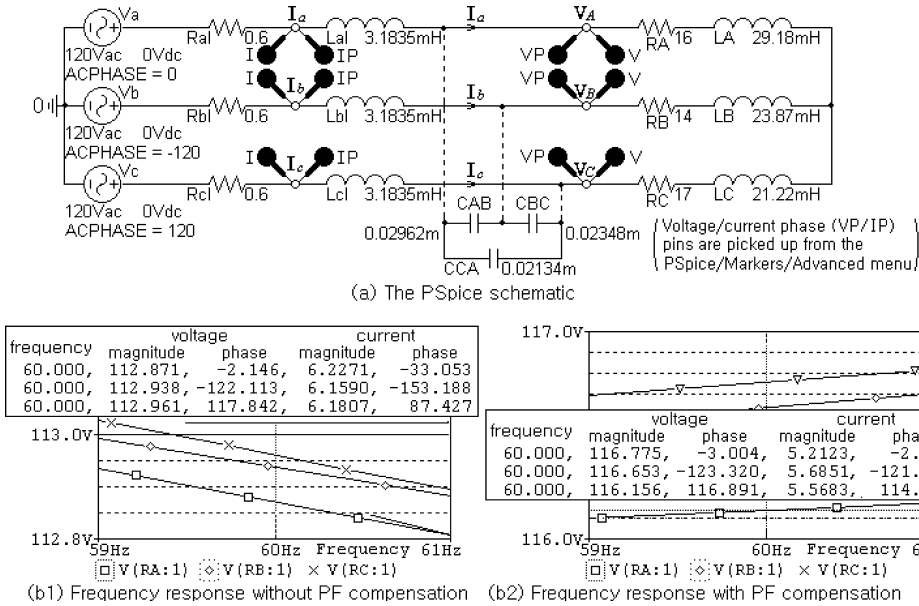


Figure 7.7 PSpice simulation of a three-phase power system

(Example 7.1) MATLAB Analysis and PSpice Simulation of a Three-Phase Power System

Consider the three-phase power system in Figure 7.7(a), where the voltage sources and the resistances/inductances of the transmission lines and loads are

$$\text{Three voltage sources : } \mathbf{V}_a = 120\angle 0^\circ, \quad \mathbf{V}_b = 120\angle -120^\circ, \quad \mathbf{V}_c = 120\angle +120^\circ \quad (\text{E7.1.1})$$

$$(\omega = 2\pi f = 2\pi \times 60 \cong 377 \text{ rad/s})$$

$$\text{Three line impedances with } R_{al} = R_{bl} = R_{cl} = 0.6 \Omega, \quad L_{al} = L_{bl} = L_{cl} = 3.1835 \text{ mH} \quad (\text{E7.1.2})$$

$$\text{Three load impedances with } R_{AL} = 16 \Omega, \quad R_{BL} = 14 \Omega, \quad R_{CL} = 17 \Omega \quad (\text{E7.1.3})$$

$$L_{AL} = 29.18 \text{ mH}, \quad L_{BL} = 23.87 \text{ mH}, \quad L_{CL} = 21.22 \text{ mH}$$

The bank of three Δ -connected capacitors connected in dotted lines with the system will be installed to improve the power factor of the three-phase load to unity (100%). To find the values of the capacitances for PF correction, the Y- Δ conversion of the Y-connected load is made and the values of the capacitances are found such that each parallel combination of a capacitor and a load in Δ -connections will be purely resistive:

$$Z_{AL} = R_{AL} + j\omega L_{AL} = 16 + j11, \quad Z_{BL} = R_{BL} + j\omega L_{BL} = 14 + j9, \quad Z_{CL} = 17 + j8 \Omega$$

$$\begin{aligned} &\text{yd\_conversion()} \\ &\rightarrow Z_{AB} = 42.77 + j31.52, \quad Z_{BC} = 45.78 + j25.40, \quad Z_{CA} = 52.53 + j28.94 \Omega \end{aligned} \quad (\text{E7.1.4})$$

$$C_{AB} = -\frac{\text{Im}\{1/Z_{AB}\}}{\omega} = 29.62 \mu\text{F}, \quad C_{BC} = -\frac{\text{Im}\{1/Z_{BC}\}}{\omega} = 23.48 \mu\text{F}, \quad C_{CA} = 21.34 \mu\text{F} \quad (\text{E7.1.5})$$

Note that in order to find the composite impedance of a Y-connected load and a Δ -connected load, as a general rule the Y- Δ conversion of the Y-connected one should be made, the parallel combination

of the two Δ -connected loads is computed, and then, as needed, the Δ -Y conversion of the composite Δ -connected circuit is made to an overall Y-connected circuit.

In any case it would be very time consuming to do Y- Δ or Δ -Y conversions and, moreover, apply the formulas (7.16) to (7.18) with the above dirty values to compute the phase voltages and currents by hand. Thus the MATLAB routines `y_y()` and `y_dy()` will be used and the PSpice simulation will be performed to analyze this circuit.

(a) The program `cir07e01.m` is composed and run to perform the following jobs:

- (1) It uses the routine `y_y()` to solve the three-phase system without the bank of capacitors, where `y_y()` finds the load-side neutral voltage V_N , the three-phase voltages V_A , V_B , and V_C at the receiving ends, and the three line currents I_a , I_b , and I_c .
- (2) It uses the routine `yd_conversion()` to get the equivalent Δ -connected loads.
- (3) It finds the bank of Δ -connected capacitances C_{AB} , C_{BC} , and C_{CA} that should be connected in parallel with the Δ -connected loads to make them purely resistive so that the PF will be raised to unity (1).
- (4) It uses the routine `y_dy()` to solve the composite three-phase system consisting of the Y-connected loads and the Δ -connected bank of PF compensating capacitors.

Note. Alternatively, the Y- Δ conversion can be made of the Y-connected loads, the equivalent Δ -connected loads combined in parallel with the Δ -connected capacitors, the Δ -Y conversion made of the composite loads, and the routine `y_y()` used to solve the equivalent Y-Y three-phase system with the bank of capacitors.

```
%cir07e01.m
clear
f=60; w=2*pi*f; jw=j*w; % The source frequency
Vabc=[120 120*exp(-j*2*pi/3) 120*exp(j*2*pi/3)]; % Eq. (E7.1.1)
Zal=0.6+jw*3.1835e-3; Zabcl=[Zal Zal Zal]; % The line impedances (E7.1.2)
% Y-connected three-phase load impedances from Eq. (E7.1.3)
ZAL=16+jw*29.18e-3; ZBL=14+jw*23.87e-3; ZCL=17+jw*21.22e-3; % Eq. (E7.1.4)
% Analysis of 3-phase 3-wire power system without PF correction
ZABCL=[ZAL ZBL ZCL]; [VN,VABCN,Iabc,SABC]=y_y(Vabc,ZABCL,Zabcl);
[ZAB,ZBC,ZCA]=yd_conversion(ZAL,ZBL,ZCL); % Eq. (E7.1.4)
disp('Capacitances to be connected in parallel with the existing load')
CABC=-imag(1./[ZAB ZBC ZCA])/w % Eq. (E7.1.5)
PFc=1; % desired Power Factor for correction
CABC=wC_for_PF_correction([ZAB ZBC ZCA],PFc)/w % Alternatively, Sec. 6.6
ZABC_C=1./(jw*CABC); % Impedances of Delta-connected compensating capacitors
disp('After PF correction')
% Analysis of 3-phase 3-wire power system with Delta-connected capacitors
[VN_c,VABCN_c,Iabc_c,SABC_c]=y_dy(Vabc,ZABCL,ZABC_C,Zabcl);
```

The above MATLAB program `cir07e01.m` is run to get the following result:

```
>>cir07e01
Neutral Voltage (Mag&Phase) at Load side= 10.8893 -137.8994
Load end voltages (Mag&Phase) Line currents (Mag&Phase) Complex powers
1.0e+002 *
    1.1287 -0.0215    0.0623 -0.3305    6.2043+4.2657i
    1.1294 -1.2211    0.0616 -1.5319    5.3107+3.4135i
    1.1296  1.1784    0.0618  0.8743    6.4942+3.0560i
```

```

Capacitances to be connected in parallel with the existing load
CABC = 1.0e-004 * 0.2962 0.2348 0.2134
After PF Correction
Neutral Voltage (Mag&Phase) at Load side= 5.9233 -170.3062
Load end voltages (Mag&Phase) Line currents (Mag&Phase) Complex powers
116.7743 -3.0040 5.2124 -2.3953 1917.3+1143.9i
116.6525 -123.3196 5.6847 -121.1170 0 -1143.9i
116.1562 116.8903 5.5684 114.0588

```

This means that

$$\begin{aligned}
 \mathbf{V}_A &= 112.87\angle -2.15^\circ, & \mathbf{V}_B &= 112.94\angle -121.11^\circ, & \mathbf{V}_C &= 112.96\angle 117.84^\circ \\
 \mathbf{I}_a &= 6.23\angle -33.05^\circ, & \mathbf{I}_b &= 6.16\angle -153.19^\circ, & \mathbf{I}_c &= 6.18\angle 87.43^\circ \\
 \xrightarrow{\text{PF correction}} \mathbf{V}_A &= 116.77\angle -3.00^\circ, & \mathbf{V}_B &= 116.65\angle -123.32^\circ, & \mathbf{V}_C &= 116.16\angle 116.89^\circ \\
 \mathbf{I}_a &= 5.21\angle -2.40^\circ, & \mathbf{I}_b &= 5.68\angle -121.12^\circ, & \mathbf{I}_c &= 5.57\angle 114.06^\circ
 \end{aligned}$$

Note. Note the following:

1. The amplitudes of the voltages $\mathbf{V}_A$, $\mathbf{V}_B$, and $\mathbf{V}_C$ at the receiving ends have become higher with the PF correction.
 2. The complex powers of the Y-connected three-phase load and the Δ -connected capacitor bank are $1917.3 + j1143.9$ and $-j1143.9$, respectively. Thus the composite complex power is purely real, implying that the resulting power factor of 100 % has been achieved by the PF correction.
- (b) Perform the PSpice simulation for the three-phase circuit in Figure 7.7(a).
- Draw the schematic as depicted in Figure 7.7(a), where the three VAC voltage sources are placed and their ACPHASE values are set to 0 or -120 or $+120$ in the Property Editor spreadsheet. Do not place the capacitors yet.
 - In the Simulation Settings dialog box, set the Analysis type to ‘AC Sweep’ with the parameters as Start Frequency = 59, End Frequency = 61, and Points/Decade = 200.
 - Place V/VP/I/IP Markers to measure the magnitudes/phases of $\mathbf{V}_A$, $\mathbf{V}_B$, $\mathbf{V}_C$, $\mathbf{I}_a$, $\mathbf{I}_b$, and $\mathbf{I}_c$ at the appropriate points as depicted in Figure 7.7(a).
 - Click the Run button on the toolbar to make the PSpice A/D (Probe) window appear on the screen as depicted in Figure 7.7(b1).
 - To get the numeric values of the measured variables, click the Toggle Cursor button on the toolbar, click the graphic symbol before each variable name at the bottom part of the Probe window by the left/right mouse button, and move the cross-type cursor to the 60 Hz position by pressing the left/right (Shift+) Arrow key or by using the left/right mouse button. Then you can read the numeric value of the measured variable from the Probe Cursor box (Figure 7.7(b1)).
 - Modify the schematic by placing the capacitors as depicted in the dotted lines, click Run, and get the new numeric values of the measured variables (Figure 7.7(b2)).

Finally, compare the numeric values of $\mathbf{V}_A$, $\mathbf{V}_B$, $\mathbf{V}_C$, $\mathbf{I}_a$, $\mathbf{I}_b$, and $\mathbf{I}_c$ with those obtained from the MATLAB analysis in (a). If they turn out to be (almost) the same, you may celebrate your success.

7.5 Electric Shock and Grounding

DC circuits have been discussed in the first four chapters and AC circuits have also been studied. Equipped with basic knowledge about circuit theory and electrical terminology such as voltage and current, we may well relate the theory to the electrical devices and systems around us and begin to think about not only the usefulness but also the potential danger of electricity. Electricity quickly endangers our lives as well as meeting our convenience. But what use is all our knowledge if we

happen to get injured or die as a result of an electrical accident? At this point, let us put aside the theoretical aspects for a moment and think about the electrical safety issue. However, while the safety issue may require several volumes for a comprehensive treatment, our discussion on this aspect will be very limited.

In the context of electrical safety, a question may arise:

‘Which electricity endangers our life, high voltage or large current?’

Even if this question may sound absurd, it should be answered sincerely as follows:

‘Both of them, but the former is dangerous as a cause, while so is the latter as a consequence.’

To be more specific, the fatality of an electrical shock depends on several factors such as how large the current is and how long and through which part of the human body the current flows, irrespective of the voltage causing it. The voltage is just a potential cause of a dangerous accident. Even though a person happens to be brought into contact with a conductor at high voltage, it would not be so dangerous as long as the resistance of the path via his/her body between the points of contact or the contact and the ground is large enough to keep the current less than a few milliamperes. Even an electrostatic voltage higher than 20 kV, which may damage some electronic devices, yields nothing more than a little discomfort to a human being because it usually causes the current to flow mainly over the body surface, and that for only a few microseconds. However, since the resistance of a human body with wet skin can be as small as a few hundred ohms, a person may be killed by 100 V AC or a much lower voltage of DC.

Before going into an example addressing the safety issue, note the following tips to avoid electrical shock when you are going to touch electrical/electronic appliances:

1. Turn off the electricity without assuming that the circuit is dead. If they have a capacitor of large capacitance, you should be very careful because it takes time to discharge after the power is off.
2. Noting that prevention is the best medicine, do not touch them when you are wet.
3. Respecting all voltage levels, use safety devices, wear suitable clothing (insulated shoes, gloves, etc.), and use just one (right) hand, especially when touching a high-voltage system.
4. Use a dry board, belt, clothing, or other available nonconductive material to free the victim from electrical shock. Do not touch the victim until the source of electricity is removed.
5. Make sure that there is a third wire on the plug for grounding in case of a short-circuit accident. The fault current should flow through the third wire to ground instead of through the operator’s body to ground if an electric power apparatus is grounded or an insulation breakdown occurs.
6. The website <<http://www.smud.org/safety/world/index.html>> is worthwhile to visit for more information about electrical safety.

Note. How can birds sit on a power line without getting an electrical shock? It is because they are not touching the ground and so the electricity cannot find a path to flow to the ground. However, if one catches one power line with one leg and another line with the other leg, it will be killed instantly before realizing how serious the mistake is. Likewise, if your kite or balloon gets tangled in a power line when you touch the string, electricity could travel down the string and into your body on its way to the ground, causing a fatal shock.

(Example 7.2) Ground Fault Interrupter (GFI) with Grounding to Prevent an Electrical Hazard

Ground fault interrupters are designed to prevent an electrical shock by interrupting a household circuit when there is a difference between the currents in the hot and neutral lines. Such a difference indicates that an abnormal diversion of current occurs from the hot line, which might be flowing in the ground line.

- (a) Figure 7.8(a) shows the connection diagram for a GFI that is used to prevent an electrical hazard against the case where the insulation of the motor winding inside the metal case fails and a user

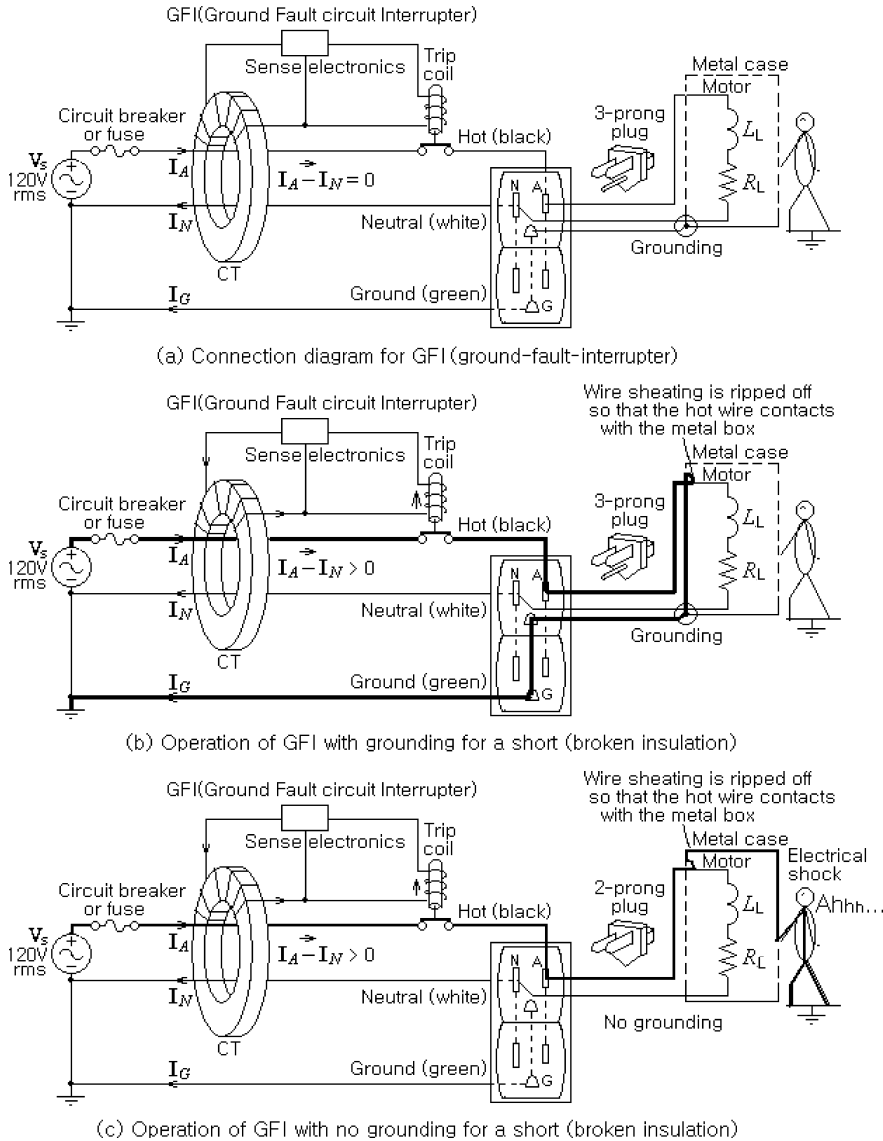


Figure 7.8 GFI process for preventing, detecting, and tripping a short-circuit, and the consequences of no grounding

touches the metal case. Note that in a normal situation with perfect insulation, the primary current of the current transformer (CT) (Problem 5.11) is $I_A - I_N = 0$ so that the secondary coil carries no current to produce a force needed to open the switch.

- (b) Figure 7.8(b) shows how the GFI detects a short-circuit and produces a tripping signal to open the switch; i.e. in the case where the insulation of the motor winding inside the metal case fails, a large current flows from the fault position to the ground. This current will be I_A , so that $I_A - I_N > 0$ and a nonzero current through the secondary coil produces a tripping signal to open the switch. Since

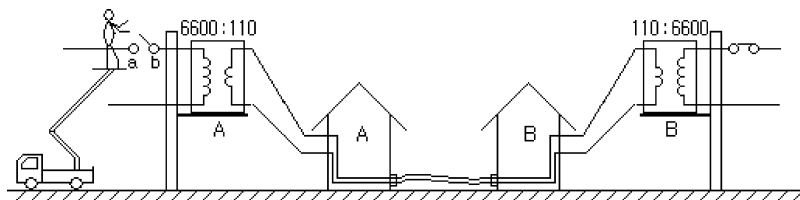


Figure 7.9 (From Reference [I-1]. Source: © Prentice-Hall)

the metal case is grounded, the user touching it will get no electrical shock regardless of whether the GFI works or not.

- (c) Figure 7.8(c) shows the situation in which the metal case is not grounded. Everything is almost the same as in (b) except that the fault current flows to the ground not directly, but via the human body till the switch is opened by operation of the GFI so that the user might get an electrical shock before the circuit is interrupted. Besides, the fault current is less than that with grounding, possibly causing some delay in the tripping operation of the GFI. This makes us realize the importance of the grounding or ‘chassis ground’ for safety.

Note. Fuses and/or breakers are used to limit the current in most household applications. However, the typical limit of current to be interrupted by them is 20 A and their tripping operation is too slow to prevent electrocution. That is why GFIs are required by the electrical code for receptacles in bathrooms and kitchens, near swimming pools, and outside. The GFI is expected to detect currents of a few milliamperes and trip a breaker to remove the shock hazard.

(Example 7.3) Danger Hidden behind Help (Source: J. D. Irwin and C. H. Wu, *Basic Engineering Circuit analysis*, 6th edition, 1999, Example 11.12 with Figure 11.20. Source: © Prentice Hall)

Figure 7.9 describes the situation where the power line feeding house A is interrupted because of some fault and the person living in the house borrows electric power from his neighbor B (fed from another power line) by connecting a long extension cord between an outside receptacle in house A and another in house B. After the fault is recovered, a line technician from the utility company comes to reconnect the circuit breaker at the primary side installed on the utility pole. Not being informed of the fact that house A is fed from another power line and so the power transformer A is alive, he/she might touch contact b (at 6600 V) without wearing any nonconductive gloves and might never see his/her family again.

Problems

7.1 An Unbalanced 3ϕ -3w (Three-Phase Three-Wire) Power System

Figure P7.1 shows a Y-Y type of 3ϕ -3w power system operated at the source frequency of 60 Hz, where a bank of capacitors are to be installed for power factor (PF) correction.

- Find the voltages (V_A , V_B , and V_C) at the load end and the line currents (I_a , I_b , and I_c) with no capacitors in the polar form as $V_A = 112\angle -1.58^\circ$ with three significant digits.
- Find the three capacitances needed to raise the power factor of the three-phase load to unity (1) in the form $C_{AB} = 49.3\ \mu\text{F}$ with three significant digits.
- Find the voltages (V_A , V_B , and V_C) at the load end and the line currents (I_a , I_b , and I_c) with the capacitors for PF correction in the polar form as $I_a = 5.15\angle -3.33^\circ$ with three significant digits.

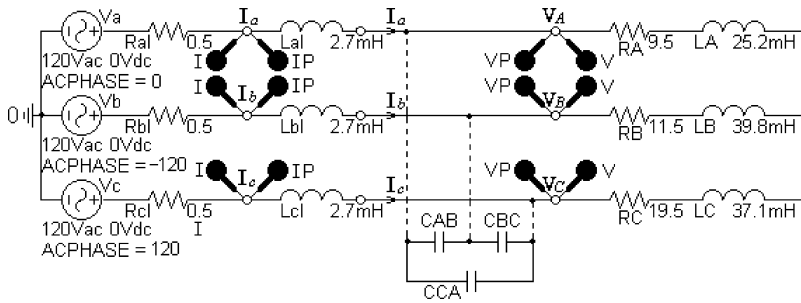


Figure P7.1

Hint. Referring to the MATLAB program `cir07e01.m` presented for solving Example 7.1 in Section 7.4, use the MATLAB routines `y_y()` and/or `y_dy()`.

- (d) Perform the PSpice simulation (AC Sweep analysis for 200 frequency points/decade between 59 Hz and 61 Hz) two times, once without the PF compensating capacitors and once with them. Fill in the blanks of Table P7.1 with the PSpice simulation results and the theoretical analysis results obtained in (a) and (c).

Table P7.1 Results of the theoretical analysis and PSpice simulation

		V_A	V_B	V_C	I_a	I_b	I_c
Before PF	Theoretical	$112\angle -1.58^\circ$					
correction	PSpice		$113\angle -120^\circ$				$5.25\angle 85.5^\circ$
After PF	Theoretical			$118/118^\circ$		$4.22\angle -128^\circ$	
correction	PSpice			$5.15\angle -3.32^\circ$			

7.2 An Unbalanced 3ϕ -4w (Three-Phase Four-Wire) Power System

Figure P7.2 shows a Y-Y type of 3ϕ -4w power system operated at the source frequency of 60 Hz. Perform the MATLAB analysis and PSpice simulation for the system two times, once with the bank of capacitors and once without it. Make a table similar to Table P7.1.

Hint. You can complete the following MATLAB program `cir07p02.m` and run it.

```
%cir07p02.m
f=60; w=2*pi*f; jw=j*w;
Vabc=[120 120*exp(-i*2*pi/3) 120*exp(i*2*pi/3)];
Zal=??? +jw*???????; Zbl=Zal; Zcl=Zal; Znl=? +jw*???????; Zabc1=[Za Zb Zcl];
ZAL=??? +j*???????; ZBL=???? +jw*???????; ZCL=???? +jw*???????;
ZABCL=[ZAL ZBL ZCL]; % The Y-connected load
[VN,VABCN,Iabc,SABC]=y_y(Vabc,ZABCL,[Zabc1 Znl]);
CABC=[ ???????? ???????? ????????]; ZABC=-1./(jw*CABC); % D-connected load
disp('After PF correction')
[VN_c,VABCN_c,Iabc_c,SABC_c]=y_dy(Vabc,ZABCL,ZABC,[Zabc1 Znl]);
```

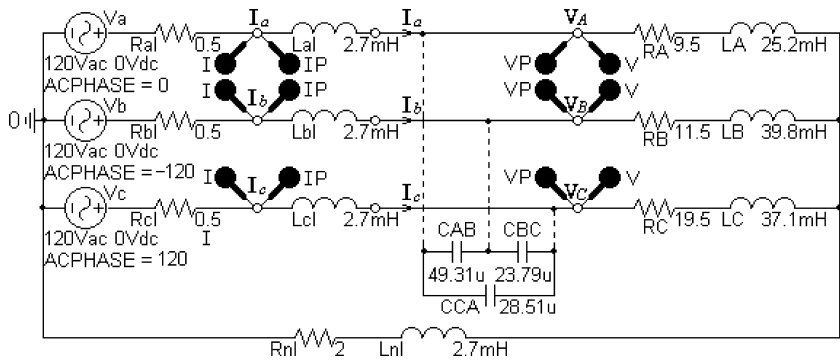


Figure P7.2

7.3 Parallel Combination of the Unbalanced Y-Connected Load and Δ -Connected Load

As mentioned in Section 7.4 and illustrated in Figure P7.3, the parallel connection of the Y-connected load and the Δ -connected load should be initiated by making the Y- Δ conversion of the Y-connected one rather than making the Δ -Y conversion of the Δ -connected one.

- (a) To be assured of this assertion, solve the circuit with the capacitor bank in Figure P7.1 to find V_A , V_B , and V_C in the following two ways:
- (1) Make the Δ -Y conversion of the Δ -connected capacitor bank and combine it with the Y-connected load in parallel (Figure P7.3(b1)–(b2)). Then use the MATLAB routine `y_y ( )` to solve the circuit and check if the results agree with those obtained in Problem 7.1(c).
 - (2) Make the Y- Δ conversion of the Y-connected load, combine it with the Δ -connected capacitor bank in parallel, and make the Δ - γ conversion (Figure P7.3(c1)–(c3)). Then use the MATLAB routine `y_y ( )` to solve the circuit and check if the results agree with those obtained in Problem 7.1(c).

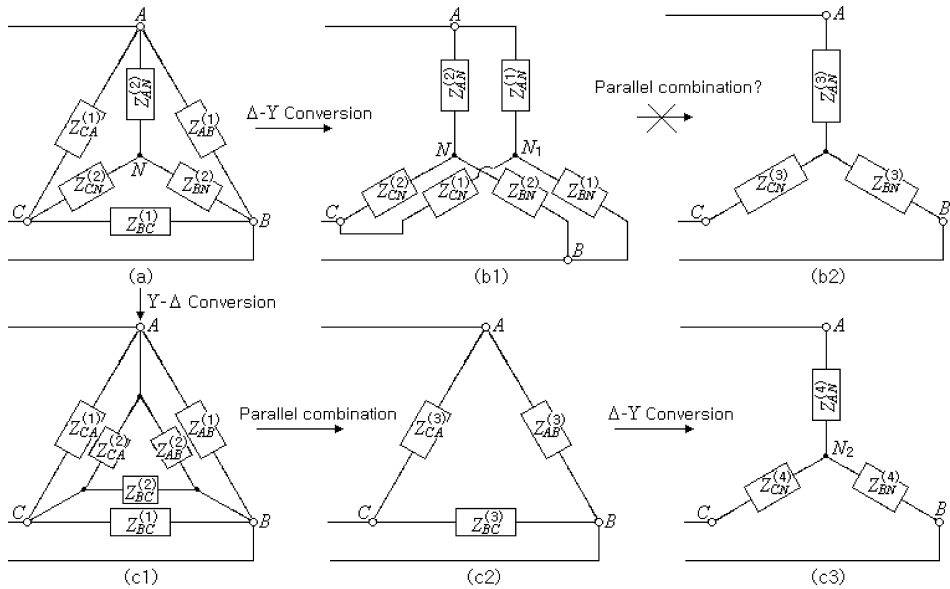


Figure P7.3 Parallel combination of the Y-connected load and the Δ -connected load

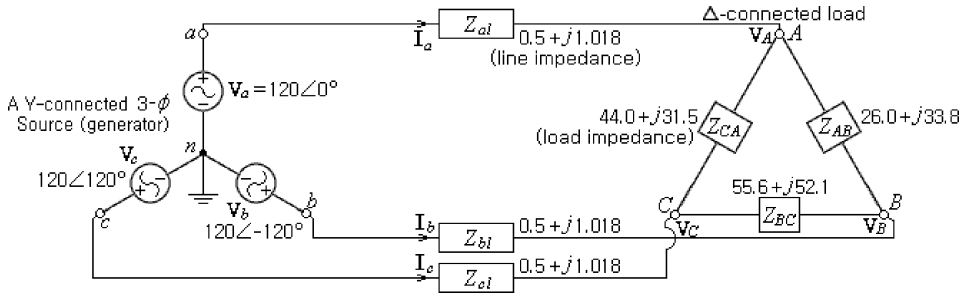


Figure P7.4 A Y-Δ connected 3φ-3w (three-phase three-wire) power system

(b) Does the parallel combination of a Y-connected load and a Δ-connected load work for a 3φ-4w power system like the one depicted in Figure P7.2?

7.4 An Unbalanced Y-Δ Connected 3φ-3w Power System

Figure P7.4 shows a Y-Δ type of 3φ-3w power system operated at the source frequency of 60 Hz. A set of node equations can be written in the three unknown node voltages V_A , V_B , and V_C as follows:

$$\begin{bmatrix} Y_{al} + Y_{AB} + Y_{CA} & -Y_{AB} & -Y_{CA} \\ -Y_{AB} & Y_{bl} + Y_{AB} + Y_{BC} & -Y_{BC} \\ -Y_{CA} & -Y_{BC} & Y_{cl} + Y_{CA} + Y_{BC} \end{bmatrix} \begin{bmatrix} V_A \\ V_B \\ V_C \end{bmatrix} = \begin{bmatrix} Y_{al} V_a \\ Y_{bl} V_b \\ Y_{cl} V_c \end{bmatrix} \quad (\text{P7.4.1})$$

After solving this set of equations for V_A , V_B , and V_C , the line currents can be obtained as

$$I_a = \frac{V_a - V_A}{Z_{al}}, \quad I_b = \frac{V_b - V_B}{Z_{bl}}, \quad I_c = \frac{V_c - V_C}{Z_{cl}} \quad (\text{P7.4.2})$$

This solution procedure for the Y-Δ connected 3φ-3w power system is cast into the following MATLAB routine `y_d()`.

```
function[VABC,Iabc,SABC]=y_d(Vabc,ZABC,Zabcl)
% To solve a 3p-3w system with Delta-connected loads
%Input: Vabc=[Va Vb Vc]: the three phase voltage sources
%       ZABC=[ZAB ZBC ZCA]: the Delta-connected three phase load impedances
%       Zabcl=[Zal Zbl Zcl]: the three line impedances
%Output: VABC=[VA;VB;VC]: the load-side end voltages
%        Iabc=[Ia;Ib;Ic]: the three line currents
%        SABC=[SAB;SBC;SCA]: the 3-phase complex power
% Copyright: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only
Va=Vabc(1); Vb=Vabc(2); Vc=Vabc(3);
YABC=1./ZABC; Yabcl=1./Zabcl;
YAB=YABC(1); YBC=YABC(2); YCA=YABC(3); % each of Y-connected admittances
Yal=Yabcl(1); Ybl=Yabcl(2); Ycl=Yabcl(3); % each line admittance
Y=[Yal+YAB+YCA -YAB -YCA; -YAB Ybl+YAB+YBC -YBC; -YCA -YBC Ycl+YBC+YCA];
VABC=Y\[Yal*Va; Ybl*Vb; Ycl*Vc]; % Solve Eq. (P7.4.1)
Iabc=Yabcl(:).*(Vabc(:)-VABC); % Eq. (P7.4.2)
VABC_Delta=VABC-VABC([2 3 1]); % Delta phase voltages
SABC=VABC_Delta.*conj(VABC_Delta./ZABC(:)); % Eq. (6.28)
disp('Load end voltages (Mag&Phase) Line currents (Mag&Phase) Complex powers')
disp([abs(VABC) angle(VABC)*180/pi abs(Iabc) angle(Iabc)*180/pi SABC])
```

- (a) Make use of the MATLAB routine `y_d ( )` to solve the power system of Figure P7.4 for V_A , V_B , V_C , I_a , I_b , and I_c .
- (b) Make the Δ -Y conversion of the Δ -connected loads and make use of the MATLAB routine `y_y ( )` to solve the power system for V_A , V_B , V_C , I_a , I_b , and I_c . Does the solution agree with that obtained in (a)?

7.5 Comparison of Various Power Transmission Schemes

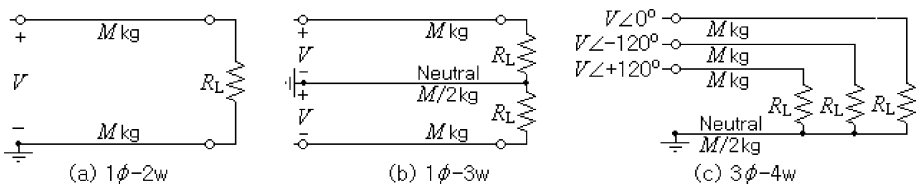


Figure P7.5

Figures P7.5(a), (b), and (c) show 1ϕ -2w, 1ϕ -3w, and 3ϕ -4w transmission schemes, respectively, where the mass of the neutral line is assumed to be half of that (M) of a hot line. Verify that the ratio of the power to the weight of power transmission lines for each of the three schemes is as listed in Table P7.5.

Table P7.5 Comparison of various power transmission schemes

Transmission scheme	1ϕ -2w	1ϕ -3w	3ϕ -4w
Transmitted power	$P_1 = V^2/R_L$	$P_2 = 2V^2/R_L = 2P_1$	$P_3 = 3V^2/R_L = 3P_1$
Weight of power lines	$2M$	$2.5M$	$3.5M$
Ratio of transmitted power to weight of lines	$(1/2)P_1/M$	$(4/5)P_1/M$	$(6/7)P_1/M$

8

Frequency Selective Circuit – Filter

While the impulse response of a circuit is the variation in its output to an impulse input with time, the frequency response is the variation in its output to an impulse input with frequency. The frequency of the input source was fixed in the previous two chapters, but in this chapter the situation is discussed where the input source has various frequency components. Any circuit having such reactive components as inductors/capacitors whose impedance varies with the frequency of the input source is called a frequency selective circuit, i.e. a filter in the sense that it passes/rejects certain frequency components of an input signal or its output varies with the input source frequency. The frequency response of a filter plotted versus frequency describes how the filter discriminates the various frequency components contained in the input.

In Sections 4.1 and 4.5, the *transfer function* is defined as the ratio of the Laplace transform of the output to the Laplace transform of the input or, equivalently, the Laplace transform of the impulse response. The *frequency response* is defined as the ratio of the phasor output to the phasor input, which turns out to be a function of the input source frequency ω . Theoretically, the frequency response $G(j\omega)$ of a system can be obtained by substituting $s = j\omega$ in its transfer function $G(s)$, where ω denotes the (input) frequency variable. Technically, it can be obtained by PSpice simulation corresponding to the analysis type 'AC Sweep'. Experimentally, it can be measured by using a sinusoidal wave generator and a spectrum analyzer. The frequency response is very useful for analysis, design, and application of many physical systems including electrical and mechanical devices.

Depending on the frequency band for which the magnitude of a filter frequency response is large or small, the filter can be classified as a lowpass/highpass/bandpass/bandstop filter. The *passband/stopband* mean the ranges of frequencies in which the magnitude of the frequency response is relatively large/small, respectively. The transfer function, the frequency response, and the filter type of a given circuit may differ depending on which pair of terminals is selected as the output port. Several important properties of a filter will be of interest, such as the cutoff frequency, bandwidth, resonance, and quality factor. While Sections 8.1 to 8.4 deal with passive filters consisting of only resistors, capacitors, and inductors, Section 8.5 introduces active filters employing OP Amps. Section 8.6 discusses the analog filter design.

8.1 Lowpass Filter (LPF)

8.1.1 Series LR Circuit

Figure 8.1(a) shows a series LR circuit where the voltage $v_R(t)$ across the resistor R is taken as the output to the input voltage source $v_i(t)$. With $V_R(s) = \mathcal{L}\{v_R(t)\}$ and $V_i(s) = \mathcal{L}\{v_i(t)\}$, its input–output

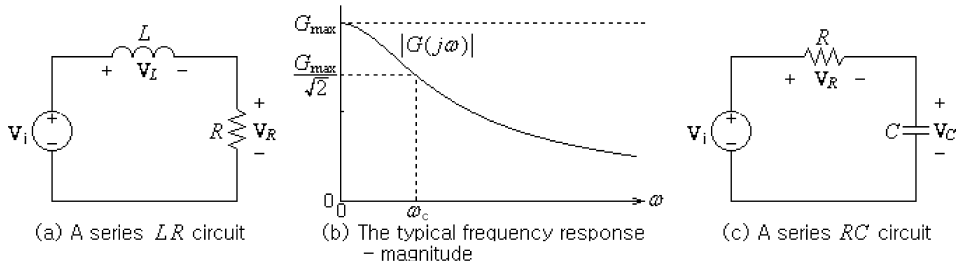


Figure 8.1 Lowpass filters and their typical frequency response

relationship can be described by the transfer function and the frequency response as

$$G(s) = \frac{V_R(s)}{V_i(s)} = \frac{R}{sL + R} = \frac{R/L}{s + R/L} \quad (8.1)$$

$$G(j\omega) = \frac{\mathbf{V}_R(\text{phasor transform of } v_R(t))}{\mathbf{V}_i(\text{phasor transform of } v_i(t))} = \frac{R}{j\omega L + R} = \frac{R/L}{j\omega + R/L} \quad (8.2)$$

Since the magnitude $|G(j\omega)|$ of this frequency response becomes smaller as the frequency ω becomes higher, as depicted in Figure 8.1(b), the circuit is a *lowpass filter* that prefers to have low-frequency components in its output. Noting that $|G(j\omega)|$ achieves the maximum $G_{\max} = G(j0) = 1$ at $\omega = 0$, we can find the *cutoff frequency* ω_c at which it equals $G_{\max}/\sqrt{2} = 1/\sqrt{2}$ as

$$\begin{aligned} |G(j\omega_c)| &\stackrel{(8.2)}{=} \left| \frac{R/L}{\sqrt{\omega^2 + (R/L)^2}} \right|_{\omega=\omega_c} = \frac{1}{\sqrt{2}} \\ \omega_c &= \frac{R}{L} \end{aligned} \quad (8.3)$$

The cutoff frequency has a physical meaning as the boundary frequency between the passband and the stopband. Why is $1/\sqrt{2}$ used in its definition? In most cases, the transfer function and the frequency response is the ratio of output and input voltages/currents. Since an electric power is proportional to the squared voltage and current, the ratio of $1/\sqrt{2}$ between voltages/currents corresponds to the ratio of $1/2$ between powers. For this reason, the cutoff frequency is also referred to as a *half-power frequency* or a *3 dB frequency* on account of the fact that $10 \log_{10}(1/2) = -3$ dB. In the case of a lowpass filter with the cutoff frequency ω_c , as this circuit, the *passband* is the range of frequency $[0, \omega_c]$ and its width is the *bandwidth*, where the bandwidth means the width of the range of frequency components that pass the filter better than others.

8.1.2 Series RC Circuit

Figure 8.1(c) shows a series RC circuit where the voltage $v_C(t)$ across the capacitor C is taken as the output to the input voltage source $v_i(t)$. With $V_C(s) = \mathcal{L}\{v_C(t)\}$ and $V_i(s) = \mathcal{L}\{v_i(t)\}$, its input–output relationship can be described by the transfer function and the frequency response as

$$G(s) = \frac{V_C(s)}{V_i(s)} = \frac{1/(sC)}{R + 1/(sC)} = \frac{1/(RC)}{s + 1/(RC)} \quad (8.4)$$

$$G(j\omega) = \frac{\mathbf{V}_C(\text{phasor transform of } v_C(t))}{\mathbf{V}_i(\text{phasor transform of } v_i(t))} = \frac{1/(j\omega C)}{R + 1/(j\omega C)} = \frac{1/(RC)}{j\omega + 1/(RC)} \quad (8.5)$$

Everything is the same with the series LR circuit in Figure 8.1(a) except that the cutoff frequency is

$$\omega_c = \frac{1}{RC} \quad (8.6)$$

[Remark 8.1] Cutoff Frequency and Time Constant of a Lowpass Filter

Referring to Section 3.4.3, the *cutoff frequency* of a system functioning as a lowpass filter like the circuits in Figures 8.1(a) and (c) is the same as the reciprocal of its *time constant*. Therefore, the shorter the time constant, the higher is the cutoff frequency. This reflects the fact that a system with a shorter time constant responds to an input faster than one with a longer time constant and consequently can process rapidly changing input signals of higher frequency.

8.2 Highpass Filter (HPF)

8.2.1 Series CR Circuit

Figure 8.2(a) shows a series CR circuit where the voltage $v_R(t)$ across the resistor R is taken as the output to the input voltage source $v_i(t)$. Its input–output relationship can be described by the transfer function and the frequency response as

$$G(s) = \frac{V_R(s)}{V_i(s)} = \frac{R}{R + 1/(sC)} = \frac{s}{s + 1/(RC)} \quad (8.7)$$

$$G(j\omega) = \frac{\mathbf{V}_R}{\mathbf{V}_i} = \frac{j\omega}{j\omega + 1/(RC)} \quad (8.8)$$

Since the magnitude $|G(j\omega)|$ of this frequency response becomes larger as the frequency ω gets higher, as depicted in Figure 8.2(b), the circuit is a *highpass filter* that prefers to have high-frequency components in its output. Noting that $|G(j\omega)|$ achieves the maximum $G_{\max} = G(j\infty) = 1$ at $\omega = \infty$, we can find the *cutoff frequency* ω_c at which it equals $G_{\max}/\sqrt{2} = 1/\sqrt{2}$ as

$$|G(j\omega_c)| = \frac{\omega}{\sqrt{\omega^2 + (1/RC)^2}} \bigg|_{\omega=\omega_c} = \frac{1}{\sqrt{2}}; \quad \omega_c = \frac{1}{RC} \quad (8.9)$$

8.2.2 Series RL Circuit

Figure 8.2(c) shows a series RL circuit where the voltage $v_L(t)$ across the inductor L is taken as the output to the input voltage source $v_i(t)$. Its input–output relationship can be described by the transfer function

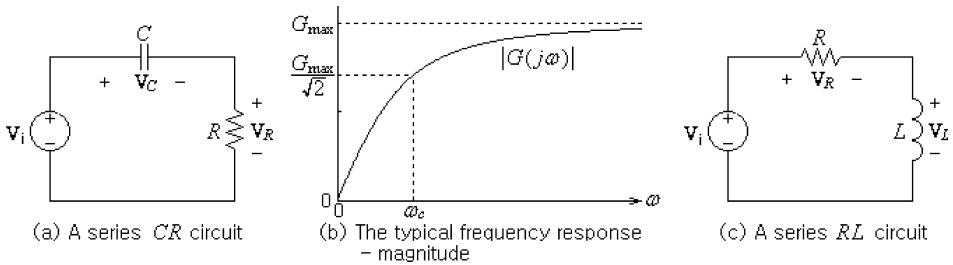


Figure 8.2 Highpass filters and their typical frequency response

and the frequency response as

$$G(s) = \frac{V_L(s)}{V_i(s)} = \frac{sL}{R + sL} = \frac{s}{s + R/L} \quad (8.10)$$

$$G(j\omega) = \frac{\mathbf{V}_L}{\mathbf{V}_i} = \frac{j\omega}{j\omega + R/L} \quad (8.11)$$

Everything is the same with the series CR circuit in Figure 8.2(a) except that the cutoff frequency is

$$\omega_c = \frac{R}{L} \quad (8.12)$$

[Remark 8.2] Frequency Characteristic of Inductor and Capacitor

1. An inductor can be regarded as a highpass filter with its current $i_L(t)$ and voltage $v_L(t) = L di_L(t)/dt$ as the input and output, respectively. This can be observed from the fact that the inductor has the output voltage $v_L(t) = \omega L I_m \cos(\omega t)$ to an input current $i_L(t) = I_m \sin(\omega t)$ and the magnitude of the output becomes larger as the frequency ω of the input becomes higher. This is based on the fact that the AC impedance is $\mathbf{V}_L/\mathbf{I}_L = j\omega L$ whose magnitude becomes larger as ω becomes higher (see the RL circuit in Figure 8.2(c)). On the other hand, an inductor can be regarded as a lowpass filter with its voltage $v_L(t)$ and current $i_L(t) = (1/L) \int v_L(\tau) d\tau$ as the input and output, respectively. This can be observed because the inductor has the output current $i_L(t) = V_m \sin(\omega t)/\omega L$ to an input voltage $v_L(t) = V_m \cos(\omega t)$ and the magnitude of the output becomes small as the frequency ω of the input becomes higher. This is based on the fact that the AC admittance is $\mathbf{I}_L/\mathbf{V}_L = 1/(j\omega L)$ whose magnitude becomes smaller as ω becomes higher. This explains why an inductor is connected in series with a load when there is a need to attenuate high-frequency components in order to apply a DC-like smooth (low-frequency) signal to the load (see the LR circuit in Figure 8.1(a)).
2. A capacitor can be regarded as a lowpass filter with its current $i_C(t)$ and voltage $v_C(t) = (1/C) \int i_C(\tau) d\tau$ as the input and output, respectively. This can be observed because the capacitor has the output voltage $v_C(t) = I_m \sin(\omega t)/(\omega C)$ to an input current $i_C(t) = I_m \cos(\omega t)$ and the magnitude of the output becomes smaller as the frequency ω of the input becomes higher. This is based on the fact that the AC impedance is $\mathbf{V}_C/\mathbf{I}_C = 1/(j\omega C)$ whose magnitude becomes smaller as ω becomes higher (see the RC circuit in Figure 8.1(c)). On the other hand, a capacitor can be regarded as a highpass filter with its voltage $v_C(t)$ and current $i_C(t) = C dv_C(t)/dt$ as the input and output, respectively. This can be observed because the capacitor has the output current $i_C(t) = \omega C V_m \cos(\omega t)$ to an input voltage $v_C(t) = V_m \sin(\omega t)$ and the magnitude of the output becomes larger as the frequency ω of the input becomes higher. This is based on the fact that the AC admittance is $\mathbf{I}_C/\mathbf{V}_C = j\omega C$ whose magnitude becomes larger as ω becomes higher. This explains why a capacitor is connected in series with a load when there is a need to cut off DC (low-frequency) components in order to apply an AC (high-frequency) signal to the load (see the CR circuit in Figure 8.2(a)).

8.3 Bandpass Filter (BPF)

8.3.1 Series RLC Circuit and Series Resonance

Consider the series RLC circuit of Figure 8.3(a) in which the voltage $v_R(t)$ across the resistor R is taken as the output to the input voltage source $v_i(t)$. Its input–output relationship can be described by the transfer

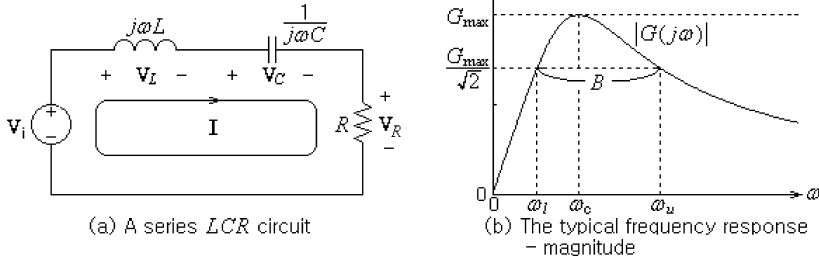


Figure 8.3 A bandpass filter and its typical frequency response

function and the frequency response as

$$G(s) = \frac{V_R(s)}{V_i(s)} = \frac{R}{sL + R + 1/(sC)} = \frac{s}{s^2 + sR/L + 1/(LC)} \quad (8.13a)$$

$$= \frac{\omega_b s}{s^2 + \omega_b s + \omega_p^2} \quad \text{with } \omega_b = \frac{R}{L} \text{ and } \omega_p = \frac{1}{\sqrt{LC}}$$

$$G(j\omega) = \frac{V_R}{V_i} = \frac{j\omega\omega_b}{(\omega_p^2 - \omega^2) + j\omega\omega_b} \quad (8.13b)$$

Referring to the typical magnitude curve of the frequency response of a *bandpass filter* (BPF) shown in Figure 8.3(b), the maximum of the magnitude, $|G(j\omega)|$, of the frequency response can be found by setting the derivative of its square $|G(j\omega)|^2$ w.r.t. the frequency ω to zero as

$$\frac{d}{d\omega} |G(j\omega)|^2 = \frac{d}{d\omega} \frac{\omega_b^2 \omega^2}{(\omega_p^2 - \omega^2)^2 + \omega_b^2 \omega^2} = \frac{2\omega_b^2 \omega (\omega_p^2 + \omega^2)(\omega_p^2 - \omega^2)}{[(\omega_p^2 - \omega^2)^2 + \omega_b^2 \omega^2]^2} = 0$$

This yields the *peak* or *center frequency*

$$\omega = \omega_p = \frac{1}{\sqrt{LC}} \quad (8.14)$$

at which the frequency response achieves the maximum magnitude as

$$G_{\max} = |G(j\omega_p)| = \frac{|\omega_b \omega|}{\sqrt{(\omega_p^2 - \omega^2)^2 + (\omega_b \omega)^2}} \Big|_{\omega=\omega_p} = 1$$

This frequency is also referred to as the *resonant frequency* ω_r in the sense that the frequency response is real so that the input and the output are *in phase* at that frequency. The lower and upper 3 dB frequencies can also be found at which the magnitude $|G(j\omega)|$ of this frequency response is $G_{\max}/\sqrt{2} = 1/\sqrt{2}$ as

$$|G(j\omega)|^2 = \frac{\omega_b^2 \omega^2}{(\omega_p^2 - \omega^2)^2 + \omega_b^2 \omega^2} = \frac{1}{2}$$

$$\omega_p^2 - \omega^2 = \pm \omega_b \omega; \quad \omega^2 \pm \omega_b \omega - \omega_p^2 = 0$$

$$\omega_l, \omega_u = \frac{\mp \omega_b + \sqrt{\omega_b^2 + 4\omega_p^2}}{2} = \mp \frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 + \frac{1}{LC}} \quad (8.15)$$

Note. The magnitudes of the frequency response of a BPF at the center frequency and the lower/upper 3 dB frequencies are 1, $1/\sqrt{2}$, and $1/\sqrt{2}$, respectively. The phases of the frequency response at the three frequencies turn out to be 0° (180°), 45° (-135°), and -45° (135°), respectively.

$$G(j\omega) = \frac{j\omega\omega_b}{(\omega_p^2 - \omega^2) + j\omega\omega_b} \bigg|_{\omega=\omega_u, \omega_l} = \frac{j\omega\omega_b}{\mp\omega\omega_b + j\omega\omega_b} = \pm 45^\circ$$

Multiplying these two 3 dB frequencies and taking the square root yields

$$\sqrt{\omega_l \omega_u} = \omega_p; \quad \frac{1}{2}(\log \omega_l + \log \omega_u) = \log \omega_p \quad (8.16)$$

This implies that the center/resonant frequency ω_r is the geometrical mean of two 3 dB frequencies ω_l and ω_u and also that the arithmetical mean of their logarithmic values is at the midpoint between the two 3 dB frequency points on the log-frequency ($\log \omega$) axis. The frequency band between the two 3 dB frequencies ω_l and ω_u is the *passband* and its width is the *bandwidth*:

$$B = \omega_u - \omega_l = \frac{R}{L} = \omega_b \quad (8.17)$$

Note that the bandwidth for a bandpass filter is the width of the range of frequency components that pass the filter better than others.

For a bandpass filter, the *quality factor* or *selectivity* (*sharpness*) is defined as

$$Q = \frac{\omega_r}{B} = \frac{\omega_r}{\omega_u - \omega_l} = \frac{1/\sqrt{LC}}{R/L} = \frac{1}{R} \sqrt{\frac{L}{C}} \quad (8.18)$$

This quality factor is referred to as the selectivity since it is a measure of how selectively the filter responds to the resonant frequency and frequencies near it within the passband, discriminating against frequencies outside the passband. It is also referred to as the sharpness in the sense that it describes how sharp the magnitude curve of the frequency response is around the peak frequency. It coincides with the *voltage magnification ratio*, i.e. the ratio of the magnitude of the voltage across the inductor or the capacitor to that of the input voltage at resonant frequency as

$$\frac{|\mathbf{V}_L|}{|\mathbf{V}_i|} = \frac{|\mathbf{V}_C|}{|\mathbf{V}_i|} = \frac{|j\omega_r L|}{|R + j\omega_r L + 1/(j\omega_r C)|} = \frac{\omega_r L = 1/(\omega_r C)}{|R + j[\omega_r L - 1/(\omega_r C)]|} = \frac{1}{R} \sqrt{\frac{L}{C}} \equiv Q \quad (8.19)$$

which may be greater than unity. It is interesting that the voltages across some components of a circuit can be higher than the applied voltage. One more thing to note is that if only the input source frequency in a series *RLC* circuit happens to be identical to the resonant frequency of the circuit, the two phasor voltages, one across the inductor and the other across the capacitor, have the same magnitude, but opposite directions (out of phase by 180°), so that their sum is zero:

$$\mathbf{V}_L + \mathbf{V}_C \big|_{\omega=\omega_r} = \left(j\omega_r L + \frac{1}{j\omega_r C} \right) \bigg|_{\omega=\omega_r} \mathbf{I} = j \left(\omega_r L - \frac{1}{\omega_r C} \right) \mathbf{I} \stackrel{(8.14)}{=} 0 \quad (8.20)$$

as if the input voltage is applied across the resistor only. Let us look at the following example.

(Example 8.1) Voltage Magnification of a Series RLC Circuit in Series Resonance

Consider the series RLC circuit consisting of $R = 100\ \Omega$, $L = 1\text{ H}$, and $C = 7.04\ \mu\text{F}$, where an AC voltage source with the rms amplitude of 120 V and the frequency of 60 Hz is applied:

$$v_i(t) = 120\sqrt{2}\sin(2\pi 60t) = 120\sqrt{2}\sin(377t)\text{ [V]} \quad (\text{E8.1.1})$$

Since the resonant frequency of the circuit

$$\omega_r \stackrel{(8.14)}{=} \frac{1}{\sqrt{LC}} = \frac{10^3}{\sqrt{7.04}} = 377\text{ rad/s} \quad (\text{E8.1.2})$$

is tuned to the input source frequency, this circuit is in *resonance* and thus use can be made of Equation (8.19) to obtain the maximum voltage across the inductor or capacitor as

$$V_{Lm} = V_{Cm} \stackrel{(8.19)}{=} \frac{\omega_r L}{R} V_{im} = \frac{377}{100} 120\sqrt{2} = 640\text{ V} \quad (\text{E8.1.3})$$

This is high enough to present the human body with a deadly shock hazard (Reference [H-2]).

Note. If you do not bear in mind that the voltages across some components of a circuit can be much higher than the applied voltage, you might be electrocuted by accident and obligated to take another boring course on circuits in the ‘heavenly’ university where the president, Mr Hades, likes to condemn students with lowest GPA to hell.

Here is an interpretation of *resonance*. As human beings enthusiastically do what they want to if only a single excuse is given, so a physical system responds to an input of its resonant frequency vigorously and even violently. What is the resonant frequency of a system? In fact, any system has one or more (undamped) resonant frequencies as long as its input–output relationship can be modeled by a second- or higher-order transfer function with complex poles (see Section 4.1.3). Since the side effects of resonance may be good or bad, we should be able to utilize the resonance to the purpose, say, of filtering and also take measures against possible bad side effects. For example, a singer may break a glass with his/her voice tone matched to the resonant frequency of the glass. Even a bridge can be broken by a periodic force of its resonant frequency that may be caused by wind or people crossing it. It is a well-known event that the first bridge across Tacoma Narrows at Puget Sound, Washington, was destroyed by wind on 7 November 1940, that is just 4 months after it was opened on 1 July 1940.

Now, let us see the following example in which a series RLC circuit is designed as a bandpass filter satisfying a specification on the passband.

(Example 8.2) A Design of a Series RLC Circuit as a Bandpass Filter (BPF)

Consider a series RLC circuit with the inductor of inductance $L = 12.9\text{ mH}$. Determine the values of the resistance R and the capacitance C such that the circuit can function as a BPF with the passband between the two frequencies 691 and 941 Hz .

Noting that the two boundary frequencies are given as the lower/upper 3 dB frequencies, we have

$$\omega_l = 2\pi \times 697 = 4379\text{ rad/s} \quad \text{and} \quad \omega_u = 2\pi \times 941 = 5912\text{ rad/s} \quad (\text{E8.2.1})$$

First, these upper/lower 3 dB frequencies are substituted into Equation (8.16) to get the resonant frequency as

$$\omega_r \stackrel{(8.16)}{=} \sqrt{\omega_u \omega_l} = \sqrt{5912 \times 4379} = 5088\text{ rad/s} \quad (\text{E8.2.2})$$

Then Equation (8.14) for the resonant frequency is used to find the value of C such that the resonant frequency of the circuit will be 5088 rad/s :

$$C \stackrel{(8.14)}{=} \frac{1}{L \omega_r^2} = \frac{1}{0.0129 \times 4379 \times 5912} = 3\ \mu\text{F} \quad (\text{E8.2.3})$$

Equation (8.17) for the bandwidth is also used to find the value of R such that the bandwidth of the circuit will be $\omega_u - \omega_l$ [rad/s]:

$$R \stackrel{(8.17)}{=} BL = (\omega_u - \omega_l)L = (5912 - 4379) \times 0.0129 = 19.8 \Omega \quad (\text{E8.2.4})$$

8.3.2 Parallel RLC Circuit and Parallel Resonance

Consider the parallel RLC circuit of Figure 8.4(a) in which the current $i_R(t)$ through the resistor R is taken as the output to the input current source $i_i(t)$. Its input–output relationship can be described by the transfer function and the frequency response as

$$G(s) = \frac{I_R(s)}{I_i(s)} = \frac{1/R}{1/R + sC + 1/(sL)} = \frac{s/(RC)}{s^2 + s/(RC) + 1/(LC)} \quad (8.21a)$$

$$= \frac{\omega_b s}{s^2 + \omega_b s + \omega_p^2} \quad \text{with} \quad \omega_b = \frac{1}{RC} \quad \text{and} \quad \omega_p = \frac{1}{\sqrt{LC}}$$

$$G(j\omega) = \frac{I_R}{I_i} = \frac{j\omega\omega_b}{(\omega_p^2 - \omega^2) + j\omega\omega_b} \quad (8.21b)$$

Since this frequency response is the same as that, Equation (8.13b), of the series RLC circuit in Figure 8.3(a) except for ω_b , the *peak or center frequency*, the *upper/lower 3 dB frequencies*, the *bandwidth*, and the *quality factor* can be obtained as follows:

$$\omega = \omega_p = \frac{1}{\sqrt{LC}} \quad (8.22a)$$

$$\omega_l, \omega_u = \frac{\mp \omega_b + \sqrt{\omega_b^2 + 4\omega_p^2}}{2} = \mp \frac{1}{2RC} + \sqrt{\left(\frac{1}{2RC}\right)^2 + \frac{1}{LC}} \quad (8.22b)$$

$$B = \omega_u - \omega_l = \frac{1}{RC} = \omega_b \quad (8.22c)$$

$$Q = \frac{\omega_p}{B} = \frac{\omega_p}{\omega_u - \omega_l} = \frac{1/\sqrt{LC}}{1/(RC)} = R\sqrt{\frac{C}{L}} \quad (8.22d)$$

The peak frequency is also referred to as the *resonant frequency* ω_r in the sense that the frequency response is real so that the input and the output are *in phase* at that frequency. The quality factor coincides with the *current magnification ratio*, i.e. the ratio of the amplitude of the current through the inductor or the capacitor to that of the input current at resonant frequency, as

$$\frac{|I_L|}{|I_i|} = \frac{|I_C|}{|I_i|} = \frac{|1/(j\omega_r L)|}{|1/R + 1/(j\omega_r L) + j\omega_r C|} = \frac{1/(\omega_r L) = \omega_r C}{|1/R + j(\omega_r C - 1/\omega_r L)|} = R\sqrt{\frac{C}{L}} \equiv Q \quad (8.23)$$

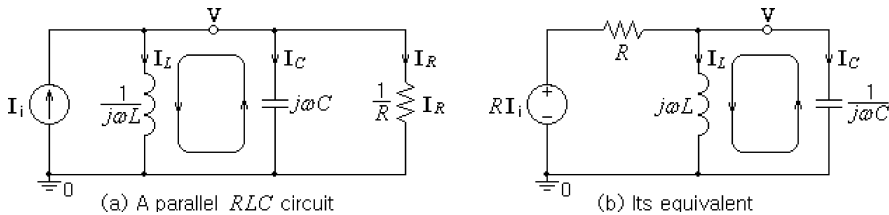


Figure 8.4 A parallel circuit and its equivalent

which may be greater than unity. How strange it is that the currents through some components of a circuit can be larger than the applied current! One more thing to note is that if only the input source frequency in a parallel RLC circuit happens to be identical to the resonant frequency of the circuit, the two phasor currents, one through the inductor and the other through the capacitor, have the same magnitude, but opposite directions (out of phase by 180°) so that their sum is zero:

$$\mathbf{I}_L + \mathbf{I}_C|_{\omega=\omega_r} = \left(\frac{1}{j\omega L} + j\omega C \right) \bigg|_{\omega=\omega_r} \mathbf{V} = j \left(\omega_r C - \frac{1}{\omega_r L} \right) \mathbf{V} \stackrel{(8.22a)}{=} 0 \quad (8.24)$$

This let us imagine that the input current is all applied through the resistor only. The two currents $\mathbf{I}_L$ and $\mathbf{I}_C$ are denoted as the current circulating around the mesh consisting of L and C in Figure 8.4(a), where the mesh is called an LC tank.

Now consider the RLC circuit of Figure 8.4(b) in which the voltage across $L||C$ (the parallel combination of L and C) is taken as the output to the input voltage source $R\mathbf{I}_i$. The voltage divider rule can be used to obtain the transfer function as

$$G(s) = \frac{V_{LC}(s)}{R\mathbf{I}_i(s)} = \frac{sL|(1/sC)}{R + [sL|(1/sC)]} = \frac{(L/C)/[sL + 1/(sC)]}{R + (L/C)/[sL + 1/(sC)]} = \frac{s/(RC)}{s^2 + s/(RC) + 1/(LC)}$$

This is the same as Equation (8.21a), which is the transfer function of the parallel RLC circuit of Figure 8.4(a). In fact, the circuit of Figure 8.4(b) is obtained by transforming the current source $\mathbf{I}_i$ in parallel with R into a voltage source $R\mathbf{I}_i$ in series with R in the circuit of Figure 8.4(a).

(Example 8.3) Current Magnification of a Parallel RLC Circuit in Parallel Resonance

Consider the parallel RLC circuit consisting of $R = 100 \text{ k}\Omega$, $L = 1 \text{ H}$, and $C = 1 \text{ }\mu\text{F}$, where an AC current source $i_i(t)$ with the amplitude of 10 mA and the angular frequency of 1000 rad/s is applied:

$$i_i(t) = 0.01 \cos(10^3 t) [\text{A}] \quad (\text{E8.3.1})$$

Since the resonant frequency of the circuit

$$\omega_r \stackrel{(8.22a)}{=} \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{10^{-6}}} = 10^3 \text{ rad/s} \quad (\text{E8.3.2})$$

is tuned to the input source frequency, this circuit is in *resonance* and thus use can be made of Equation (8.23) to obtain the maximum current through the inductor or the capacitor as

$$I_{Lm} = I_{Cm} \stackrel{(8.23)}{=} \frac{R}{\omega_r L} I_{im} = \frac{10^5}{10^3} 0.01 = 1 \text{ A} \quad (\text{E8.3.3})$$

This may be large enough to damage some electronic components.

[Remark 8.3] Step Response of a BPF (Bandpass Filter)

The *step response* of a system is defined to be the output of the system to an input that can be described by the unit step function $u_s(t)$. Consider a BPF whose transfer function is

$$G(s) = \frac{\omega_b s}{s^2 + \omega_b s + \omega_p^2} = \frac{\omega_b s}{(s + \sigma)^2 + \omega_d^2} \quad \text{with } \omega_d = \sqrt{\omega_p^2 - (\omega_b/2)^2}, \quad D = \omega_b^2 - 4\omega_p^2 < 0$$

Noting that the Laplace transform of the unit step function is $\mathcal{L}\{u_s(t)\} \stackrel{\text{Table A.1(3)}}{=} 1/s$, the step response of the system can be obtained as

$$Y(s) = G(s)X(s) = \frac{\omega_b s}{s^2 + \omega_b s + \omega_p^2} \frac{1}{s} = \frac{\omega_b}{\omega_d} \frac{\omega_d}{(s + \sigma)^2 + \omega_d^2}$$

$$y(t) = \mathcal{L}^{-1}\{Y(s)\} \stackrel{\text{Table A.1(9)}}{=} \frac{\omega_b}{\omega_d} e^{-\sigma t} \sin(\omega_d t) u_s(t)$$

[Remark 8.4] Filtering Characteristic of *RLC* Circuits

1. The series *RLC* circuit functions as a lowpass filter if the voltage across the capacitor is taken as the output, because the frequency response is

$$G(j\omega) = \frac{V_C}{V_i} = \frac{1/(j\omega C)}{j\omega L + R + 1/(j\omega C)} = \frac{1/(LC)}{[1/(LC) - \omega^2] + j\omega R/L} \quad (8.25)$$

A justification is that a capacitor with its current/voltage as the input/output, respectively, has the frequency response as

$$G_C(j\omega) = \frac{V_C}{I} = \frac{1}{j\omega C}$$

which can be regarded as the frequency response of a lowpass filter.

2. The series *RLC* circuit functions as a highpass filter if the voltage across the inductor is taken as the output, because the frequency response is

$$G(j\omega) = \frac{V_L}{V_i} = \frac{j\omega L}{j\omega L + R + 1/(j\omega C)} = \frac{-\omega^2}{[1/(LC) - \omega^2] + j\omega R/L} \quad (8.26)$$

A justification is that an inductor with its current/voltage as the input/output, respectively, has the frequency response as

$$G_L(j\omega) = \frac{V_L}{I} = j\omega L$$

which can be regarded as the frequency response of a highpass filter.

3. The parallel *RLC* circuit functions as a lowpass filter if the current through the inductor is taken as the output, because the frequency response is

$$G(j\omega) = \frac{I_L}{I_i} = \frac{1/(j\omega L)}{j\omega C + 1/R + 1/(j\omega L)} = \frac{1/(LC)}{[1/(LC) - \omega^2] + j\omega/(RC)} \quad (8.27)$$

A justification is that an inductor with its voltage/current as the input/output, respectively, has the frequency response as

$$G_L(j\omega) = \frac{I_C}{V} = \frac{1}{j\omega L}$$

which can be regarded as the frequency response of a lowpass filter.

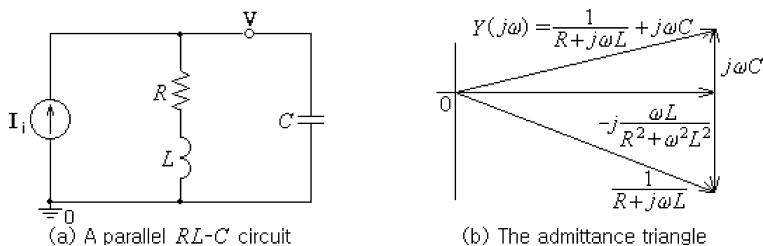


Figure 8.5 A practical parallel resonant circuit

4. The parallel RLC circuit functions as a highpass filter if the current through the capacitor is taken as the output, because the frequency response is

$$G(j\omega) = \frac{\mathbf{I}_C}{\mathbf{I}_i} = \frac{j\omega C}{j\omega C + 1/R + 1/(j\omega L)} = \frac{-\omega^2}{[1/(LC) - \omega^2] + j\omega/(RC)} \quad (8.28)$$

A justification is that a capacitor with its voltage/current as the input/output, respectively, has the frequency response as

$$G_C(j\omega) = \frac{\mathbf{I}_C}{\mathbf{V}} = j\omega C$$

which can be regarded as the frequency response of a highpass filter.

[Remark 8.5] Derivation of the Resonance Condition for a BPF

Instead of setting the derivative of the squared magnitude of the frequency response to zero, as in Section 8.3.1, the *resonance condition* for a bandpass filter can be derived by setting the imaginary part of the impedance or admittance to zero, as suggested by Equation (8.20) or (8.24).

(Example 8.4) Practical Resonance Condition

As suggested in Remark 8.5, the resonance condition of the circuit of Figure 8.5(a) can be obtained by setting the imaginary part of the impedance or admittance to zero. Since the admittance is easier to compute than the impedance for this circuit, the imaginary part of the admittance is set to zero so that the frequency response will be real:

$$Y(j\omega) = \frac{1}{R + j\omega L} + j\omega C = \frac{R}{R^2 + \omega^2 L^2} + j\omega \left(C - \frac{L}{R^2 + \omega^2 L^2} \right) \quad (E8.4.1)$$

$$\omega \left(C - \frac{L}{R^2 + \omega^2 L^2} \right) = 0; \quad C - \frac{L}{R^2 + \omega^2 L^2} = 0$$

This yields the resonant frequency as

$$\omega = \sqrt{\frac{1}{LC} - \frac{R^2}{L^2}} \quad (E8.4.2)$$

8.4 Bandstop Filter (BSF)

8.4.1 Series RLC Circuit

Consider the series RLC circuit of Figure 8.6(a) in which the voltage $v_{LC}(t)$ across the series combination of L and C is taken as the output to the input voltage source $v_i(t)$. Its input–output relationship can be

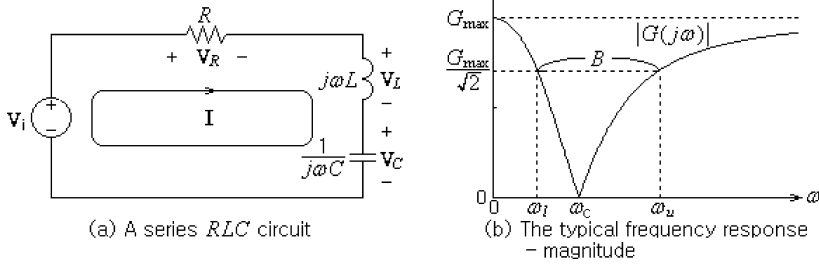


Figure 8.6 A bandstop filter and its typical frequency response

described by the transfer function and the frequency response as

$$G(s) = \frac{V_L(s) + V_C(s)}{V_i(s)} = \frac{sL + 1/(sC)}{sL + R + 1/(sC)} = \frac{s^2 + 1/(LC)}{s^2 + sR/L + 1/(LC)} \quad (8.29a)$$

$$= \frac{s^2 + \omega_0^2}{s^2 + \omega_b s + \omega_0^2} \quad \text{with} \quad \omega_b = \frac{R}{L} \quad \text{and} \quad \omega_0 = \frac{1}{\sqrt{LC}}$$

$$G(j\omega) = \frac{\mathbf{V}_R}{\mathbf{V}_i} = \frac{\omega_0^2 - \omega^2}{(\omega_0^2 - \omega^2) + j\omega\omega_b} \quad (8.29b)$$

Referring to the typical magnitude curve of the frequency response of a bandstop filter shown in Figure 8.6(b), the minimum of the magnitude of the frequency response can be found by setting the derivative of its square w.r.t. the frequency ω to zero as

$$\frac{d}{d\omega} |G(j\omega)|^2 = \frac{d}{d\omega} \frac{(\omega_0^2 - \omega^2)^2}{(\omega_0^2 - \omega^2)^2 + \omega_b^2 \omega^2} = -\frac{2\omega(\omega_0^2 - \omega^2)(\omega_0^2 + \omega^2)\omega_b^2}{[(\omega_0^2 - \omega^2)^2 + \omega_b^2 \omega^2]^2} = 0$$

This yields the *notch* or *rejection* or *center* frequency

$$\omega = \omega_r = \omega_0 = \frac{1}{\sqrt{LC}} \quad (8.30)$$

at which the frequency response achieves the minimum magnitude as

$$G_{\min} = |G(j\omega_0)| = \frac{|\omega_0^2 - \omega^2|}{\sqrt{(\omega_0^2 - \omega^2)^2 + (\omega_b \omega)^2}} \bigg|_{\omega=\omega_r=\omega_0} = 0$$

Noting that the magnitude of the frequency response achieves its maximum of unity at another extremum frequency $\omega = 0$ or $\omega = \infty$ as

$$G_{\max} = |G(j0)| = \frac{|\omega_0^2 - \omega^2|}{\sqrt{(\omega_0^2 - \omega^2)^2 + (\omega_b \omega)^2}} \bigg|_{\omega=0} = 1$$

we can find the lower and upper 3 dB frequencies at which the magnitude $|G(j\omega)|$ of this frequency response is $G_{\max}/\sqrt{2} = 1/\sqrt{2}$ as

$$\begin{aligned} |G(j\omega)|^2 &= \frac{(\omega_0^2 - \omega^2)^2}{(\omega_0^2 - \omega^2)^2 + \omega_b^2 \omega^2} = \frac{1}{2} \\ \omega_0^2 - \omega^2 &= \pm \omega_b \omega; \quad \omega^2 \pm \omega_b \omega - \omega_0^2 = 0 \\ \omega_l, \omega_u &= \frac{\mp \omega_b + \sqrt{\omega_b^2 + 4\omega_0^2}}{2} = \mp \frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 + \frac{1}{LC}} \end{aligned}$$

Multiplying these two 3 dB frequencies and taking the square root yields

$$\sqrt{\omega_l \omega_u} = \omega_0 = \omega_r, \quad \frac{1}{2}(\log \omega_l + \log \omega_u) = \log \omega_r \quad (8.31)$$

This implies that the notch frequency ω_r is the geometrical mean of two 3 dB frequencies ω_l and ω_u and also the arithmetical mean of their logarithmic values, which is at the midpoint between the two 3 dB frequency points on the log-frequency ($\log \omega$) axis. The band of frequencies between the two 3 dB frequencies ω_l and ω_u is the *stopband* and its width is the *bandwidth*:

$$B = \omega_u - \omega_l = \frac{R}{L} = \omega_b \quad (8.32)$$

Note that the bandwidth for a bandstop filter is the width of the range of frequency components that are relatively more attenuated or rejected by the filter than other frequencies.

For a bandstop filter, the *quality factor* or *selectivity (sharpness)* can be defined as

$$Q = \frac{\omega_r}{B} = \frac{\omega_r}{\omega_u - \omega_l} = \frac{1/\sqrt{LC}}{R/L} = \frac{1}{R} \sqrt{\frac{L}{C}} \quad (8.33)$$

This quality factor is referred to as the selectivity since it is a measure of how selectively the filter rejects the notch frequency and frequencies near it within the stopband, favoring frequencies outside the stopband. It is also referred to as the sharpness in the sense that it describes how sharp the magnitude curve of the frequency response is around the rejection or notch frequency.

(Example 8.5) A Design of a Series *RLC* Circuit as a Bandstop Filter (BSF)

Consider a series *RLC* circuit with the inductor of inductance $L = 13.9$ mH. Determine the values of the resistance R and the capacitance C such that the circuit can function as a BSF with the stopband between the two frequencies 637 and 1432 Hz.

Noting that the two boundary frequencies are given as the lower/upper 3 dB frequencies,

$$\omega_l = 2\pi \times 637 = 4000 \text{ rad/s} \quad \text{and} \quad \omega_u = 2\pi \times 1432 = 9000 \text{ rad/s} \quad (\text{E8.5.1})$$

First, these upper/lower 3 dB frequencies are substituted into Equation (8.31) to get the notch frequency as

$$\omega_r \stackrel{(8.31)}{=} \sqrt{\omega_u \omega_l} = \sqrt{4000 \times 9000} = 6000 \text{ rad/s} \quad (\text{E8.5.2})$$

Then Equation (8.30) for the notch frequency is used to find the value of C such that the notch frequency of the circuit will be 6000 rad/s:

$$C \stackrel{(8.30)}{=} \frac{1}{L\omega_i^2} = \frac{1}{0.0139 \times 4000 \times 9000} = 2 \mu\text{F} \quad (\text{E8.5.3})$$

Equation (8.32) for the bandwidth is also used to find the value of R such that the bandwidth of the circuit will be $\omega_u - \omega_l$ [rad/s]:

$$R \stackrel{(8.32)}{=} BL = (\omega_u - \omega_l)L = (9000 - 4000) \times 0.0139 = 69.5 \Omega \quad (\text{E8.5.4})$$

8.4.2 Parallel RLC Circuit

Consider the parallel RLC circuit of Figure 8.7(a) in which the current through the parallel combination of L and C , $i_L(t) + i_C(t)$, is taken as the output to the input current source $i_i(t)$. Its input–output relationship can be described by the transfer function and the frequency response as

$$\begin{aligned} G(s) &= \frac{I_L(s) + I_C(s)}{I_i(s)} = \frac{1/(sL) + sC}{1/R + s/(RC) + 1/(LC)} = \frac{s^2 + 1/(LC)}{s^2 + s/(RC) + 1/(LC)} \quad (8.34a) \\ &= \frac{s^2 + \omega_0^2}{s^2 + \omega_b s + \omega_0^2} \quad \text{with} \quad \omega_b = \frac{1}{RC} \quad \text{and} \quad \omega_0 = \frac{1}{\sqrt{LC}} \end{aligned}$$

$$G(j\omega) = \frac{\mathbf{I}_R}{\mathbf{I}_i} = \frac{\omega_0^2 - \omega^2}{(\omega_0^2 - \omega^2) + j\omega\omega_b} \quad (8.34b)$$

Since this frequency response is the same as that, Equation (8.29b), of the series RLC circuit in Figure 8.6(a) except for ω_b , the same results are obtained for the notch or rejection or center frequency, the upper/lower 3 dB frequencies, the bandwidth, and the quality factor.

Now consider the RLC circuit of Figure 8.7(b) in which the voltage across R is taken as the output to the input voltage source $\mathbf{V}_i$. The voltage divider rule can be used to obtain the transfer function as

$$G(s) = \frac{V_R(s)}{RI_i(s)} = \frac{R}{R + [sL \parallel (1/sC)]} = \frac{s^2 + 1/(LC)}{s^2 + s/(RC) + 1/(LC)}$$

This is exactly the same as Equation (8.34a), which is the transfer function of the parallel RLC circuit of Figure 8.7(a). In fact, this circuit is obtained by transforming the current source $\mathbf{I}_i$ in parallel with R into a voltage source $\mathbf{R}\mathbf{I}_i$ in series with R in the circuit of Figure 8.7(a).

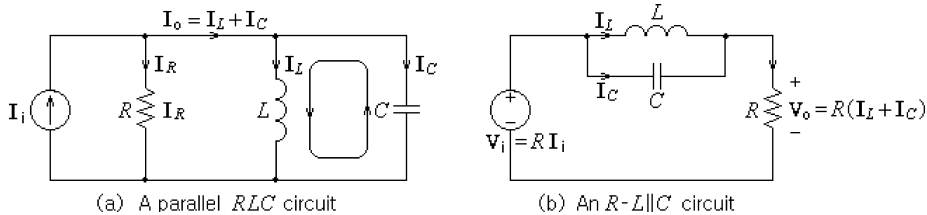


Figure 8.7 A bandstop filter

8.5 Active Filter

8.5.1 First-Order Active Filter

Consider the circuit of Figure 8.8(a) in which the OP Amp has a negative feedback path between the output terminal and the negative input terminal N. By the short principle (Remark 1.2(2)) the voltage at node N is (almost) zero, being the same as that at node P (the positive input terminal), which is grounded. Thus the current flowing from the input $V_i(s)$ to node N (whose voltage is zero) is $I_i(s) = V_i(s)/R_1$ and this current flows through $Z_f(s) = R_2 || [1/(sC)]$ (the parallel combination of R_2 and C) towards the output terminal since no current flows into or out of the negative input terminal of the OP Amp by the open principle (Remark 1.2(1)). Consequently, the output voltage is obtained by subtracting the voltage drop across Z_f from zero (the voltage at node N) as

$$V_o(s) = 0 - Z_f(s)I_i(s) = -Z_f(s) \frac{V_i(s)}{R_1}$$

and the transfer function of the OP Amp circuit turns out to be

$$\begin{aligned} G(s) &= \frac{V_o(s)}{V_i(s)} = -\frac{Z_f(s)}{R_1} = -\frac{R_2 || [1/(sC)]}{R_1} = -\frac{R_2 [1/(sC)]}{R_1 [R_2 + 1/(sC)]} = -\frac{R_2}{R_1} \frac{1/(R_2 C)}{s + 1/(R_2 C)} \\ G(s) &= -K \frac{\omega_c}{s + \omega_c} \quad \text{with} \quad K = \frac{R_2}{R_1} \quad \text{and} \quad \omega_c = 1/(R_2 C) \end{aligned} \quad (8.35)$$

This indicates that the OP Amp circuit of Figure 8.8(a) works as a lowpass filter (LPF) with the cutoff frequency of $\omega_c = 1/(R_2 C)$ and the DC gain of $-K = -R_2/R_1$ for $s = j\omega = 0$.

The circuit of Figure 8.8(b) is the same as that of Figure 8.8(a) except that R_1 and Z_f are replaced by Z_i and R_2 , respectively. Therefore it has the transfer function

$$\begin{aligned} G(s) &= \frac{V_o(s)}{V_i(s)} = -\frac{R_2}{Z_i(s)} = -\frac{R_2}{R_1 + 1/(sC)} = -\frac{R_2}{R_1} \frac{s}{s + 1/(R_1 C)} \\ G(s) &= -K \frac{s}{s + \omega_c} \quad \text{with} \quad K = \frac{R_2}{R_1} \quad \text{and} \quad \omega_c = \frac{1}{R_1 C} \end{aligned} \quad (8.36)$$

This indicates that the OP Amp circuit in Figure 8.8(b) works as a highpass filter (HPF) with the cutoff frequency of $\omega_c = 1/(R_1 C)$ and the DC gain of zero for $s = j\omega = 0$.

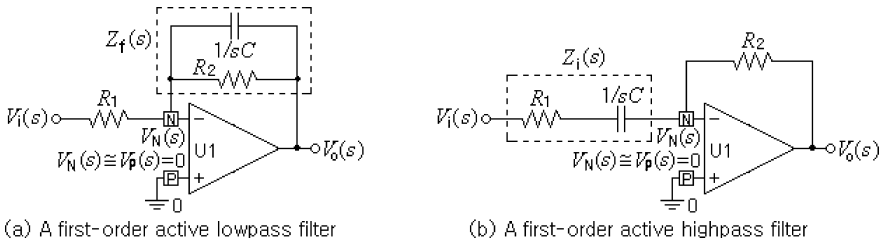


Figure 8.8 First-order active filters

8.5.2 Second-Order Active LPF/HPF

Consider the Sallen–Key circuits of Figure 8.9, in each of which the OP Amp has a negative feedback path between the output terminal and the negative input terminal N and the output voltage is $K = (R_3 + R_4)/R_3$ times the voltage at the positive input terminal P of the OP Amp as

$$V_o(s) = \frac{R_3 + R_4}{R_3} V_N(s) \stackrel{\text{short principle}}{=} K V_P(s) \quad \text{with} \quad K = \frac{R_3 + R_4}{R_3} \quad (8.37)$$

First, for the circuit of Figure 8.9(a), KCL is applied to node 1 and P to write the node equations as

$$\begin{aligned} \frac{V_1 - V_i}{R_1} + \frac{V_1 - V_P}{R_2} + \frac{V_1 - V_o}{1/(sC_1)} &= 0 \\ \frac{V_P - V_1}{R_2} + \frac{V_P}{1/sC_2} &= 0 \end{aligned}$$

where (s) has been omitted from $V_i(s)$, $V_1(s)$, $V_P(s)$, and $V_o(s)$ to write them as V_i , V_1 , V_P , or V_o for simplicity. With $V_o = KV_P$, this set of node equations is arranged in matrix–vector form as

$$\begin{bmatrix} G_1 + G_2 + sC_1 & -G_2 - sC_1K \\ -G_2 & G_2 + sC_2 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_P(s) \end{bmatrix} = \begin{bmatrix} G_1 V_i(s) \\ 0 \end{bmatrix}$$

This can be solved to get V_P , $V_o = KV_P$, and finally the transfer function $G(s)$ as

$$\begin{aligned} \begin{bmatrix} V_1(s) \\ V_P(s) \end{bmatrix} &= \frac{1}{\Delta} \begin{bmatrix} G_2 + sC_2 & G_2 + sC_1K \\ G_2 & G_1 + G_2 + sC_1 \end{bmatrix} \begin{bmatrix} G_1 V_i(s) \\ 0 \end{bmatrix} \\ \text{with } \Delta &= C_1 C_2 s^2 + [C_2(G_1 + G_2) + (1 - K)C_1 G_2]s + G_1 G_2 \\ V_o(s) = K V_P(s) &= \frac{KG_1 G_2}{C_1 C_2 s^2 + [C_2(G_1 + G_2) + (1 - K)C_1 G_2]s + G_1 G_2} V_i(s) \\ G(s) = \frac{V_o(s)}{V_i(s)} &= \frac{KG_1 G_2 / (C_1 C_2)}{s^2 + [(G_1 + G_2)/C_1 + (1 - K)G_2/C_2]s + G_1 G_2 / C_1 C_2} \quad (8.38) \end{aligned}$$

Having only a constant term in the numerator, this transfer function indicates that the circuit will function as a lowpass filter (LPF).

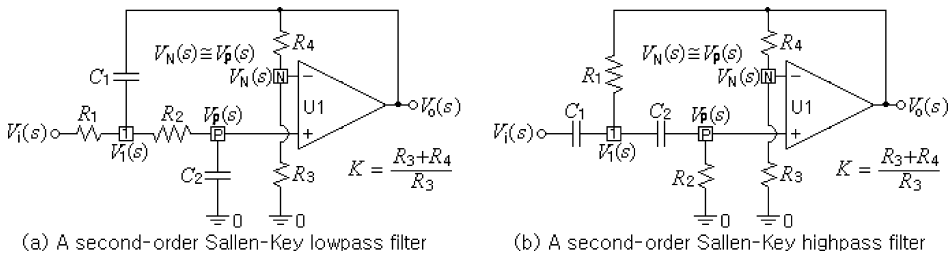


Figure 8.9 Second-order active filters

Second, for the circuit in Figure 8.9(b), KCL is applied to the nodes 1 and P to write the node equations as

$$\begin{aligned}\frac{V_1 - V_i}{1/(sC_1)} + \frac{V_1 - V_P}{1/(sC_2)} + \frac{V_1 - V_o}{R_1} &= 0 \\ \frac{V_P - V_1}{1/sC_2} + \frac{V_P}{R_2} &= 0\end{aligned}$$

With $V_o \stackrel{(8.37)}{=} KV_P$, this set of node equations can be arranged in matrix–vector form as

$$\begin{bmatrix} sC_1 + sC_2 + G_1 & -sC_2 - G_1K \\ -sC_2 & sC_2 + G_2 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_P(s) \end{bmatrix} = \begin{bmatrix} sC_1 V_i(s) \\ 0 \end{bmatrix}$$

This can be solved to get V_P , $V_o = KV_P$, and finally the transfer function $G(s)$ as

$$\begin{aligned}\begin{bmatrix} V_1(s) \\ V_P(s) \end{bmatrix} &= \frac{1}{\Delta} \begin{bmatrix} sC_2 + G_2 & sC_2 + G_1K \\ sC_2 & G_1 + G_2 + sC_1 \end{bmatrix} \begin{bmatrix} sC_1 V_i(s) \\ 0 \end{bmatrix} \\ \text{with } \Delta &= C_1 C_2 s^2 + [G_2(C_1 + C_2) + (1 - K)G_1 C_2]s + G_1 G_2 \\ V_o(s) = KV_P(s) &= \frac{KC_1 C_2 s^2}{C_1 C_2 s^2 + [G_2(C_1 + C_2) + (1 - K)G_1 C_2]s + G_1 G_2} V_i(s) \\ G(s) = \frac{V_o(s)}{V_i(s)} &= \frac{K s^2}{s^2 + [G_2(1/C_1 + 1/C_2) + (1 - K)G_1 / C_1]s + G_1 G_2 / C_1 C_2} \quad (8.39)\end{aligned}$$

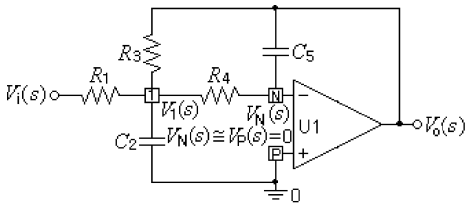
Having only a second-degree term in the numerator, this transfer function indicates that the circuit will function as a highpass filter (HPF).

Now consider the MFB (multiple/dual feedback) circuits of Figures 8.10(a) and (b) where the OP Amp has a negative feedback path so that by the short principle, the voltage at node N is (almost) zero, being the same as that at node P (the positive input terminal), which is grounded. KCL is applied to nodes 1 and N of the circuit of figure 8.10(a) to write the node equations as

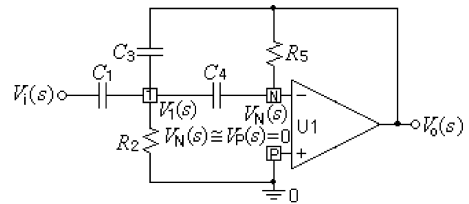
$$\begin{aligned}\frac{V_1 - V_i}{R_1} + \frac{V_1}{1/(sC_2)} + \frac{V_1 - V_o}{R_3} + \frac{V_1}{R_4} &= 0 \\ \frac{0 - V_1}{R_4} + \frac{0 - V_o}{1/(sC_5)} &= 0\end{aligned}$$

which can be arranged in compact (matrix–vector) form as

$$\begin{bmatrix} G_1 + G_3 + G_4 + sC_2 & -G_3 \\ G_4 & sC_5 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_o(s) \end{bmatrix} = \begin{bmatrix} G_1 V_i(s) \\ 0 \end{bmatrix}$$



(a) A second-order MFB (multi-feedback) LPF



(b) A second-order MFB (multi-feedback) HPF

Figure 8.10 Second-order active filters

This can be solved to get V_o and finally the transfer function $G(s)$ as

$$\begin{aligned} \begin{bmatrix} V_1(s) \\ V_o(s) \end{bmatrix} &= \frac{1}{\Delta} \begin{bmatrix} sC_5 & G_3 \\ -G_4 & G_1 + G_3 + G_4 + sC_2 \end{bmatrix} \begin{bmatrix} G_1 V_i(s) \\ 0 \end{bmatrix} \\ \text{with } \Delta &= C_2 C_5 s^2 + C_5 (G_1 + G_3 + G_4) s + G_3 G_4 \\ G(s) &= \frac{V_o(s)}{V_i(s)} = \frac{-G_1 G_4}{C_2 C_5 s^2 + C_5 (G_1 + G_3 + G_4) s + G_3 G_4} \end{aligned} \quad (8.40)$$

This transfer function having only a constant term in the numerator indicates that the circuit will function as a lowpass filter (LPF).

Last, the MFB (multiple/dual feedback) circuit of Figure 8.10(b) is considered, where the OP Amp has a negative feedback path so that by the short principle, the voltage at node N is (almost) zero, being the same as that at node P (the positive input terminal), which is grounded. KCL is applied to nodes 1 and N to write the node equations as

$$\frac{V_1 - V_i}{1/(sC_1)} + \frac{V_1}{R_2} + \frac{V_1 - V_o}{1/(sC_3)} + \frac{V_1}{1/(sC_4)} = 0 \quad \text{and} \quad \frac{0 - V_1}{1/(sC_4)} + \frac{0 - V_o}{R_5} = 0$$

which can be arranged in compact (matrix–vector) form as

$$\begin{bmatrix} sC_1 + sC_3 + sC_4 + G_2 & -sC_3 \\ sC_4 & G_5 \end{bmatrix} \begin{bmatrix} V_1(s) \\ V_o(s) \end{bmatrix} = \begin{bmatrix} sC_1 V_i(s) \\ 0 \end{bmatrix}$$

This can be solved to get V_o and finally the transfer function $G(s)$ as

$$\begin{aligned} \begin{bmatrix} V_1(s) \\ V_o(s) \end{bmatrix} &= \frac{1}{\Delta} \begin{bmatrix} G_5 & sC_3 \\ -sC_4 & s(C_1 + C_3 + C_4) + G_2 \end{bmatrix} \begin{bmatrix} sC_1 V_i(s) \\ 0 \end{bmatrix} \\ \text{with } \Delta &= C_3 C_4 s^2 + G_5 (C_1 + C_3 + C_4) s + G_2 G_5 \\ G(s) &= \frac{V_o(s)}{V_i(s)} = \frac{-C_1 C_4 s^2}{C_3 C_4 s^2 + G_5 (C_1 + C_3 + C_4) s + G_2 G_5} \end{aligned} \quad (8.41)$$

Having only a second-degree term in the numerator, this transfer function indicates that the circuit will function as a highpass filter (HPF).

8.5.3 Second-Order Active BPF

Noting that the circuit in Figure 8.11(a) is the same as that in Figure 8.10(b) except that C_1 is replaced by R_1 , its transfer function can be written as

$$\begin{aligned} G(s) &= \frac{V_o(s)}{V_i(s)} = \frac{-G_1 C_4 s}{C_3 C_4 s^2 + G_5 (C_3 + C_4) s + (G_1 + G_2) G_5} \\ &= \frac{-(G_1/C_3) s}{s^2 + [G_5 (C_3 + C_4)/(C_3 C_4)] s + (G_1 + G_2) G_5/(C_3 C_4)} \end{aligned} \quad (8.42)$$

This transfer function having only a first-degree term in the numerator indicates that the circuit will function as a bandpass filter (BPF).

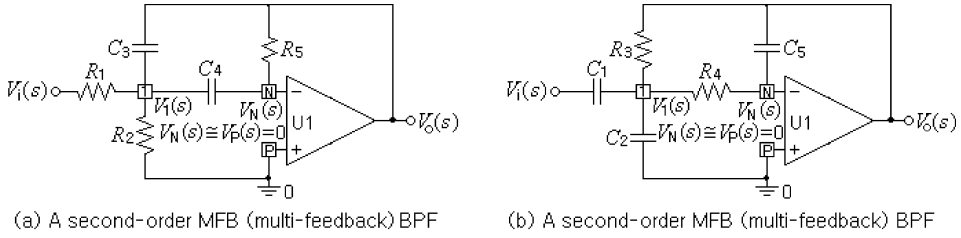


Figure 8.11 Second-order active filters

Now, noting that the circuit in Figure 8.11(b) is the same as that in Figure 8.11(a), except that R and C are exchanged, its transfer function can be written as

$$\begin{aligned}
 G(s) = \frac{V_o(s)}{V_i(s)} &= \frac{-C_1 G_4 s}{(C_1 + C_2) C_5 s^2 + C_5 (G_3 + G_4) s + G_3 G_4} \\
 &= \frac{-[C_1 G_4 / (C_1 + C_2) C_5] s}{s^2 + [(G_3 + G_4) / (C_1 + C_2)] s + G_3 G_4 / (C_1 + C_2) C_5} \quad (8.43)
 \end{aligned}$$

Having only a first-degree term in the numerator, this transfer function indicates that the circuit will also function as a bandpass filter (BPF).

8.5.4 Second-Order Active BSF

To analyze the circuit shown in Figure 8.12 (from Reference [N-1]), KCL is applied to nodes 1, 2, and 3 to write the node equations

$$\begin{aligned}
 \text{Node 1:} \quad & \frac{V_1 - V_i}{R} + \frac{V_1 - V_o}{R} + \frac{V_1 - \sigma V_o}{1/(2sC)} = 0 \\
 \text{Node 2:} \quad & \frac{V_2 - V_i}{1/(sC)} + \frac{V_2 - V_o}{1/(sC)} + \frac{V_2 - \sigma V_o}{R/2} = 0 \\
 \text{Node 3:} \quad & \frac{V_o - V_1}{R} + \frac{V_o - V_2}{1/(sC)} = 0
 \end{aligned}$$

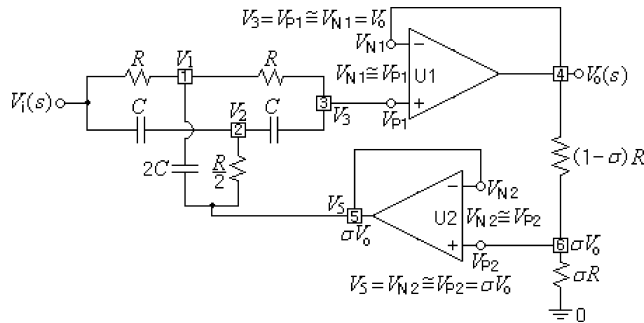


Figure 8.12 A second-order active (twin-T) bandstop filter (Source: J.W. Nilsson and S.A. Riedel, *Electric Circuits*, 5th edition, 1996. Source: © Addison-Wesley)

which can be arranged in compact (matrix–vector) form as

$$\begin{bmatrix} 2sC + 2G & 0 & -2\sigma sC - G \\ 0 & 2sC + 2G & -sC - 2\sigma G \\ -G & -sC & sC + G \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_o \end{bmatrix} = \begin{bmatrix} GV_i \\ sCV_i \\ 0 \end{bmatrix}$$

This can be solved to get V_o and finally the transfer function $G(s)$ as

$$\begin{bmatrix} V_1 \\ V_2 \\ V_o \end{bmatrix} = \frac{1}{\Delta} \begin{bmatrix} C^2s^2 + 2(2 - \sigma)CGs + 2G^2 & CGs + 2\sigma C^2s^2 & \times \\ CGs + 2\sigma G^2 & 2C^2s^2 + 2(2 - \sigma)CGs + G^2 & \times \\ 2CGs + 2G^2 & 2C^2s^2 + 2CGs & \times \end{bmatrix} \begin{bmatrix} GV_i \\ sCV_i \\ 0 \end{bmatrix}$$

with $\Delta = 2(sC + G)[C^2s^2 + 2(2 - \sigma)CGs + 2G^2] - (2\sigma sC + G)2G(sC + G)$

$$= 2(sC + G)[C^2s^2 + 4(1 - \sigma)CGs + G^2]$$

$$G(s) = \frac{V_o(s)}{V_i(s)} = \frac{C^2s^2 + G^2}{C^2s^2 + 4(1 - \sigma)CGs + G^2} = \frac{s^2 + 1/(R^2C^2)}{s^2 + 4(1 - \sigma)s/(RC) + 1/(R^2C^2)} \quad (8.44)$$

$$= \frac{s^2 + \omega_0^2}{s^2 + \omega_b s + \omega_0^2} \quad \text{with} \quad \omega_b = \frac{4(1 - \sigma)}{RC} \quad \text{and} \quad \omega_0 = \frac{1}{RC}$$

This transfer function, having both a second-degree term and a constant term in the numerator like Equations (8.29a) and (8.34a), indicates that the circuit will function as a bandstop filter (BSF). It is noteworthy that this BSF has an adjustable quality factor

$$Q \stackrel{(8.33)}{=} \frac{\omega_r}{\omega_b} = \frac{\omega_0}{\omega_b} \stackrel{(8.44)}{=} \frac{1}{4(1 - \sigma)} \quad (8.45)$$

which can be changed by moving the sliding contact, at node 6, of the variable resistor (Reference [N-1]).

Before going on to the next example, there are a couple of things to note, which are summarized in the following remark.

[Remark 8.6] Frequency Response Scaling on Magnitude and Frequency

1. *Magnitude Scaling.* If the frequency response of a circuit is a kind of AC impedance or admittance as a phasor voltage-to-current ratio in V/A or a phasor current-to-voltage ratio in A/V, its magnitude curve can be scaled along the vertical (magnitude) axis with the scale factor k_m by multiplying all resistances/inductances by k_m and all capacitances by $1/k_m$. For example, the frequency response of a series RLC circuit can be scaled by changing its impedance as

$$Z(j\omega) = R + j\omega L + \frac{1}{j\omega C} \rightarrow k_m R + j\omega k_m L + \frac{1}{j\omega C/k_m} = k_m Z(j\omega) \quad (8.46)$$

However, if the frequency response is dimensionless as a phasor voltage-to-voltage ratio in V/V or a phasor current-to-current ratio in A/A, its magnitude curve cannot be scaled along the vertical (magnitude) axis.

2. *Frequency Scaling.* The magnitude/phase curve of frequency response can be scaled along the horizontal (frequency) axis with the scale factor k_f by multiplying all inductances and capacitances by $1/k_f$ and leaving all resistances as they are. This frequency scaling is based on the fact that the reactances of an inductor and a capacitor, ωL and $1/(\omega C)$, are not changed when the frequency variable ω is multiplied by k_f and L and C are divided by k_f .

(Example 8.6) Tuning of a BPF with MATLAB and Plot of the Frequency Response with PSpice

Consider the OP Amp circuit of Figure 8.11(a) with $R_2 = 100\ \Omega$ and $C_3 = C_4 = C$ [F].

(a) Determine the values of R_1 , R_5 , and $C_3 = C_4 = C$ to make the circuit have the transfer function

$$G(s) \stackrel{(8.42)}{=} \frac{-(G_1/C_3)s}{s^2 + [G_5(C_3 + C_4)/C_3C_4]s + (G_1 + G_2)G_5/(C_3C_4)} = \frac{-100s}{s^2 + 100s + 100^2} \quad (\text{E8.6.1})$$

so that the bandwidth, the resonant frequency, the lower/upper 3 dB frequencies of the BPF are

$$\omega_b = 100 \text{ rad/s}, \quad \omega_p = \omega_r = 100 \text{ rad/s} \ (f_r = 15.92 \text{ Hz}) \quad (\text{E8.6.2})$$

$$\omega_l, \omega_u \stackrel{(8.15)}{=} \frac{\mp \omega_b + \sqrt{\omega_b^2 + 4\omega_p^2}}{2} = \mp 50 + \sqrt{50^2 + 100^2} = 61.80, 161.80 \text{ rad/s} \quad (\text{E8.6.3})$$

$$f_l = \frac{\omega_l}{2\pi} = \frac{61.80}{2\pi} = 9.84 \text{ Hz}, \quad f_u = \frac{\omega_u}{2\pi} = \frac{161.80}{2\pi} = 25.75 \text{ Hz} \quad (\text{E8.6.4})$$

A set of nonlinear equations is written in three unknowns G_1 , G_5 , and $C_3 = C_4 = C$:

$$G_5\left(\frac{1}{C_3} + \frac{1}{C_4}\right) - 100 = 0, \quad \frac{(G_1 + G_2)G_5}{C_3C_4} - 10000 = 0, \quad \frac{G_1}{C_3} - 100 = 0 \quad (\text{E8.6.5})$$

After saving the MATLAB function `nle_cir08e06()` that describes this set of equations, the following MATLAB program `cir08e06.m` can be run to obtain

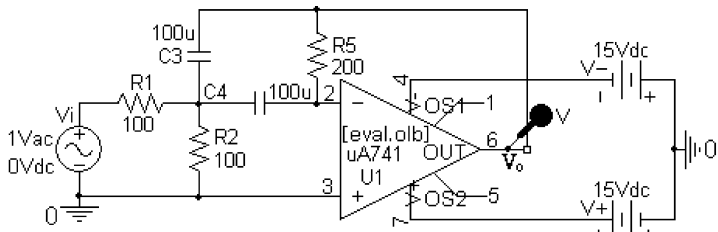
$$G_1 = 0.01 \text{ S}, \quad G_5 = 0.005 \text{ S}, \quad \text{and} \quad C_3 = C_4 = C = 10^{-4} \text{ F} \quad (\text{E8.6.6})$$

```
%cir08e06.m
clear
syms w wb wp
% The squared magnitude of BPF frequency response
Gw2= (wb*w)^2/((wp^2-w^2)^2 + (wb*w)^2);
solve(diff(Gw2)) % Derivation of Eq. (8.14) for resonant frequency
Gmax=1; nonlinear_eq= Gw2-Gmax/2;
solve(nonlinear_eq) % Derivation of Eq. (8.15) for upper/lower 3 dB freq
wb=100; wp=100; % bandwidth and resonant frequency
wlu= ((-wb+wb)+sqrt(wb^2+4*wp^2))/2; flu= wlu/2/pi % Eq. (8.15)
% Indispensable part starts from the next statement.
R2=100; G2=1/R2; % Pre-determined value of R2
G10=0.01; G50=0.01; C0=0.01; x0=[G10 G50 C0]; % Initial guess on G1,G5,C
x= fsolve('nle_cir08e06',x0,[],G2) % Solution of nonlinear eq in x
G1=x(1), G5=x(2), C3=x(3), C4=x(3) % Interpret the solution x=[G1 G5 C]
```

```
function fx=nle_cir08e06(x,G2)
G1=x(1); G5=x(2); C3=x(3); C4=x(3);
fx= [G5*(1/C3+1/C4)-100; (G1+G2)*G5/C3/C4-10000; G1/C3-100]; % Eq. (8.6.5)
```

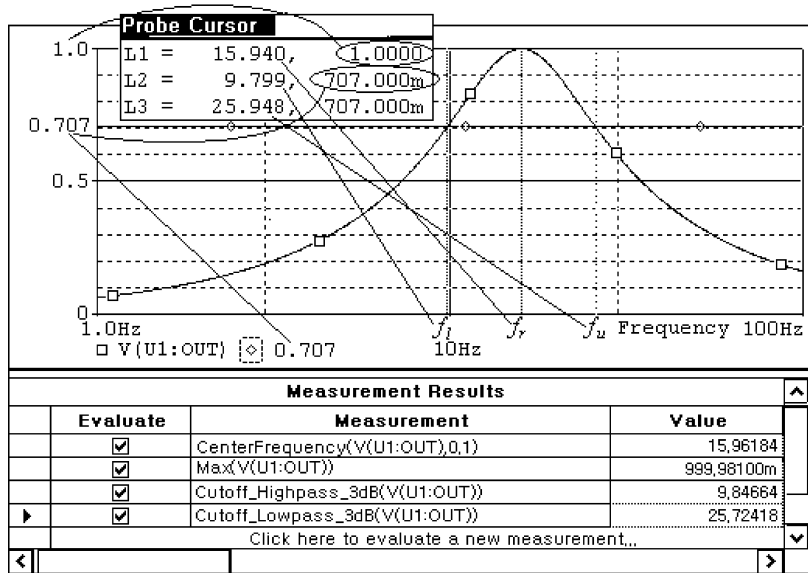
(b) The PSpice simulation is performed to obtain the frequency response of the circuit tuned in (a) and to check if the resonant frequency and the lower/upper 3 dB frequencies are as expected in (a). For this job, the following steps are taken:

- Draw the schematic as depicted in Figure 8.13(a) where a VAC voltage source is placed together with a uA741 OP Amp, three resistors $R_1 = 100\Omega$, $R_2 = 100\Omega$, and $R_3 = 200\Omega$, and two capacitors $C_3 = 10^{-4}\text{ F}$ and $C_4 = 10^{-4}\text{ F}$.
- In the Simulation Settings dialog box, set the Analysis type to ‘AC Sweep’ and the AC Sweep type as Decade (with the frequency plotted in the horizontal log scale)
Start Frequency: 1, End Frequency: 100, Points/Decade: 200
- Put the Voltage Marker at the output terminal of the OP Amp to measure $V_o(j\omega)$.
- Click the Run button on the toolbar to make the PSpice A/D (Probe) window appear on the screen, as depicted in Figure 8.13(b).
- To get the resonant frequency and the upper/lower 3 dB frequencies from the frequency response on $V_o(j\omega)$, do the following:
- Click the Toggle Cursor button on the toolbar of the Probe window.



- You can use the VdB marker available from the menu PSpice/Markers/Advanced to obtain the frequency response magnitude curve in dB.

(a) The PSpice schematic for the circuit in Fig. 8.11(a)



(b) The frequency response obtained from PSpice simulation

Figure 8.13 The PSpice schematic and the simulation result for Example 8.6

- Click the Cursor Peak button to read the peak frequency 15.94 Hz and the peak magnitude $G_{\max} = 1$ from the Probe Cursor box.
- Noting that $G_{\max}/\sqrt{2} = 1/\sqrt{2} = 0.707$, click the Add Trace button to open the Add Traces dialog box on which you type '0.707' (without the quotation marks) into the Trace Expression field at the bottom.
- Use the left mouse button or the arrow key to move the cursor to the two intersection points of the frequency (AC sweep) response curve with the horizontal line as high as 0.707 and read the two 3 dB frequencies, 9.799 Hz and 25.948 Hz, from the Probe Cursor box.
- As an alternative, click the Evaluate Measurement button on the toolbar of the Probe window and type

'Cutoff_Highpass_3 dB(V(U1:OUT))', 'Cutoff_Lowpass_3 dB(V(U1:OUT))',
and 'CenterFrequency(V(U1:OUT),0.1)'

into the Trace Expression field at the bottom of the Evaluate Measurement dialog box.

8.6 Analog Filter Design

Figures 8.14(a) to (d) show typical lowpass/bandpass/bandstop/highpass filter specifications on their log-magnitude, $20 \log_{10} |G(j\omega)|$ [dB], of the frequency response. The filter specification can be described as follows:

$$20 \log_{10} |G(j\omega_p)| \geq -R_p [\text{dB}] \quad \text{for the passband} \quad (8.47a)$$

$$20 \log_{10} |G(j\omega_s)| \leq -A_s [\text{dB}] \quad \text{for the stopband} \quad (8.47b)$$

where ω_p , ω_s , R_p , and A_s are referred to as the *passband edge frequency*, the *stopband edge frequency*, the *passband ripple*, and the *stopband attenuation*, respectively. The most commonly used analog filter design techniques are the Butterworth, Chebyshev I, II, and elliptic ones (Reference [K-3], Chapter 8). MATLAB has the built-in routines `butt()`, `cheby1()`, `cheby2()`, and `ellip()` for designing the four types of analog/digital filter. As summarized below, `butt()` needs the

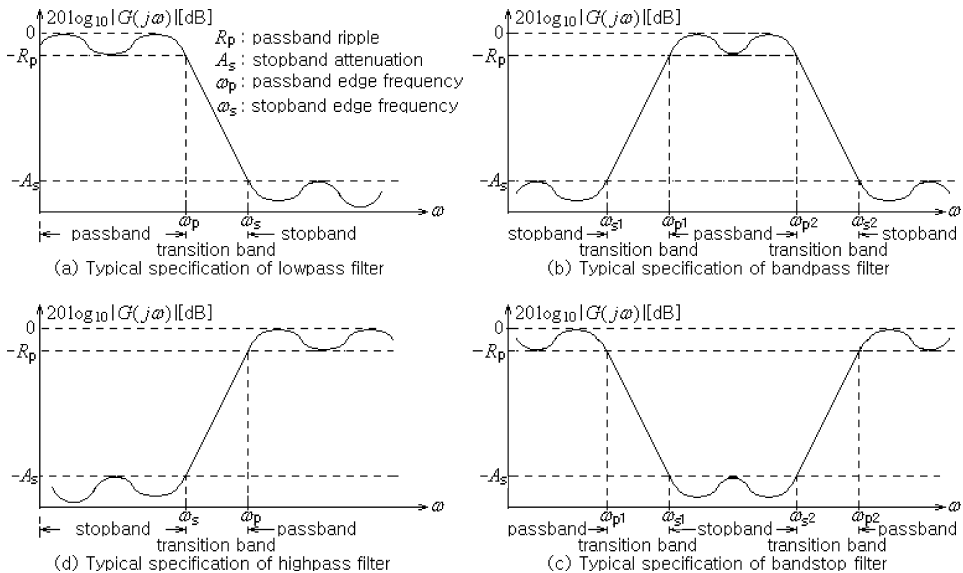


Figure 8.14 Specification on the log-magnitude of the frequency response of an analog filter

3 dB cutoff frequency while `cheby1()` and `ellip()` have the critical passband edge frequency and `cheby2()` the critical stopband edge frequency as one of their input arguments. The parametric frequencies together with the filter order can be predetermined using `buttord()`, `cheblord()`, `cheb2ord()`, and `ellipord()`. The frequency input argument should be given in a two-dimensional vector for designing BPF or BSF. Also for HPF/BSF, the string 'high' / 'stop' should be given as an optional input argument together with 's' for the analog filter design.

```
function [N,wc] = buttord(wp,ws,Rp,As,opt)
% For opt='s', this routine selects the lowest order N and cutoff frequency wc of analog
% Butterworth filter.
% that has the passband ripple(=Rp[ dB] and stopband attenuation)=As[dB]
% for the passband edge frequency wp and stopband edge frequency ws.
% Note that for the BPF/BSF, the passband edge frequency wp and stopband edge frequency
% ws should be
% given as two-dimensional vectors like [ wp1 wp2] and [ ws1 ws2].
```

```
function [B,A] =butter (N,wc,opt)
% designs a digital/analog Butterworth filter, returning the numerator/denominator of
% transfer function.
% [B,A] =butter (N,wc, 's') for the analog LPF of order N with the cutoff frequency wc
% [rad/s]
% butter (N,[wc1 wc2] , 's') for the analog BPF of order 2N with the passband
% wc1<w<wc2[ rad/s]
% butter (N,[ wc1 wc2] , 'stop' , 's') for the analog BSF of order 2N with the stopband
% wc1<w<wc2[ rad/s]
% butter (N,wc, 'high' , 's') for the analog HPF of order N with cutoff frequency
% wc[ rad/s]
% Note that N and wc can be obtained from [N,wc] =buttord (wp,ws,Rp,As,opt).
```

```
function [B,A] =cheby1 (N,Rp,wpc,opt)
% designs a digital/analog Chebyshev type I filter with the passband ripple
% Rp[dB]
% and the critical passband edge frequency wpc (Use Rp=0.5 as a starting point, if not
% sure) .
% Note that N and wpc can be obtained from [N,wpc] =chebylord (wp,ws,Rp,As,opt).
```

```
function [ B,A] =cheby2 (N,As,wsc,opt)
% designs a digital/analog Chebyshev type II filter with the stopband
% attenuation As[dB] down
% and the critical stopband edge frequency wsc (Use As=20 as a starting point,
% if not sure) .
% Note that N and wsc can be obtained from [N,wsc] =cheby2ord
% (wp,ws,Rp,As,opt) .
```

```
function [ B,A] =ellip (N,Rp,As,wpc,opt)
% designs a digital/analog Elliptic filter with the passband ripple Rp, the stopband
% attenuation As,
% and the critical passband edge frequency wpc (Use Rp=0.5[dB]
% As=20[dB] , if unsure).
% Note that N and wpc can be obtained from ellipord(wp,ws,Rp,As,opt).
```

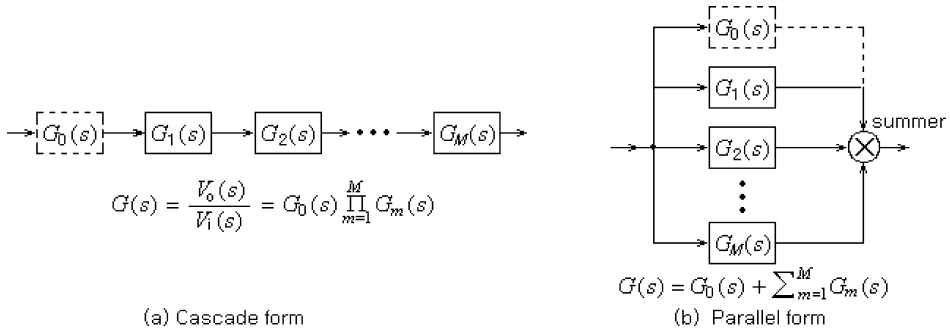


Figure 8.15 Two realizations of an analog filter (transfer function)

The designed filter transfer functions are often factored into the sum or product of second-order sections called *biquads* (possibly with an additional first-order section in the case of an odd filter order) as

$$G(s) = KG_0(s) \prod_{m=1}^M G_m(s) = K \frac{b_{00}s + b_{01}}{s + a_{01}} \prod_{m=1}^M \frac{b_{m0}s^2 + b_{m1}s + b_{m2}}{s^2 + a_{m1}s + a_{m2}} \quad \text{with } M = \text{floor}\left(\frac{N}{2}\right) \quad (8.48a)$$

$$G(s) = G_0(s) + \sum_{m=1}^M G_m(s) = \frac{b_{00}s + b_{01}}{s + a_{01}} + \sum_{m=1}^M \frac{b_{m0}s^2 + b_{m1}s + b_{m2}}{s^2 + a_{m1}s + a_{m2}} \quad \text{with } M = \text{floor}\left(\frac{N}{2}\right) \quad (8.48b)$$

and then realized in *cascade* or *parallel* form, respectively, as depicted in Figure 8.15.

Rather than reviewing the design procedures, let us use the MATLAB routines to design a Butterworth lowpass filter, a Chebyshev I bandpass filter, a Chebyshev II bandstop filter, and an elliptic highpass filter in the following example.

(Example 8.7) Filter Design Using the MATLAB Routines

Find the transfer functions of analog filters meeting the specifications given below.

- (a) The transfer function of a *Butterworth* lowpass filter will be determined with

$$\omega_p = 2\pi \times 6000 \text{ rad/s}, \quad \omega_s = 2\pi \times 15000 \text{ rad/s}, \quad R_p = 2 \text{ dB}, \quad A_s = 25 \text{ dB} \quad (\text{E8.7.1})$$

First, the MATLAB routine `buttord()` is used to find the filter order N and the cutoff frequency ω_c at which $20 \log_{10} |G(j\omega_c)| = -3 \text{ dB}$ by typing the following statements into the MATLAB command window:

```
>> wp=2*pi*6000; ws=2*pi*15000; Rp=2; As=25;
>> format short e, [N,wc]=buttord(wp,ws,Rp,As,'s')
N = 4, wc = 4.5914e+004
```

These parameter values N and wc are put into the Butterworth filter design routine `butter()` as its first and second input arguments:

```
>> [Bb,Ab]=butter(N,wc,'s')
Bb =
0 0 0 0 4.4440e+018
Ab = 1.0000e+000 1.1998e+005 7.1974e+009 2.5292e+014 4.4440e+018
```

This means that the transfer function of the designed Butterworth filter of order $N=4$ is

$$G(s) = \frac{4.444 \times 10^{18}}{s^4 + 1.1998 \times 10^5 s^3 + 7.1974 \times 10^9 s^2 + 2.5292 \times 10^{14} s + 4.444 \times 10^{18}} \quad (\text{E8.7.2})$$

The cascade and parallel realizations of this transfer function can be found by typing the following statements into the MATLAB command window:

```
>>[SOS,K]=tf2sos(Bb,Ab); % cascade realization
>>Ns=size(SOS,1); % the number of sections
>>Gm=K^(1/Ns) % the (distributed) gain of each SOS connected in cascade
>>BBc=SOS(:,1:3), AAc=SOS(:,4:6) % the numerator/denominator matrix
Gm=2.1081e+009
BBc=0    0    1    AAc=1.0000e+000    3.5141e+004    2.1081e+009
      0    0    1          1.0000e+000    8.4838e+004    2.1081e+009
>>[BBp,AAp]=tf2par_s(Bb,Ab) % parallel realization
BBp=0    4.2419e+004    3.5987e+009
      0   -4.2419e+004   -1.4906e+009
AAp=1.0000e+000    8.4838e+004    2.1081e+009
      1.0000e+000    3.5141e+004    2.1081e+009
```

This means that the designed transfer function can be realized in cascade and parallel form as

$$G(s) = \frac{2.108 \times 10^9}{s^2 + 3.514 \times 10^4 s + 2.108 \times 10^9} \times \frac{2.108 \times 10^9}{s^2 + 8.484 \times 10^4 s + 2.108 \times 10^9} \quad (\text{E8.7.3a})$$

$$G(s) = \frac{4.242 \times 10^4 s + 3.599 \times 10^9}{s^2 + 8.484 \times 10^4 s + 2.108 \times 10^9} - \frac{4.242 \times 10^4 s + 1.491 \times 10^9}{s^2 + 3.514 \times 10^4 s + 2.108 \times 10^9} \quad (\text{E8.7.3b})$$

- (b) The transfer function of a Chebyshev I bandpass filter will be determined with

$$\omega_{s1} = 2\pi \times 6000, \quad \omega_{p1} = 2\pi \times 10\,000, \quad \omega_{p2} = 2\pi \times 12\,000, \quad \omega_{s2} = 2\pi \times 15\,000 \text{ rad/s}, \\ R_p = 2 \text{ dB}, \quad \text{and} \quad A_s = 25 \text{ dB} \quad (\text{E8.7.4})$$

First, the MATLAB routine `cheblord( )` is used to find the filter order N and the critical passband edge frequencies ω_{pc1} and ω_{pc2} at which the passband ripple condition is closely met, i.e. $20 \log_{10} |G(j\omega_{pc})| = -R_p [\text{dB}]$ by typing the following statements:

```
>>ws1=2*pi*e3; wp1=2*pi*1e4; wp2=2*pi*12e3; ws2=2*pi*15e3; Rp=2; As=25;
>>[N,wpc]=cheblord([wp1 wp2],[ws1 ws2],Rp,As,'s')
N=2, wpc=6.2832e+004    7.5398e+004
```

The (half) filter order N , the passband ripple R_p , and the critical passband edge frequency vector $wpc = [\omega_{pc1} \ \omega_{pc2}]$ are put into the Chebyshev I filter design routine `cheby1( )` as

```
>>[Bc1,Ac1]=cheby1(N,Rp,wpc,'s')
Bc1=0    0    0    1.0324e+008    0    0
Ac1=1.0000e+000    1.0101e+004    9.6048e+009    4.7853e+013    2.2443e+019
```

This means that the transfer function of the designed Chebyshev I filter of order $2N=4$ is

$$G(s) = \frac{1.0324 \times 10^8 s^2}{s^4 + 10101 s^3 + 9.6048 \times 10^9 s^2 + 4.7853 \times 10^{13} s + 2.2443 \times 10^{19}} \quad (\text{E8.7.5})$$

The cascade and parallel realizations of this transfer function can be found by typing the following statements into the MATLAB command window:

```
>>[SOS,K]=tf2sos(Bc1,Ac1); % cascade realization
>>Ns=size(SOS,1); Gm=K^(1/Ns), BBc=SOS(:,1:3), AAC=SOS(:,4:6)
Gm=1.0161e+004
BBc=0 0 1 AAC=1.0000e+000 5.4247e+003 5.4956e+009
1 0 0 1.0000e+000 4.6763e+003 4.0838e+009
>>[BBp,AAp]=tf2par_s(Bc1,Ac1) % parallel realization
BBp=0 1.8390e+002 4.0242e+008
0 -1.8390e+002 -2.9904e+008
AAp=1.0000e+000 5.4247e+003 5.4956e+009
1.0000e+000 4.6763e+003 4.0838e+009
```

This means that the designed transfer function can be realized in cascade and parallel form as

$$G(s) = \frac{1.0161 \times 10^4 s}{s^2 + 5.425 \times 10^3 s + 5.496 \times 10^9} \times \frac{1.0161 \times 10^4 s}{s^2 + 4.676 \times 10^3 s + 4.084 \times 10^9} \quad (\text{E8.7.6a})$$

$$G(s) = \frac{1.839 \times 10^2 s + 4.024 \times 10^8}{s^2 + 5.425 \times 10^3 s + 5.496 \times 10^9} - \frac{1.839 \times 10^2 s + 2.990 \times 10^8}{s^2 + 4.676 \times 10^3 s + 4.084 \times 10^9} \quad (\text{E8.7.6b})$$

- (c) The transfer function of a Chebyshev II bandstop filter will be determined with

$$\omega_{p1} = 2\pi \times 6000, \quad \omega_{s1} = 2\pi \times 10000, \quad \omega_{s2} = 2\pi \times 12000, \quad \omega_{p2} = 2\pi \times 15000 \text{ rad/s} \\ R_p = 2 \text{ dB}, \quad \text{and} \quad A_s = 25 \text{ dB} \quad (\text{E8.7.7})$$

First, the MATLAB routine `cheb2ord( )` is used to find the filter order N and the critical stopband edge frequencies ω_{sc1} and ω_{sc2} at which the stopband attenuation condition is closely met, i.e. $20 \log_{10} |G(j\omega_{sc})| = -A_s [\text{dB}]$ by typing the following statements:

```
>>wp1=2*pi*6000; ws1=2*pi*10000; ws2=2*pi*12000; wp2=2*pi*15000;
>>Rp=2; As=25;
>>[N,wsc]=cheb2ord([ wp1 wp2 ],[ ws1 ws2 ],Rp,As,'s')
N=2, wsc=6.2798e+004 7.5438e+004
```

The (half) filter order N , the stopband attenuation A_s , and the critical stopband edge frequency vector $wsc = [\omega_{sc1} \omega_{sc2}]$ is put into the Chebyshev II filter design routine `cheby2( )` as

```
>>[Bc2,Ac2]=cheby2(N,As,wsc,'stop','s')
Bc2=1.0000e+000 1.0979e-010 9.5547e+009 4.9629e-001 2.2443e+019
Ac2=1.0000e+000 5.1782e+004 1.0895e+010 2.4531e+014 2.2443e+019
```

This means that the transfer function of the designed Chebyshev II filter of order $2N=4$ is

$$G(s) = \frac{s^4 + 9.5547 \times 10^9 s^2 + 4.9629 \times 10^{-1} s + 2.2443 \times 10^{19}}{s^4 + 51782 s^3 + 1.0895 \times 10^{10} s^2 + 2.4531 \times 10^{14} s + 2.2443 \times 10^{19}} \quad (\text{E8.7.8})$$

The cascade and parallel realizations of this transfer function are found by typing the following statements into the MATLAB command window:

```
>>[SOS,K]=tf2sos(Bc2,Ac2); % cascade realization
>>Ns=size(SOS,1); Gm=K^(1/Ns), BBc=SOS(:,1:3), AAC=SOS(:,4:6)
Gm=1
```



```

BBc=1.0000e+000 7.7795e-011 5.3938e+009
      1.0000e+000 2.9104e-011 4.1609e+009
AAc=1.0000e+000 3.1028e+004 7.0828e+009
      1.0000e+000 2.0754e+004 3.1687e+009
>>[BBp,AAp]=tf2par_s(Bc2,Ac2) % parallel realization
BBp=5.0000e-001 -1.5688e+004 3.4426e+009
      5.0000e-001 -1.0204e+004 1.6285e+009
AAp=1.0000e+000 3.1028e+004 7.0828e+009
      1.0000e+000 2.0754e+004 3.1687e+009

```

This means that the designed transfer function can be realized in cascade and parallel form as

$$G(s) = \frac{s^2 + 5.394 \times 10^9}{s^2 + 3.103 \times 10^4 s + 7.083 \times 10^9} \times \frac{s^2 + 4.161 \times 10^9}{s^2 + 2.075 \times 10^4 s + 3.169 \times 10^9} \quad (\text{E8.7.9a})$$

$$G(s) = \frac{0.5s^2 - 1.569 \times 10^4 s + 3.443 \times 10^9}{s^2 + 3.103 \times 10^4 s + 7.083 \times 10^9} + \frac{0.5s^2 - 1.020 \times 10^4 s + 1.6285 \times 10^9}{s^2 + 2.075 \times 10^4 s + 3.169 \times 10^9} \quad (\text{E8.7.9b})$$

(d) The transfer function of an *elliptic* highpass filter will be determined with

$$\omega_s = 2\pi \times 6000 \text{ rad/s}, \quad \omega_p = 2\pi \times 15000 \text{ rad/s}, \quad R_p = 2 \text{ dB}, \quad \text{and} \quad A_s = 25 \text{ dB} \quad (\text{E8.7.10})$$

First, the MATLAB routine `ellipord( )` is used to find the filter order N and the critical passband edge frequency ω_{pc} at which $20 \log_{10} |G(j\omega_{pc})| = -R_p [\text{dB}]$ by typing the following statements into the MATLAB command window:

```

>>ws=2*pi*6000; wp=2*pi*15000; Rp=2; As=25;
>>format short e, [N,wc]=ellipord(wp,ws,Rp,As,'s')
N=3, wc=9.4248e+004

```

The parameter values N , R_p , A_s , and w_c are put into the elliptic filter design routine `ellip( )` as

```

>>[Be,Ae]=ellip(N,Rp,As,wc,'high','s')
Be=1.0000e+000 8.9574e-009 3.9429e+009 -5.6429e+002
Ae=1.0000e+000 2.3303e+005 1.4972e+010 1.9511e+015

```

This means that the transfer function of the designed elliptic filter of order $N = 3$ is

$$G(s) = \frac{s^3 + 3.9429 \times 10^9 s - 5.6429 \times 10^2}{s^3 + 2.3303 \times 10^5 s^2 + 1.4972 \times 10^{10} s + 1.9511 \times 10^{15}} \quad (\text{E8.7.11})$$

The cascade and parallel realizations of this transfer function can be found by typing the following statements into the MATLAB command window:

```

>>[SOS,K]=tf2sos(Be,Ae); % cascade realization
>>Ns=size(SOS,1); Gm=K^(1/Ns), BBc=SOS(:,1:3), AAc=SOS(:,4:6)
Gm=1.0000e+000
BBc=1.0000e+000 -1.4311e-007 0
      1.0000e+000 1.5207e-007 3.9429e+009
AAc=1.0000e+000 2.0630e+005 0
      1.0000e+000 2.6731e+004 9.4575e+009
>>[BBp,AAp]=tf2par_s(Be,Ae) % parallel realization
BBp=5.0000e-001 -1.3365e+004 4.7287e+009

```

```

0      5.0000e-001  -1.0315e+005
AAp = 1.0000e+000  2.6731e+004  9.4575e+009
0      1.0000e+000  2.0630e+005

```

This means that the designed transfer function can be realized in cascade and parallel form as

$$G(s) = \frac{s}{s + 2.063 \times 10^5} \times \frac{s^2 + 3.943 \times 10^9}{s^2 + 2.673 \times 10^4 s + 9.458 \times 10^9} \quad (\text{E8.7.12a})$$

$$G(s) = \frac{0.5s^2 - 1.337 \times 10^4 s + 4.729 \times 10^9}{s^2 + 2.673 \times 10^4 s + 9.458 \times 10^9} + \frac{0.5s - 1.032 \times 10^5}{s + 2.063 \times 10^5} \quad (\text{E8.7.12b})$$

- (e) All of the above filter design works are put into the M-file named `cir08e07.m`, which plots the frequency responses of the designed filters so that a check can be made to find if the design specifications are satisfied. Figure 8.16, obtained by running the program `cir08e07.m`, shows the following points:

- Figure 8.16(a) shows that the cutoff frequency ω_c given as an input argument of `butter()`, is the frequency at which $20 \log_{10} |G(j\omega_c)| = -3 \text{ dB}$. Note that the frequency response magnitude of the Butterworth filter is monotonic, i.e. has no ripple.

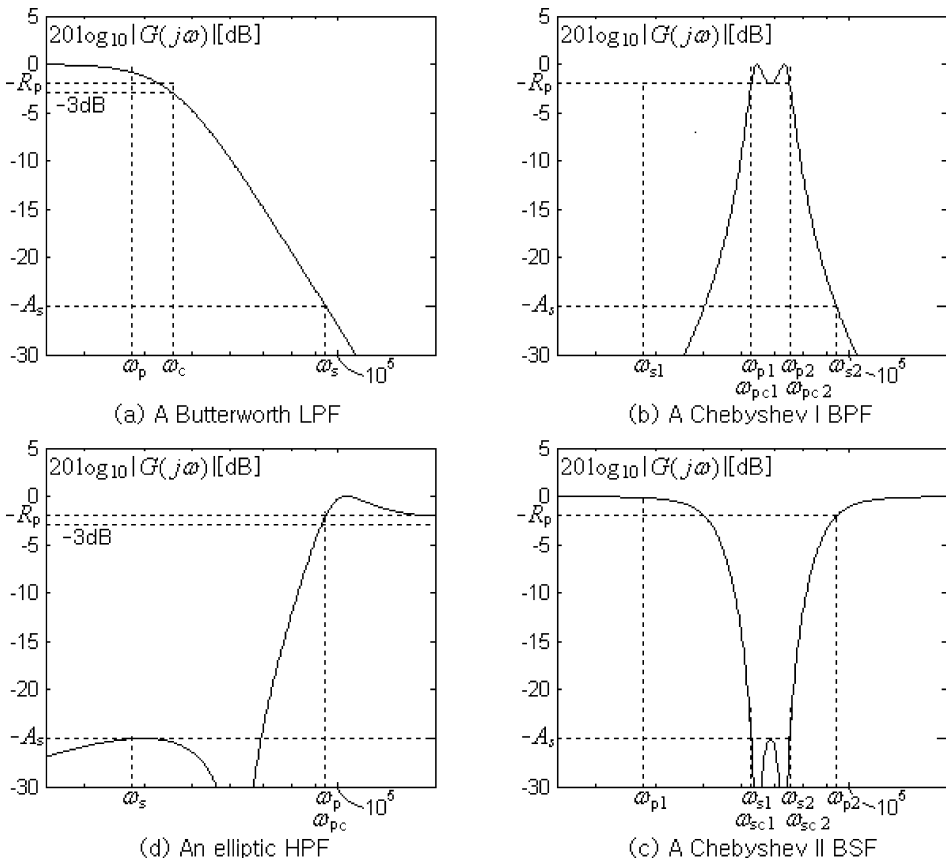


Figure 8.16 Frequency responses of the filters designed in Example 8.7

- Figure 8.16(b) shows that the critical passband edge frequencies ω_{pc1} and ω_{pc2} , given as an input argument $wpc=[wpc1\ wpc2]$ of `cheby1 ( )`, are the frequencies at which the passband ripple condition is closely met, i.e. $20\log_{10}|G(j\omega_{pc})| = -R_p[\text{dB}]$. Note that the frequency response magnitude of the Chebyshev I filter closely satisfying the passband ripple condition has a ripple in the passband, which is traded off for a narrower transition band than the Butterworth filter (with the same filter order).
- Figure 8.16(c) shows that the critical stopband edge frequencies ω_{sc1} and ω_{sc2} , given as an input argument $wsc=[wsc1\ wsc2]$ of `cheby2 ( )`, are the frequencies at which the stopband attenuation condition is closely met, i.e. $20\log_{10}|G(j\omega_{sc})| = -A_s[\text{dB}]$. Note that the frequency response magnitude of the Chebyshev II filter closely satisfying the stopband ripple condition has a ripple in the stopband.
- Figure 8.16(d) shows that the critical passband edge frequency ω_{pc} , given as an input argument wpc of `ellip ( )`, is the frequency at which the passband ripple condition is closely met, i.e. $20\log_{10}|G(j\omega_{pc})| = -R_p[\text{dB}]$. Note that the frequency response magnitude of the elliptic filter has ripples in both the passband and the stopband, yielding a relatively narrow transition band with the smallest filter order $N=3$ among the four filters.

```
%cir08e07.m for the filter design and frequency response plot
clear, clf, format short e
disp(' (a) Butterworth LPF ')
wp=2*pi*6000; ws=2*pi*15000; Rp=2; As=25;
[ Nb,wcb] = buttord(wp,ws,Rp,As,'s') % Order of analog BW LPF
[ Bb,Ab] = butter(Nb,wcb,'s') % num/den of analog BW LPF transfer ftn
[ SOS,K] = tf2sos(Bb,Ab); % cascade realization
Ns=size(SOS,1); Gm=K^(1/Ns), BBc=SOS(:,1:3), AAC=SOS(:,4:6)
[ BBp,AAp] = tf2par_s(Bb,Ab) % parallel realization
ww = logspace(4,6,1000); % log frequency vector from 1e4 to 1e6[ rad/s]
subplot(221), semilogx(ww,20*log10(abs(freqs(Bb,Ab,ww))))
title('Butterworth LPF')
disp(' (b) Chebyshev I BPF ')
ws1=2*pi*6e3; wp1=2*pi*1e4; wp2=2*pi*12e3; ws2=2*pi*15e3; Rp=2; As=25;
[ Nc1,wpc] = cheblord([ wp1 wp2 ],[ ws1 ws2 ],Rp,As,'s')
[ Bc1,Ac1] = cheby1(Nc1,Rp,wpc,'s')
[ SOS,K] = tf2sos(Bc1,Ac1); % cascade realization
Ns=size(SOS,1); Gm=K^(1/Ns), BBc=SOS(:,1:3), AAC=SOS(:,4:6)
[ BBp,AAp] = tf2par_s(Bc1,Ac1) % parallel realization
subplot(222), semilogx(ww,20*log10(abs(freqs(Bc1,Ac1,ww))))
title('Chebyshev I BPF')
disp(' (c) Chebyshev II BSF ')
wp1=2*pi*6e3; ws1=2*pi*1e4; ws2=2*pi*12e3; wp2=2*pi*15e3; Rp=2; As=25;
[ Nc2,wsc] = cheb2ord([ wp1 wp2 ],[ ws1 ws2 ],Rp,As,'s')
[ Bc2,Ac2] = cheby2(Nc2,As,wsc,'stop','s')
[ SOS,K] = tf2sos(Bc2,Ac2); % cascade realization
Ns=size(SOS,1); Gm=K^(1/Ns), BBc=SOS(:,1:3), AAC=SOS(:,4:6)
[ BBp,AAp] = tf2par_s(Bc2,Ac2) % parallel realization
subplot(224), semilogx(ww,20*log10(abs(freqs(Bc2,Ac2,ww))))
title('Chebyshev II BSF')
disp(' (d) Elliptic HPF ')
ws=2*pi*6000; wp=2*pi*15000; Rp=2; As=25;
[ Ne,wpc] = ellipord(wp,ws,Rp,As,'s')
[ Be,Ae] = ellip(Ne,Rp,As,wpc,'high','s')
```

```
[ SOS,K] = tf2sos(Be,Ae); % cascade realization
Ns=size(SOS,1); Gm=K^(1/Ns), BBc=SOS(:,1:3), AAC=SOS(:,4:6)
[ BBp,AAp] = tf2par_s(Be,Ae) % parallel realization
subplot(223), semilogx(ww,20*log10(abs(freqs(Be,Ae,ww))))
```

```
function[ BB,AA,K] =tf2par_s(B,A)
% Copyleft: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only
EPS= 1e-6;
B= B/A(1); A= A/A(1);
I= find(abs(B)>EPS); K= B(I(1)); B= B(I(1):end);
p= roots(A); p= cplxpair(p); Np= length(p);
NB= length(B); N= length(A); M= floor(Np/2);
for m=1:M
    m2=m*2; AA(m,:)= [ 1 -p(m2-1) -p(m2) p(m2-1)*p(m2)] ;
end
if Np>2*M
    AA(M+1,:)= [ 0 1 -p(Np)] ; % For a single pole
end
M1= M+ (Np>2*M); b= [ zeros(1,Np-NB) B] ; KM1= K/M1;
% In case B(s) and A(s) has the same degree, we let all the coefficients
% of the 2nd-order term in the numerator of each SOS be B1=1/M1:
if NB==N, b= b(2:end); end
for m=1:M1
    polynomial= 1; m2=2*m;
    for n=1:M1
        if n~=m, polynomial= conv(polynomial,AA(n,:)); end
    end
if m<=M
    if M1>M, polynomial=polynomial(2:end); end
    if NB==N, b= b - [ polynomial(2:end)*KM1 0 0] ; end
    Ac(m2-1,:)= [ polynomial 0] ;
    Ac(m2,:)= [ 0 polynomial] ;
else
    if NB==N, b= b - [ polynomial(2:end)*KM1 0] ; end
    Ac(m2-1,:)= polynomial;
    end
end
Bc= b/Ac; Bc( find(abs(Bc)<EPS) ) = 0;
for m=1:M1
    m2= 2*m;
    if m<=M
        BB(m,:)= [ 0 Bc(m2-1:m2)] ; if NB==N, BB(m,1) = KM1; end
    else
        BB(m,:)= [ 0 0 Bc(end)] ; if NB==N, BB(m,2) = KM1; end
    end
end
end
```

This section will now be concluded with some MATLAB routines that can be used to determine the parameters of the circuits depicted in Figures 8.9, 8.10, and 8.11 so that they can realize the designed transfer functions.

```
function[ CR1,CR2,Gs]=filter_LPF_8_9a(A2,A3,K,RC1,RC2,KC)
% Design an LPF with the circuit in Figure 8.9(a)
%           KG1G2/C1C2           B3=K*A3
% G(s) = ----- = ----- (8.38)
%      s^2 + (G1+G2)/C1 + (1-K)G2/C2*s + G1G2/C1C2      s^2 + A2*s + A3
if K<1, error('We must have K=(R3+R4)/R3 >= 1!'); end
if nargin<6, KC=1; end
if KC==1 % Find C1 and C2 for given K, R1, and R2.
    R1=RC1; R2=RC2; G1=1/R1; G2=1/R2;
    a=G1+G2; b=-(K-1)*G2; c=A2; d=A3/G1/G2; tmp=c^2-4*a*b*d;
    C1=2*a/(c+sqrt(tmp)); C2=1/d/C1; CR1=C1; CR2=C2;
else % Find R1 and R2 for given K, C1, and C2.
    C1=RC1; C2=RC2;
    a=1/C1; b=1/C1 - (K-1)/C2; c=A2; d=A3*C1*C2; tmp=c^2-4*a*b*d;
    if tmp<0, error('Increase C1 and K, or decrease C2'); end
    G1=(c+sqrt(tmp))/2/a; G2=d/G1; R1=1/G1; R2=1/G2; CR1=R1; CR2=R2;
end
B3=K*A3; A2=(G1+G2)/C1 + (1-K)*G2/C2; A3=G1*G2/C1/C2;
syms s; Gs=B3/(s^2+A2*s+A3);
```

```
function[ CR1,CR2,Gs]=filter_HPF_8_9b(A2,A3,K,RC1,RC2,KC)
% Design a HPF with the circuit in Figure 8.9(b)
%           K*s^2           K*s^2
% G(s) = ----- = ----- (8.39)
%      s^2 + (G2(1/C1+1/C2)-(K-1)G1/C1)s + G1G2/C1C2      s^2 + A2*s + A3
if K<1, error('We must have K=(R3+R4)/R3 >= 1!'); end
if nargin<6, KC=1; end
if KC==1 % Find C1 and C2 for given K, R1, and R2.
    R1=RC1; R2=RC2; G1=1/R1; G2=1/R2;
    a=G2+(1-K)*G1; b=G2; c=A2; d=A3/G1/G2; tmp=c^2-4*a*b*d;
    if tmp<0, error('Try with smaller/greater values of R1/K'); end
    C1=2*a/(c+sign(a)*sqrt(tmp)); C2=1/d/C1; CR1=C1; CR2=C2;
else % Find R1 and R2 for given K, C1, and C2.
    C1=RC1; C2=RC2;
    a=(1-K)/C1; b=1/C1+1/C2; c=A2; d=A3*C1*C2; tmp=c^2-4*a*b*d;
    if tmp<0, error('Try with smaller/greater values of C2/K'); end
    if abs(a)<eps, G2=A2/b; G1=d/G2;
    else G1=(c+sign(a)*sqrt(tmp))/2/a; G2=d/G1;
    end
    R1=1/G1; R2=1/G2; CR1=R1; CR2=R2;
end
```

```
B1= K; A2= G2* (1/C1+1/C2) - (K-1)*G1/C1; A3= G1*G2/C1/C2;
syms s; Gs= B1*s^2/ (s^2+A2*s+A3);
```

```
function[ R1,C2R3,C5R4,Gs]=filter_LPF_8_10a(B3,A2,A3,R3C2,R4C5,KC)
% Design an LPF with the circuit in Figure 8.10 (a)
%      -G1G4/C2C5                      -B3
% G(s) = ----- = ----- (8.40)
%      s^2 + (G1+G3+G4)/C2*s + G3G4/C2C5      s^2 + A2*s + A3
if nargin<6, KC=1; end
if KC==1 % Find R1, C2 and C5 for given R3 and R4.
    R3=R3C2; R4= R4C5; G3= 1/R3; G4= 1/R4;
    G1=G3*B3/A3; C2=(G1+G3+G4)/A2; C5=G3*G4/C2/A3; R1=1/G1; C2R3=C2; C5R4=C5;
else % Find R1, R3 and R4 for given C2 and C5.
    C2=R3C2; C5=R4C5; a=1+B3/A3; b=1; c=A2*C2; d=A3*C2*C5; tmp= c^2-
    4*a*b*d;
    if tmp<0, error('Try with greater/smaller values of C2/C5'); end
    G3= (c + sign(a)*sqrt(tmp))/2/a; G4= d/G3;
G1= B3/A3*G3; R3= 1/G3; R4= 1/G4; R1=1/G1; C2R3= R3; C5R4= R4;
end
B3= G1*G4/C2/C5; A2= (G1+G3+G4)/C2; A3= G3*G4/C2/C5;
syms s; Gs= -B3/ (s^2+A2*s+A3);
```

```
function[ C1,C3R2,C4R5,Gs]=filter_HPF_8_10b(B1,A2,A3,R2C3,R5C4,KC)
% Design a HPF with the circuit in Figure 8.10 (b)
%      -(C1/C3)*s^2                      -B1*s^2
% G(s) = ----- = ----- (8.41)
%      s^2 + G5(C1+C3+C4)/C3/C4*s + G2G5/C3C4      s^2 + A2*s + A3
if nargin<6, KC=1; end
if KC==1 % Find C1, C3 and C4 for given R2 and R5.
    R2= R2C3; R5= R5C4; G2= 1/R2; G5= 1/R5;
    a= 1; b= 1+B1; c= A2/G5; d= A3/G2/G5; tmp= c^2-4*a*b*d;
    if tmp<0, error('Try with smaller/greater values of R2/R5'); end
    C3= 2*a/(c + sqrt(tmp)); C4= 1/d/C3; C1= B1*C3; C3R2= C3; C4R5= C4;
else % Find C1, R2 and R5 for given C3 and C4.
    C3= R2C3; C4= R5C4;
    C1= B1*C3; G5= A2/ (C1+C3+C4)*C3*C4; G2= A3*C3*C4/G5;
    R2= 1/G2; R5= 1/G5; C3R2= R2; C4R5= R5;
end
B1= C1/C3; A2= G5* (C1+C3+C4)/C3/C4; A3= G2*G5/C3/C4;
syms s; Gs= -B1*s^2/ (s^2+A2*s+A3);
% Examples of Usage
%>>B1=2; A2=100; A3=10000;
%>>R2=1e4; R5=2e5; [ C1,C3,C4,Gs]=filter_HPF_8_10b(B1,A2,A3,R2,R5,1)
%>>C3=1e-7; C4=2e-6; [ C1,R2,R5,Gs]=filter_HPF_8_10b(B1,A2,A3,C3,C4,2)
```

```
function[ C3R1,C4R2,R5,Gs]=filter_BPF_8_11a(B2,A2,A3,R1C3,R2C4,KC)
% Design a BPF with the circuit in Figure 8.11 (a)
```

```

%      - (G1/C3)*s                      -B2*s
% G(s) = ----- = ----- (8.42)
%      s^2 + G5(1/C3+1/C4)*s + (G1+G2)G5/C3C4      s^2 + A2*s + A3
if nargin<6, KC=1; end
if KC==1 % Find C3, C4, and R5 for given R1 and R2.
    R1=R1C3; R2=R2C4; G1=1/R1; G2=1/R2; C3=G1/B2;
    G5=(A2-A3*C3/(G1+G2))*C3;
if G5<0, error('Try with smaller values of R2'); end
    C4=G5*(G1+G2)/C3/A3;
    R5=1/G5; C3R1=C3; C4R2=C4;
    fprintf('C3=%10.4e, C4=%10.4e, R5=%10.4e\n', C3,C4,R5)
elseif KC==2 % Find R1, R2 and R5 for given C3 and C4.
    C3=R1C3; C4=R2C4;
    G1=B2*C3; G5=A2/(1/C3+1/C4); G2=A3*C3*C4/G5-G1;
    R5=1/G5; R1=1/G1; R2=1/G2; C3R1=R1; C4R2=R2;
fprintf('R1=%10.4e, R2=%10.4e, R5=%10.4e\n', R1,R2,R5)
else % Find R1, R5, and C3=C4=C for given R2 and C3=C4.
    G2=1/R1C3;
    nonlinear_eq= inline('[-2*x(1)-A2*x(2); x(1).*(B2*x(2)+G2)-A3*x(2).
    ^2]',...
        'x','G2','B2','A2','A3');
    G50=0.1; C0=0.1; x0=[ G50 C0]; % Initial guesses of G5 and C
    x=fsolve(nonlinear_eq,x0,optimset('TolFun',1e-8),G2,B2,A2,A3)
    %tol=1e-5; MaxIter=100;
x=newtons(nonlinear_eq,x0,tol,MaxIter,G2,B2,A2,A3)
    G5=x(1); C=x(2); C3=C; C4=C; G1=B2*C3;
    R1=1/G1; R5=1/G5; C3R1=C3; C4R2=R1;
    fprintf('C3=C4=%10.4e, R1=%10.4e, R5=%10.4e\n', C,R1,R5)
end
B1=G1/C3; A2=G5*(1/C3+1/C4); A3=(G1+G2)*G5/C3/C4;
syms s; Gs=-B2*s/(s^2+A2*s+A3);
% Examples of Usage
%>>B2=100; A2=100; A3=10000;
%>>R1=1e2; R2=1e2; [ C3,C4,R5,Gs]=filter_BPF_8_11a(B2,A2,A3,R1,R2,1)
%>>C3=1e-4; C4=1e-4; [ R1,R2,R5,Gs]=filter_BPF_8_11a(B2,A2,A3,C3,C4,2)
%>>R2=1e2; [ C3,R1,R5,Gs]=filter_BPF_8_11a(B2,A2,A3,R2,0,3)
for Example 8.6

```

```

function[ C1,C2R3,C5R4,Gs]=filter_BPF_8_11b(B2,A2,A3,R3C5,R4C5,KC)
% Design a BPF with the circuit in Figure 8.11 (b)
%      - (C1G4/(C1+C2)C5)*s                      -B2*s
% G(s) = ----- = ----- (8.43)
%      s^2 + ((G3+G4)/(C1+C2))*s + G3G4/(C1+C2)C5      s^2 + A2*s + A3
if nargin<6, KC=1; end
if KC==1 % Find C1, C2 and C5 for given R3 and R4.
    R3=R3C5; R4=R4C5; G3=1/R3; G4=1/R4;
    C1pC2=(G3+G4)/A2; C5=G3*G4/A3/C1pC2; C1=B2*C1pC2*C5/G4 %B2*G3/A3
    C2=C1pC2-C1; C2R3=C2; C5R4=C5;

```

```

    if C2<0, error('Try with greater/smaller values of R3/R4' ); end
    fprintf('C1=%10.4e, C2=%10.4e, C5=%10.4e\n', C1,C2,C5)
else % Find C1, R3 and R4 for given C5 and C1=C2.
    C5=R3C5; G4= 2*C5*B2; G3_2C= A3/G4*C5; %A3/2/B2: not adjustable
    C= G4/2/(A2-G3_2C); C1=C; C2=C;
    if C<0, error('How about increasing B2    A2 and/or decreasing A3' ); end
    G3= G3_2C*C; R3= 1/G3; R4= 1/G4; C2R3= R3; C5R4= R4;
    fprintf('C1=C2=%10.4e, R3=%10.4e, R4=%10.4e\n', C,R3,R4)
end
B3= C1*G4/(C1+C2)/C5; A2= (G3+G4)/(C1+C2); A3= G3*G4/(C1+C2)/C5;
syms s; Gs= -B2*s/(s^2+A2*s+A3);
%>>B2=50; A2=20; A3=1000; R3=1e4; R4=1e5;
%>>[ C1,C2,C5,Gs]=filter_BPF_8_11b(B2,A2,A3,R3,R4,1)
%>>B2=20; A2=50; A3=1000; C5=1e-6; C=0;
%>>[ C1,R3,R4,Gs]=filter_BPF_8_11b(B2,A2,A3,C5,C,2)

```

For example, the MATLAB routine `filter_BPF_8_11a( )` can be used to tune the parameters of the MFB (multiple feedback) circuit of Figure 8.11(a) so that the circuit realizes the BPF transfer function (E8.6.1) designed in Example 8.6:

$$G(s) \stackrel{(8.42)}{=} \frac{-(G_1/C_3)s}{s^2 + [G_5(C_3 + C_4)/C_3C_4]s + (G_1 + G_2)G_5/(C_3C_4)} \stackrel{(E8.6.1)}{=} \frac{-100s}{s^2 + 100s + 100^2} \quad (8.49)$$

To this end, only the following statements need to be typed into the MATLAB command window:

```

>>B2=100; A2=100; A3=10000; % The desired transfer function B2*s/(s^2+A2*s+A3)
>>R2=1e2; KC=3; % With the given value of R2=100 and the assumption that C3=C4
>>[C3,R1,R5,Gs]=filter_BPF_8_11a(B2,A2,A3,R2,0,KC)
    C3=C4=1.0000e-004, R1=9.9999e+001, R5=2.0000e+002
    Gs= -100*s/(s^2+100*s+5497540047884755/549755813888)
>>5497540047884755/549755813888 % To see the weird constant term
    1.0000e+004

```

For another example, the MATLAB routine `filter_LPF_8_9a( )` could be used to tune the parameters of the Sallen–Key circuit of Figure 8.9(a) so that the circuit realizes the following LPF transfer function:

$$G(s) \stackrel{(8.38)}{=} \frac{KG_1G_2/(C_1C_2)}{s^2 + [(G_1 + G_2)/C_1 + (1 - K)G_2/C_2]s + G_1G_2/(C_1C_2)} = \frac{K\omega_r^2}{s^2 + \omega_b s + \omega_r^2} \quad (8.50)$$

More specifically, suppose the values of R_1 and R_2 of the Sallen–Key circuit of Figure 8.9(a) need to be determined with the predetermined values of capacitances $C_1 = C_2 = 100$ pF so that a second-order LPF is realized with the DC gain $K = 1.5$, the corner frequency $\omega_r = 2\pi \times 10^7$ rad/s, and the quality factor $Q = 0.707$ (for $\omega_b \stackrel{(8.17),(8.18)}{=} \omega_r/Q$). To this end, only the following statements need to be typed into the MATLAB command window:

```

>>K=1.5; C1=1e-10; C2=1e-10; wr=2*pi*1e7; Q=0.707;
    % The coefficients of denominator of desired transfer ftn
>>A2=wr/Q; A3=wr^2; % G(s)=K*A3/(s^2+A2*s+A3)

```



```

>>KC=2; [ R1,R2,Gs] = filter_LPF_8_9a(A2,A3,K,C1,C2,KC)
R1=221.2010 % tuned resistance
R2=114.5126
Gs=5921762640653615/(s^2+5964037174912491/67108864*s+7895683520871487/2)

```

Note. For reference, you can visit the website <<http://www.national.com/pf/LM/LMH6628.html>> to see the application note OA-26 for Designing Active High Speed Filters.

To illustrate the filter design and realization procedure collectively, find the cascade realization of a fourth-order Butterworth LPF with the cutoff frequency $\omega_c = 10$ kHz using the Sallen–Key circuit of Figure 8.9(a). For this job, the following program is composed and run:

```

%cir08_06_2.m
N=4; fc=1e4; wc=2*pi*fc; % the order and cutoff frequency of the LPF
format short e
[B,A] = butter(N,wc,'s') % Butterworth LPF transfer function G(s)=B(s)/A(s)
f= logspace(3,5,400); % frequency vector of 400 points between 1e3~1e5 [Hz]
Gw= freqs(B,A,2*pi*f); % frequency response G(jw)
semilogx(f,20*log10(abs(Gw))) % plot |G(jw)| in dB versus frequency [Hz]
[SOS,K0] = tf2sos(B,A); % cascade realization
BBc=SOS(:,1:3); AAC=SOS(:,4:6); % numerator/denominator of each SOS
K=1; R1= 1e4; R2= 1e4; KC=1; % predetermined values of R1 and R2
for n=1:floor(N/2)
    A2= AAC(n,2); A3= AAC(n,3);
    [C1,C2,Gs] = filter_LPF_8_9a(A2,A3,K,R1,R2,KC) % filter tuning
end
% Check the LPF design results obtained at the web site
% <http://www.daycounter.com/Filters/Sallen-Key-LP-Calculator.phtml>

```

Note. Multiplying/dividing all the resistances/capacitances by the same constant does not change the transfer function and frequency response. This implies that if you want to scale up/down the tuned capacitances/resistances without affecting the transfer function and frequency response, you can scale up/down the predetermined values of resistances/capacitances in the same ratio.

Problems

8.1 Passive Filter

Consider the circuit of Figure P8.1.

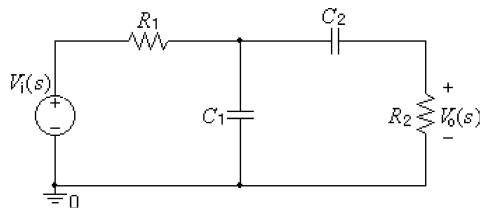


Figure P8.1

- (a) Verify that this circuit has the following transfer function:

$$(s) = \frac{V_o(s)}{V_i(s)} = \frac{s/(R_1 C_1)}{s^2 + s[(1/R_2 C_2) + (1/R_2 C_1) + (1/R_1 C_1)] + (1/R_1 R_2 C_1 C_2)} \quad (\text{P8.1.1})$$

- (b) With $R_1 = 200\ \Omega$, $R_2 = 1\ \text{k}\Omega$, $C_1 = 2.5\ \mu\text{F}$, and $C_2 = 1\ \mu\text{F}$, verify that this circuit has the following frequency response:

$$G(j\omega) = \frac{j2000\omega}{(j\omega)^2 + j\omega_b\omega + \omega_p^2} = \frac{j2000\omega}{(j\omega)^2 + j3400\omega + 2 \times 10^6} \quad (\text{P8.1.2})$$

- (c) Find the resonant frequency ω_r at which the frequency response $G(j\omega)$ is real and the peak magnitude response $G_{\max} = G(j\omega_r)$. Find also the lower and upper 3 dB frequencies.
 (d) Support the results obtained in (c) by PSpice simulation.

8.2 Another Form of Second-Order Bandpass Filter

Consider a system whose transfer function is

$$G(s) = \frac{s + a}{s^2 + \omega_b s + \omega_0^2} \quad (\text{P8.2.1})$$

- (a) Verify that the peak frequency ω_p at which the magnitude of the frequency response, $|G(j\omega)|$, is maximized is

$$\omega_p = \sqrt{-a^2 + \sqrt{(\omega_0^2 + a^2)^2 - (\omega_b a)^2}} \quad (\text{P8.2.2})$$

- (b) Verify that the resonant frequency ω_r at which $G(j\omega)$ is real is

$$\omega_r = \sqrt{\omega_0^2 - a\omega_b} \quad (\text{P8.2.3})$$

Hint. You can run the following MATLAB program `cir08p02.m` to verify (P8.2.2).

```
%cir08p02.m
syms s w wb w0 a
Gw2= (w^2+a^2) / ((w0^2-w^2)^2 + (wb*w)^2);
wrs=solve(diff(Gw2, 'w'))
```

8.3 Passive Second-Order Bandpass Filters

- (a) Verify that the circuit of Figure P8.3(a) has the following transfer function:

$$G(s) = \frac{(s + R_2/L)R_1 C}{s^2 + s[(1/R_1 C) + R_2/L] + (1 + R_2/R_1)/LC} \quad (\text{P8.3.1})$$

- (b) Verify that the circuit of Figure P8.3(b) has the following transfer function:

$$G(s) = \frac{(R_1/L)[s + (1/R_2 C)]}{s^2 + [R_1/L + (1/R_2 C)]s + (1 + R_1/R_2)/LC} \quad (\text{P8.3.2})$$

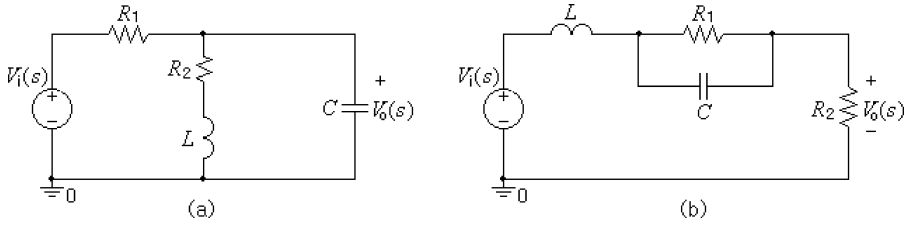


Figure P8.3

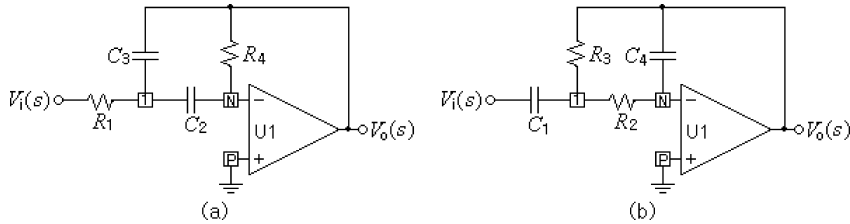


Figure P8.4

8.4 Active Second-Order Bandpass Filters

(a) Verify that the circuit of Figure P8.4(a) has the following transfer function:

$$G(s) = \frac{-s/(R_1 C_3)}{s^2 + s(1/C_2 + 1/C_3)/R_4 + 1/(R_1 R_4 C_2 C_3)} \quad (\text{P8.4.1})$$

(b) Verify that the circuit of Figure P8.4(b) has the following transfer function:

$$G(s) = \frac{-s/(R_2 C_4)}{s^2 + s(1/R_2 + 1/R_3)/C_1 + 1/(R_2 R_3 C_1 C_4)} \quad (\text{P8.4.2})$$

(c) Let $C_2 = C_3 = 100 \text{ nF}$ for the circuit in Figure P8.4(a). Find the values of R_1 and R_4 to make the circuit a bandpass filter (BPF) with the center frequency $\omega_p = 1 \text{ kHz}$ and the quality factor $Q = \omega_p/\omega_b = 10$.

8.5 Active First-Order Filter

Figure P8.5.1 shows two OP Amp circuits.

(a) Find the transfer function $G(s) = V_o(s)/V_i(s)$ of each circuit.

(b) Choose the magnitude/phase curves of the frequency response for each of the following circuits from (a), (b), and (c) in Figure P8.5.2.

- (1) The circuit in Figure P8.5.1(a) with $R_1 = 1 \text{ k}\Omega$, $R_2 = 10 \text{ k}\Omega$, and $C_1 = C_2 = 1 \text{ }\mu\text{F}$. ()
- (2) The circuit in Figure P8.5.1(a) with $R_1 = 10 \text{ k}\Omega$, $R_2 = 1 \text{ k}\Omega$, and $C_1 = C_2 = 1 \text{ }\mu\text{F}$. ()
- (3) The circuit in Figure P8.5.1(b) with $R_1 = 1 \text{ k}\Omega$, $R_2 = 10 \text{ k}\Omega$, and $C = 1 \text{ }\mu\text{F}$. ()

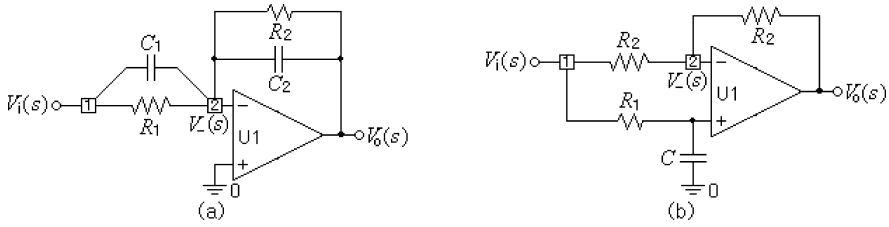


Figure P8.5.1

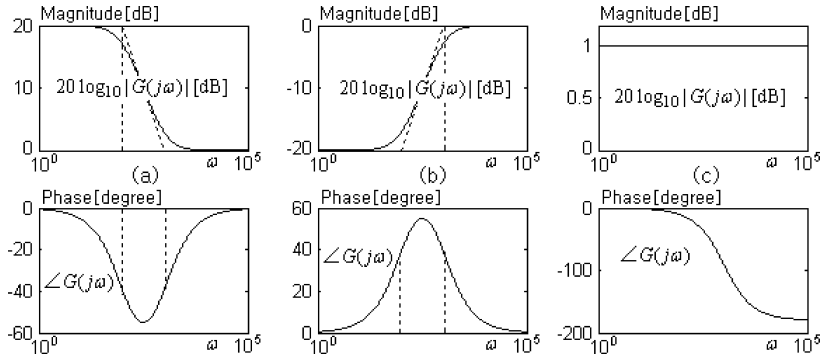


Figure P8.5.2

One of the three circuits is referred to as a *phase-lag* filter since the phase of its frequency response is (locally) negative. Another one is referred to as a *phase-lead* filter since the phase of its frequency response is (locally) positive. The remaining one is referred to as an *all-pass* filter since it has a flat gain, i.e. the magnitude of its frequency response does not vary with the frequency ω . Distinguish the type of filter for each of the above three circuits.

Note. Note that the frequency response (magnitude) curves shown in Figure P8.5.2 are the plots of $20 \log_{10} |G(j\omega)|$ [dB] with the frequency on the horizontal log scale. This magnitude curve together with the phase curve composes a *Bode plot* (or *Bode diagram*), which can easily be obtained by using MATLAB if the transfer function is given as a rational function in s . For example, you can type the following statements into the MATLAB command window to get the Bode plot for

$$G(s) = \frac{6s^2 + 0s + 8}{s^4 + 2s^3 + 3s^2 + 4s + 5}$$

```
>>w=logspace(0,5,1001); % frequency range [ 1, 10^5]
>>num=[ 6 0 8]; den=[ 1 2 3 4 5]; bode(num,den,w)
```

8.6 Lowpass Notch Filter (LPNF), Highpass Notch Filter (HPNF), and Allpass Filter (APF)

Suppose three systems A, B, and C have the following transfer functions (1), (2), and (3), respectively.

$$(1) \quad G(s) = \frac{s^2 + \omega_n^2}{s^2 + \omega_b s + \omega_0^2} \quad \text{with } \omega_n = 200, \quad \omega_b = 100, \quad \omega_0 = 100 \text{ rad/s} \quad (\quad)$$

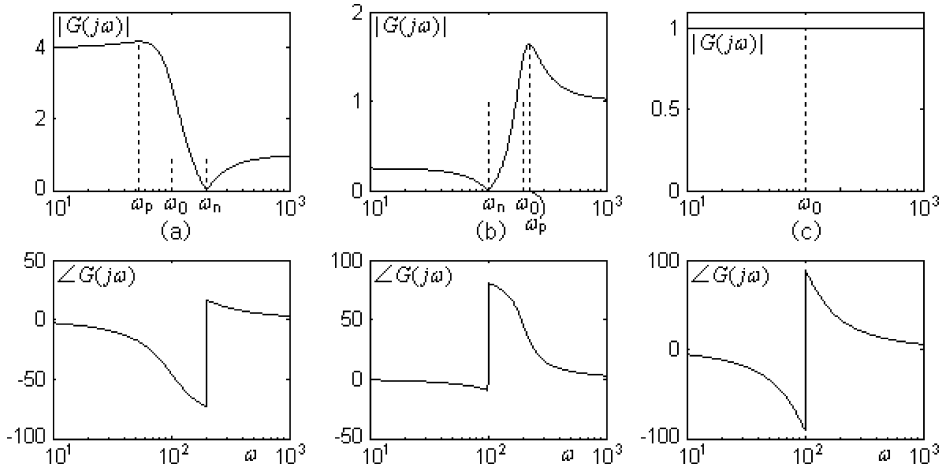


Figure P8.6

$$(2) \quad G(s) = \frac{s^2 + \omega_n^2}{s^2 + \omega_b s + \omega_0^2} \quad \text{with } \omega_n = 100, \quad \omega_b = 100, \quad \omega_0 = 200 \text{ rad/s} \quad ()$$

$$(3) \quad G(s) = \frac{s^2 - \omega_b s + \omega_0^2}{s^2 + \omega_b s + \omega_0^2} \quad \text{with } \omega_b = 100, \quad \omega_0 = 100 \text{ rad/s} \quad ()$$

(a) Choose the magnitude/phase curves of the frequency response for each of them from (a), (b), and (c) in Figure P8.6. One of the three systems is referred to as a *lowpass notch filter* (LPNF) since the magnitude of the response is larger for low frequencies than for high frequencies, also having a notch frequency ω_n at which $|G(j\omega_n)| = 0$. Another one is referred to as a *highpass notch filter* (HPNF) since the magnitude of the response is larger for high frequencies than for low frequencies, also having a notch frequency ω_n at which $|G(j\omega_n)| = 0$. The remaining one is referred to as an *all-pass* (AP) filter since the magnitude of its frequency response is constant, not varying with the frequency ω . Distinguish the type of filter for each of the above systems.

(b) For systems (1) and (2), find the peak frequency ω_p at which the magnitude of the frequency response is maximized and the maximum magnitude $|G(j\omega_p)|$ of the frequency response. Note that the peak frequency differs from ω_0 as seen from Figures P8.6(a) and (b).

Hint. The following MATLAB program `cir08p06.m` might be used.

```
%cir08p06.m
wn = 200; wb=100; w0=100;
Gw = (-w^2 + wn^2) / (-w^2 + j*w*wb + w0^2); % frequency response
wmax = solve(diff((wn^2 - w^2)^2 / ((w0^2 - w^2)^2 + (w*wb)^2)));
wmax = eval(wmax(2)); % symbol-to-double(numeric) conversion
Gmax = abs(subs(Gw, 'w', wmax))
```

8.7 Realization of Various Types of Filter Using OP Amps

(a) Show that the input impedance of the OP Amp circuit in Figure P8.7(a) is

$$Z_{Th}(s) = \frac{V_T(s)}{I_T(s)} = \frac{Z_1(s)Z_3(s)}{Z_2(s)Z_4(s)} Z_5(s) \quad (P8.7.1)$$

(b) Show that the input impedance of the OP Amp circuit in Figure P8.7(b) is

$$Z_{Th}(s) = \frac{V_T(s)}{I_T(s)} = sC_2 \frac{R_1 R_3}{R_4} Z_5(s) \quad (P8.7.2)$$

(c) Show that the transfer function of the circuit in Figure P8.7(c1) equivalent to that in Figure P8.7(c2) is

$$G_c(s) = \frac{V_o(s)}{V_i(s)} = \frac{Z_{Th}(s)}{R_s + Z_{Th}(s)} = \frac{R_3 / (R_s R_2 R_4 C_1 C_5)}{s^2 + s / (R_5 C_5) + R_3 / (R_s R_2 R_4 C_1 C_5)} \quad (P8.7.3)$$

(d) Show that the transfer function of the circuit in Figure P8.7(d1) equivalent to that in Figure P8.7(d2) is

$$G_d(s) = \frac{V_o(s)}{V_i(s)} = \frac{R_s || (sL_{eq})}{1 / (sC_s) + [R_s || (sL_{eq})]} = \frac{s^2}{s^2 + s / (R_s C_s) + R_4 / (R_1 R_3 R_5 C_s C_2)} \quad (P8.7.4)$$

(e) Show that the transfer function of the circuit in Figure P8.7(e1) equivalent to that in Figure P8.7(e2) is

$$G_e(s) = \frac{V_o(s)}{V_i(s)} = \frac{(1/sC_s) || (sL_{eq})}{R_s + [(1/sC_s) || (sL_{eq})]} = \frac{s / (R_s C_s)}{s^2 + s / (R_s C_s) + R_4 / (R_1 R_3 R_5 C_s C_2)} \quad (P8.7.5)$$

(f) Show that the transfer function of the circuit in Figure P8.7(f1) equivalent to that in Figure P8.7(f2) is

$$G_f(s) = \frac{V_o(s)}{V_i(s)} = \frac{R_s}{R_s + [(1/sC_s) || (sL_{eq})]} = \frac{s^2 + R_4 / (R_1 R_3 R_5 C_s C_2)}{s^2 + s / (R_s C_s) + R_4 / (R_1 R_3 R_5 C_s C_2)} \quad (P8.7.6)$$

Note. The two circuits in Figures P8.7(e1) and (f1) are complementary to each other in the sense that the sum of their transfer functions makes unity: $G_e(s) + G_f(s) = 1$. In fact, interchanging the input and the ground in a linear circuit makes its complement.

(g) Show that the transfer function of the circuit in Figure P8.7(g1), which is equivalent to that in Figure P8.7(g2), is

$$G_g(s) = \frac{R_s || (1/sC_{s2})}{[R_s || (1/sC_{s2})] + [(1/sC_{s1}) || (sL_{eq})]} = \frac{\frac{C_{s1}}{C_{s1} + C_{s2}} \left(s^2 + \frac{R_4}{R_1 R_3 R_5 C_2 C_{s1}} \right)}{s^2 + s \frac{1}{R_s (C_{s1} + C_{s2})} + \frac{R_4}{R_1 R_3 R_5 C_2 (C_{s1} + C_{s2})}} \quad (P8.7.7)$$

(h) Show in two ways that the transfer function of the circuit in Figure P8.7(h1) is

$$G_h(s) = \frac{V_o(s)}{V_i(s)} = \frac{s^2 + R_4 / (R_1 R_3 R_{51} C_s C_2)}{s^2 + s / (R_s C_s) + R_4 (1/R_{51} + 1/R_{52}) / (R_1 R_3 C_s C_2)} \quad (P8.7.8)$$

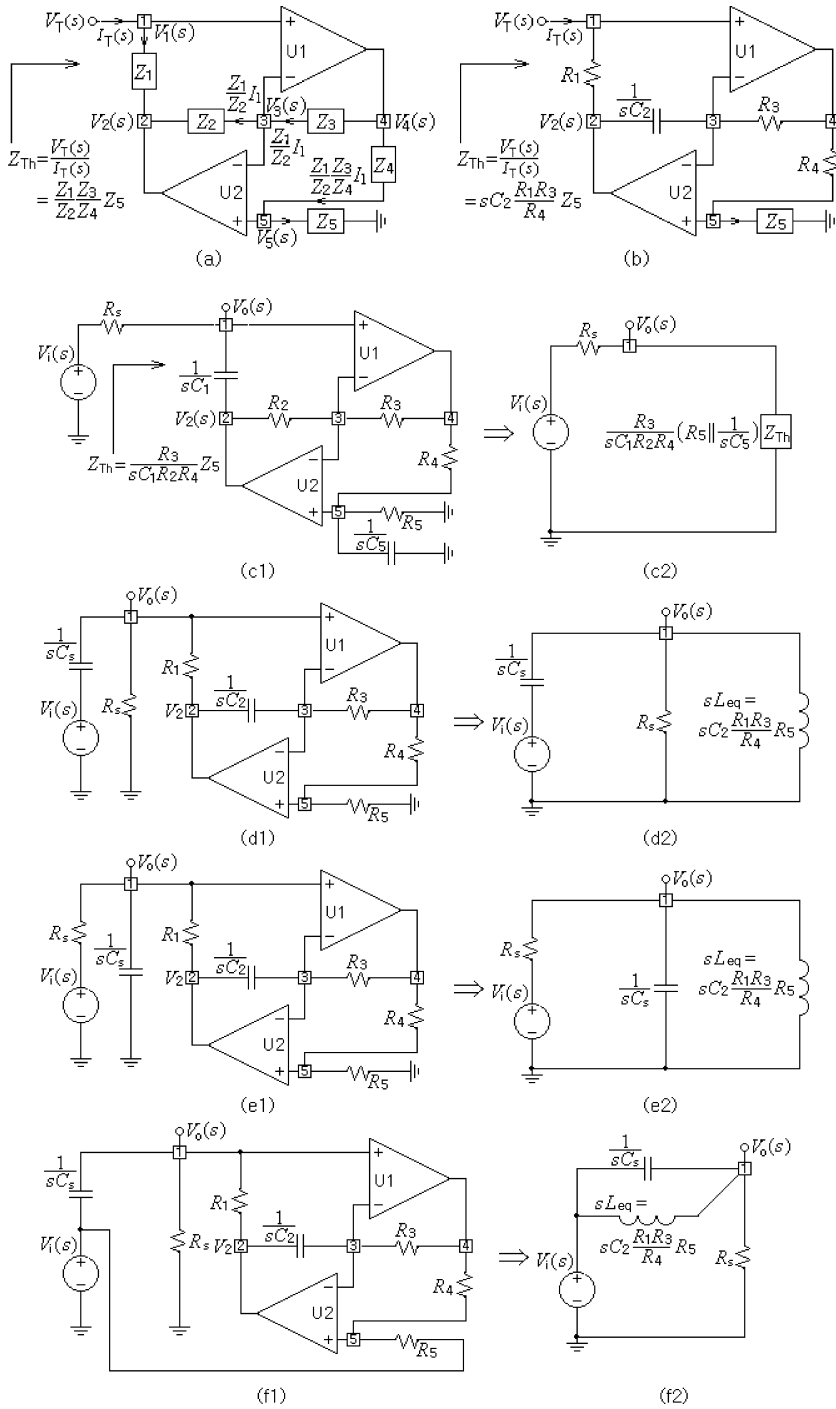


Figure P8.7

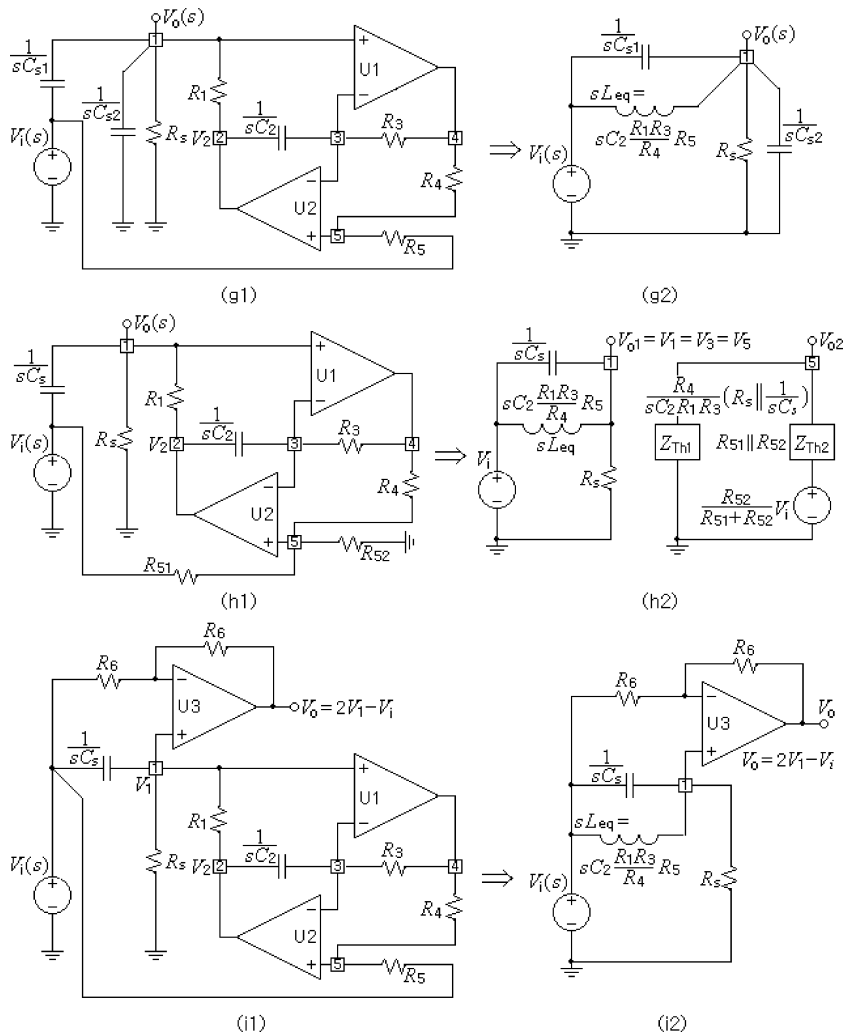


Figure P8.7 (Continued)

First, the superposition principle can be used as suggested in Figure P8.7(h2). Second, you can apply KCL to nodes 1, 3, and 5 to write a set of node equations in the three unknown node voltages $V_1(s) = V_o(s)$, $V_2(s)$, and $V_4(s)$ and solve it.

- (i) Show that the transfer function of the circuit in Figure P8.6(i1), which is equivalent to that in Figure P8.6(i2), is

$$G_i(s) = \frac{s^2 - s/(R_s C_s) + 1/(L_{eq} C_s)}{s^2 + s/(R_s C_s) + 1/(L_{eq} C_s)} \quad (\text{P8.7.9})$$

Note. $2G_e(s) + G_i(s) = 1$.

- (j) For each of the circuits in Figures P8.7(c1), (d1), (e1), (f1), (g1), (h1), and (i1), choose the type of filter from the following set of filter types: {LP(Lowpass), HP(Highpass), BP(Bandpass), BS(Bandstop), LPN(Lowpass Notch), HPN(Highpass Notch), AP(Allpass)}.

8.8 Two-Integrator Biquad Circuits

- (a) Consider the circuit of Figure P8.8(a). Verify that the circuit can have the following transfer functions with $K = R_2/R_1$, $R_3 = R_4$, and $\omega_0 = 1/(RC)$ depending on the output node:

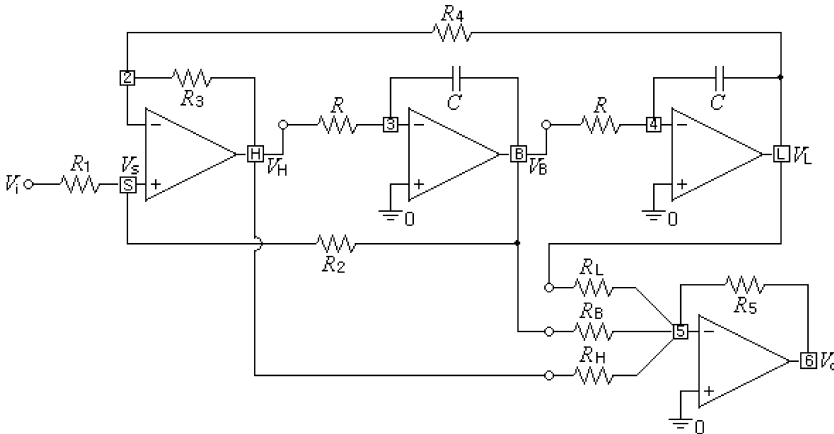
$$G_S(s) = \frac{V_S(s)}{V_i(s)} = \frac{K}{K+1} \frac{s^2 + \omega_0^2}{s^2 + 2s\omega_0/(K+1) + \omega_0^2} \quad (\text{P8.8.1})$$

$$G_H(s) = \frac{V_H(s)}{V_i(s)} = \frac{2K}{K+1} \frac{s^2}{s^2 + 2s\omega_0/(K+1) + \omega_0^2} \quad (\text{P8.8.2})$$

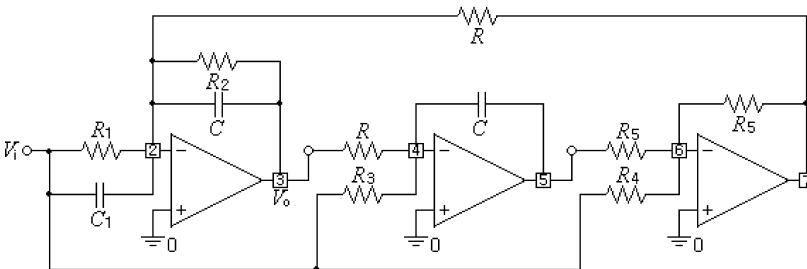
$$G_B(s) = \frac{V_B(s)}{V_i(s)} = \frac{-2K}{K+1} \frac{\omega_0 s}{s^2 + 2s\omega_0/(K+1) + \omega_0^2} \quad (\text{P8.8.3})$$

$$G_L(s) = \frac{V_L(s)}{V_i(s)} = \frac{2K}{K+1} \frac{\omega_0^2}{s^2 + 2s\omega_0/(K+1) + \omega_0^2} \quad (\text{P8.8.4})$$

$$G_a(s) = \frac{V_o(s)}{V_i(s)} = \frac{2K}{K+1} \frac{(R_5/R_H)s^2 - s\omega_0(R_5/R_B) + (R_5/R_L)\omega_0^2}{s^2 + 2s\omega_0/(K+1) + \omega_0^2} \quad (\text{P8.8.5})$$



(a) The Kerwin-Huelsman-Newcomb (KHN) biquad circuit with an OP Amp summer



(b) The Tow-Thomas biquad circuit with feedforward

Figure P8.8

Hint. The following MATLAB program `cir08p08.m` might be used.

```
%cir08p08.m
clear
syms s R1 R2 R3 R4 R C C1 w0 K
%R1=1e3; R2=7e3; R3=1e3; R4=1e3; R=1e3; C=1e-6; sC=s*C;
C=1/w0/R; sC=s*C; R2=K*R1; R4=R3;
Y=[1/R1+1/R2 0 -1/R2 0; -1/R3-1/R4 1/R3 0 1/R4; 0 1/R sC 0; 0 0 1/R sC];
Vs=Y\[1/R1; 0; 0; 0]
```

- (b) Let $R_1 = R_3 = R_4 = R_5 = 1 \text{ k}\Omega$ and $C = 1 \mu\text{F}$ for the circuit in Figure P8.8(a). Find the values of R , R_2 , and R_B to make the circuit function as a bandpass filter (BPF) with a unity peak frequency gain at $\omega_p = 1 \text{ kHz}$ and the quality factor $Q = \omega_p/\omega_b = 4$.
- (c) Verify that the circuit of Figure P8.8(b) has the following transfer function with $K = R_2/R$ and $\omega_0 = 1/(RC)$:

$$G_b(s) = -\frac{s^2(C_1/C) + s\omega_0(R/R_1 - R_5/R_4) + (R/R_3)\omega_0^2}{s^2 + s\omega_0/K + \omega_0^2} \quad (\text{P8.8.6})$$

- (d) For the circuit in Figure P8.8(b) with the following values of parameters, choose the type of filter from {LP(Lowpass), HP(Highpass), positive BP(Bandpass), negative BP, BS(Bandstop), AP(All-pass)}:

(1) $R_1 = \infty$, $R_4 = \infty$, $C_1 = 0$	()
(2) $R_1 = \infty$, $R_3 = \infty$, $R_4 = \infty$	()
(3) $R_1 = \infty$, $R_3 = \infty$, $C_1 = 0$	()
(4) $R_3 = \infty$, $R_4 = \infty$, $C_1 = 0$	()
(5) $R_1 = \infty$, $R_4 = \infty$	()
(1) $R_1 = \infty$, $R_3 = RC/C_1$, $R_4 = KR_5C/C_1 = R_2R_5C/(RC_1)$	()

In particular, letting $R = R_5 = 1 \text{ k}\Omega$ and $C_1 = C = 1 \mu\text{F}$ for the circuit, find the values of R_1 , R_2 , R_3 , and R_4 so that the circuit functions as a bandstop filter (BSF) with a unity rejection frequency gain at $\omega_r = 1 \text{ kHz}$ and the quality factor $Q = \omega_r/\omega_b = 4$.

8.9 Active Filters Using a Bridged-T Network

- (a) Verify that the circuit of Figure P8.9(a) has the following transfer function:

$$G(s) = \frac{V_o(s)}{V_i(s)} = \frac{\sigma/(R_3R_4C_1C_2)}{s^2 + s(1/C_1 + 1/C_2)/R_3 + 1/(R_3R_4C_1C_2)} \quad (\text{P8.9.1})$$

- (b) Verify that the circuit of Figure P8.9(b) has the following transfer function:

$$G(s) = \frac{V_o(s)}{V_i(s)} = \frac{s^2\sigma}{s^2 + s(1/R_1 + 1/R_2)/C_4 + 1/(R_1R_2C_3C_4)} \quad (\text{P8.9.2})$$

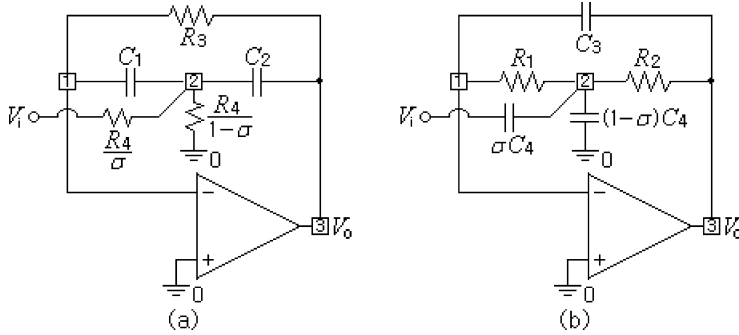


Figure P8.9

8.10 Active Filters

(a) Verify that the circuit of Figure P8.10(a) has the following transfer function:

$$G(s) = \frac{V_o(s)}{V_i(s)} = \frac{s^2}{s^2 + s(1/C_1 + 1/C_2)/R_3 + 1/(R_3 R_4 C_1 C_2)} \quad (\text{P8.10.1})$$

(b) Verify that the circuit of Figure P8.10(b) has the following transfer functions depending on which terminal is taken as the output node.

$$G_1(s) = \frac{V_o(s)}{V_i(s)} = \frac{1/(R_1 R_2 C_3 C_4)}{s^2 + s(1/R_1 + 1/R_2)/C_4 + 1/(R_1 R_2 C_3 C_4)} \quad (\text{P8.10.2})$$

$$G_2(s) = \frac{V_1(s)}{V_i(s)} = \frac{(s + 1/R_2 C_3)/(R_1 C_4)}{s^2 + s(1/R_1 + 1/R_2)/C_4 + 1/(R_1 R_2 C_3 C_4)} \quad (\text{P8.10.3})$$

8.11 Active Filters

(a) Verify that the circuit of Figure P8.11(a) has the following transfer function:

$$G(s) = \frac{V_o(s)}{V_i(s)} = \frac{1/(R_1 R_2 C_3 C_4)}{s^2 + s/(R_1 C_3) + 1/(R_1 R_2 C_3 C_4)} \quad (\text{P8.11.1})$$

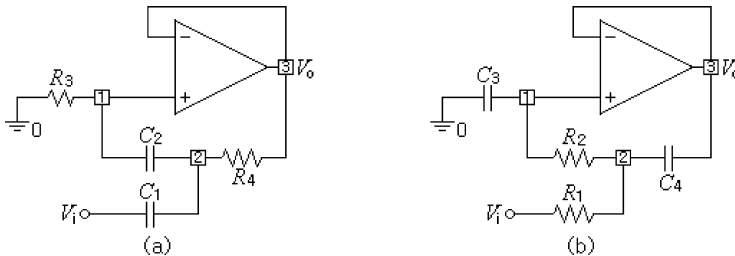


Figure P8.10

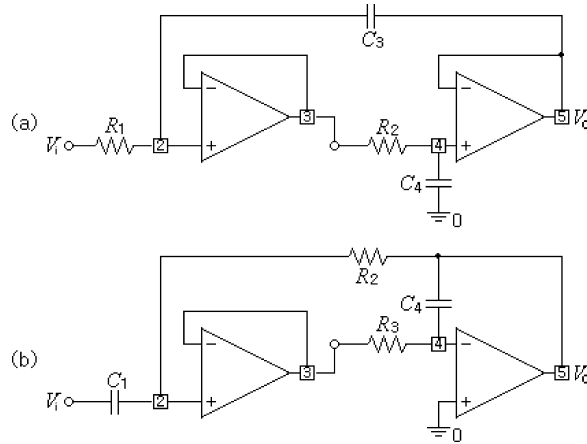


Figure P8.11

(b) Verify that the circuit of Figure P8.11(b) has the following transfer function:

$$G(s) = \frac{V_o(s)}{V_i(s)} = \frac{-s/(R_3 C_4)}{s^2 + s/(R_2 C_1) + 1/(R_2 R_3 C_1 C_4)} \quad (\text{P8.11.2})$$

8.12 Active Filters and Their Frequency Responses in Bode Plots

(a) Verify that the circuits in Figures P8.12(a) and (b) have the following transfer functions, respectively:

$$G_a(s) = \frac{V_o(s)}{V_i(s)} = \frac{-s/(R_1 C_3)}{s^2 + s(1/C_3 + 1/C_4)/R_5 + (1/R_1 + 1/R_2)/(R_5 C_3 C_4)} \quad (\text{P8.12.1})$$

$$G_b(s) = \frac{Ks^2 + s[K(1/C_3 + 1/C_4)/R_5 + (K - 1)/(R_1 C_3)] + K/(R_1 R_5 C_3 C_4)}{s^2 + s(1/C_3 + 1/C_4)/R_5 + 1/(R_1 R_5 C_3 C_4)}$$

$$\text{with } K = \frac{R_6}{R_2 + R_6} \quad (\text{P8.12.2})$$

(b) Using MATLAB or its equivalent, plot the magnitude in dB of the frequency response $G_a(j2\pi f)$, i.e. $20 \log_{10} |G_a(j2\pi f)|$ with $R_1 = 56 \text{ k}\Omega$, $R_2 = 12 \text{ k}\Omega$, $R_5 = 100 \text{ k}\Omega$, $C_3 = 20 \text{ nF}$,

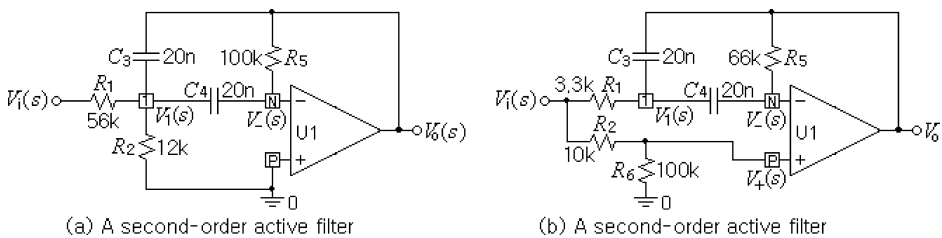


Figure P8.12

and $C_4 = 20 \text{ nF}$ for the 801 log-spaced frequency points between 10 Hz and 1 kHz on a horizontal log scale. Which kind of filter is the circuit of Figure P8.12(a), LPF, HPF, BPF, or BSF? Find the DC gain $|G_a(j0)|$. Find the peak (maximum) magnitude, $G_{a,\max}[\text{dB}]$, of $|G_a(j2\pi f)|$ and the peak frequency f_p . Find the upper/lower 3 dB frequencies f_u and f_l at which $|G_a(j2\pi f)| = G_{a,\max}/\sqrt{2}$ or, equivalently, $|G_a(j2\pi f)| [\text{dB}] = G_{a,\max} - 3 [\text{dB}]$.

Hint. You may complete the following program `cir08p12b.m` and run it.

Note. As introduced at the end of Problem 8.5, `[Gmag,Gph]=bode(num,den,w)` can be used to find the magnitude(`Gmag`) and phase(`Gph`) of the frequency response $G(j\omega)$ for a (radian) frequency vector (`w`), where the numerator/denominator polynomials of $G(j\omega)$ in $s = j\omega$ are given as the first/second input arguments `num` and `den`, respectively; `bode(num,den,w)` with no output argument yields the Bode plot consisting of the magnitude and phase curves.

Note. Since the magnitude in dB of the frequency response has been computed, there is no need to divide the maximum value by $\sqrt{2}$, but simply to subtract 3 dB from the maximum value in dB when finding the upper/lower 3 dB frequencies. This idea also works for the PSpice simulation.

```
%cir08p12b.m
clear, clf
syms s R1 R2 R5 C3 C4
sC3=s*C3; sC4=s*C4;
% Node admittance matrix
Y=[ 1/R1+1/R2+sC3+sC4 -sC3; sC4 1/R5];
% Solution of the node equation
Vs=Y\[ 1/R1; 0];
% Transfer function Eq. (P8.12.1)
Gs=Vs(2)
Gs=subs(Gs,{R1,R2,R5,C3,C4},{56e3,????,1e5,????,2e-8});
pretty(simplify(Gs))
% Numerator and Denominator of the transfer function
[num,den]=numden(Gs); num=sym2poly(num); den=sym2poly(den);
% Frequency vector from 10^1 to 10^3
f=logspace(1,3,801); w=2*pi*f; jw=j*w;
% Magnitude in dB of the frequency response G(jw)
Gw_mag_dB=20*log10(abs(polyval(num,jw)./polyval(den,jw)));
Gw_mag_dB1=20*log10(abs(freqs(num,den,jw))); % Alternatively
subplot(221), semilogx(f,Gw_mag_dB)
% Just to plot magnitude & phase curves of G(jw) - Bode plot
subplot(222), bode(num,den,w)
% To get the numeric values of the magnitude & phase curves of G(jw)
[Gw_mag,Gw_phase]=bode(num,den,w);
% the magnitude in dB of G(jw) using bode()
Gw_mag_dB1=20*log10(Gw_mag); discrepancy=norm(Gw_mag_dB-Gw_mag_dB1.')
% Peak frequency fp of the BPF
[Gm dB_max,imax]=max(Gw_mag_dB), fp=f(imax)
% DC gain at w=0
Gw0=abs(polyval(num,0)./polyval(den,0))
% Upper/lower 3 dB-frequencies
[G1,il]=min(abs(Gw_mag_dB(1:imax)-(Gm dB_max-3))); f1=f(il)
[Gu,iu]=min(abs(Gw_mag_dB(imax:end)-(Gm dB_max-3))); fu=f(imax+iu-1)
```

Table P8.12 Results of the MATLAB analysis and PSpice simulation

		MATLAB	PSpice
Figure P8.12(a)	DC gain $ G_a(j0) $		$10^{-33.1/20} = 0.0221$
	$G_{a,\max}$ [dB]	-0.985	
	Peak frequency f_p [Hz]		253
	Lower 3 dB frequency f_l [Hz]	186	
	Upper 3 dB frequency f_u [Hz]		343
Figure P8.12(b)	$G_{b,\max}$ [dB]	-0.828	
	$G_{b,\min}$ [dB]		-43.2
	Rejection frequency f_r [Hz]	540	
	Lower 3 dB frequency f_l [Hz]		431
	Upper 3 dB frequency f_u [Hz]	672	

- (c) Using MATLAB or its equivalent, plot the magnitude in dB of the frequency response $G_b(j2\pi f)$, i.e. $20 \log_{10} |G_b(j2\pi f)|$ with $R_1 = 3.3 \text{ k}\Omega$, $R_2 = 10 \text{ k}\Omega$, $R_5 = 66 \text{ k}\Omega$, $R_6 = 100 \text{ k}\Omega$, $C_3 = 20 \text{ nF}$, and $C_4 = 20 \text{ nF}$ for the 801 log-spaced frequency points between 10 Hz and 1 kHz on a horizontal log scale. Which kind of filter is the circuit of Figure P8.12(b), LPF, HPF, BPF, or BSF? Find the peak (maximum) magnitude, $G_{b,\max}$, of $|G_b(j2\pi f)|$. Find the minimum magnitude, $G_{b,\min}$, of $|G_b(j2\pi f)|$ and the rejection frequency f_r . Find the upper/lower 3 dB frequencies f_u and f_l at which $|G_b(j2\pi f)| = G_{b,\max}/\sqrt{2}$ or, equivalently, $|G_b(j2\pi f)|$ [dB] = $G_{b,\max} - 3$ [dB].
- (d) Perform the PSpice simulation to obtain the magnitude curve of the frequency responses $|G_a(j2\pi f)|$ and $|G_b(j2\pi f)|$ for 400 frequency points per decade between 10 Hz and 1 kHz on a horizontal log scale. Based on the PSpice simulations together with the MATLAB analysis results obtained in (b) and (c), fill in the corresponding blanks in Table P8.12 with the appropriate values rounded to three significant digits.

Note. To plot $20 \log_{10} |G(j2\pi f)|$ instead of $|G(j2\pi f)|$ in the Probe window, click the ADD Trace button and type the following expression

$$20*\text{LOG10}(\text{ABS}(\text{V}(\text{U1:OUT})))$$

into the Trace Expression field at the bottom of the Add Traces dialog box (see Figure H.8(a) in Appendix H).

8.13 Design and Realization of a High-Order Butterworth Lowpass Filter (LPF)

A unity-gain Butterworth LPF has a frequency response whose magnitude is

$$|G(j\omega)| = \frac{1}{\sqrt{1 + (\omega/\omega_c)^{2N}}} \quad (\text{P8.13.1})$$

where ω_c [rad/s] is the cutoff frequency and N is the order of the filter, i.e. the degree of the denominator polynomial (in s) of the transfer function. It can be seen from the magnitude curves for $N = 1, 2, 3$, and 4 in Figure P8.13.1(a) that the transition at the cutoff frequency becomes sharper as the filter order N increases. If the frequency response magnitude curve of a filter is extremely sharp with no transition band so that it has a discontinuity at the cutoff frequency, it is an ideal (lowpass)

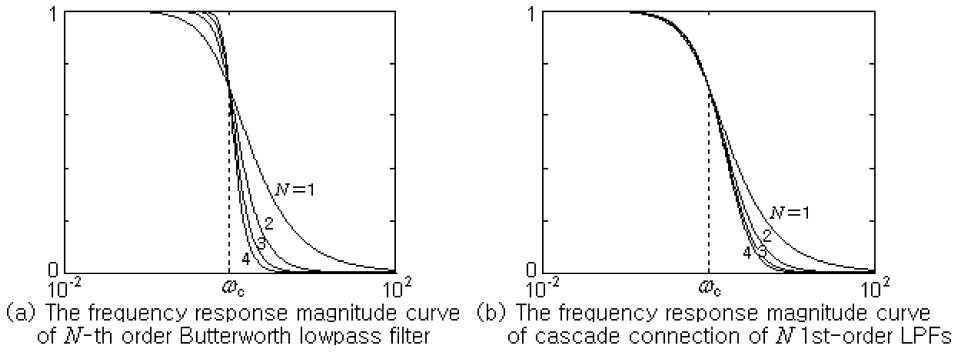


Figure P8.13.1

filter since the frequency components lower/above the cutoff frequency will be completely passed/stopped by the filter. However, an ideal filter cannot be realized by a physical system.

It might be thought that similar frequency responses will be obtained with a cascade connection of N first-order LPFs which has the transfer function

$$G(s) = \frac{\omega_{c1}^N}{(s + \omega_{c1})^N} \quad \text{with } \omega_{c1} = \frac{\omega_c}{\sqrt{2^{1/N} - 1}} \quad (\text{P8.13.2})$$

However, the magnitude curve of the frequency response for this transfer function does not become as sharp as that of the Butterworth filter of the same order, as seen from Figure P8.13.1(b). That is the main reason why the classical type of filters such as Butterworth, Chebyshev I, II, and elliptic filters are preferred to a simple cascade connection of first-order filters. As discussed at the end of Chapter 8, MATLAB functions such as 'butter()' can be used to design such classical filters.

Given the order, say, $N = 4$ and the cutoff frequency, say, $\omega_c = 2\pi \times f_c = 2\pi \times 10^4$ rad/s of an LPF, the following MATLAB statements can be used to find the numerator (B) and denominator (A) polynomials in s of the transfer function of the desired fourth-order Butterworth LPF:

```
>>format short e; N=4; fc=1e4; wc=2*pi*fc;
>>[ B,A]=butter(N,wc,'s')
B =
    0         0         0         0  1.5585e+019
A = 1.0000e+000  1.6419e+005  1.3479e+010  6.4819e+014  1.5585e+019
```

This means the following transfer function:

$$G(s) = \frac{1.5585 \times 10^{19}}{s^4 + 1.6419 \times 10^5 s^3 + 1.3479 \times 10^{10} s^2 + 6.4819 \times 10^{14} s + 1.5585 \times 10^{19}} \quad (\text{P8.13.3})$$

This transfer function can be expressed in the form of the product of second-order transfer functions as

$$G(s) = G_{C1}(s)G_{C2}(s) = \frac{39478 \times 10^5}{s^2 + 48089s + 39478 \times 10^5} \frac{39478 \times 10^5}{s^2 + 116100s + 39478 \times 10^5} \quad (\text{P8.13.4})$$

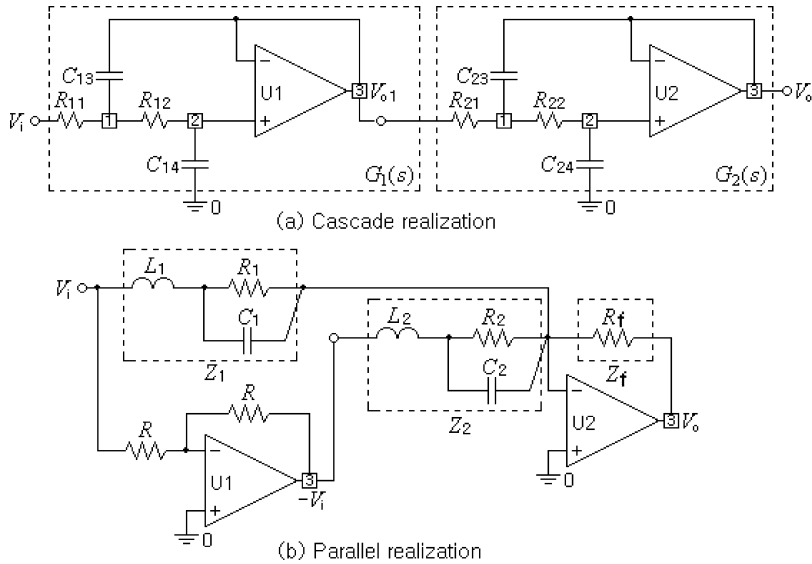


Figure P8.13.2 Two realizations of a fourth-order Butterworth lowpass filter (LPF)

or the sum of second-order transfer functions as

$$G(s) = G_{P1}(s) + G_{P2}(s) = \frac{58049(s + 116100)}{s^2 + 116100s + 39478 \times 10^5} - \frac{58049(s + 48089)}{s^2 + 48089s + 39478 \times 10^5} \quad (\text{P8.13.5})$$

In fact, this implies that the transfer function can be realized as a cascade connection of the two SOSs (second-order sections), $G_{C1}(s)$ and $G_{C2}(s)$, or a parallel connection of the two SOSs, $G_{P1}(s)$ and $G_{P2}(s)$.

(a) Noting that each SOS of the circuit in Figure P8.13.2(a) has the following transfer function:

$$G(s) \stackrel{(8.38)}{=} \frac{1/(R_1 R_2 C_3 C_4)}{s^2 + s(1/R_1 + 1/R_2)/C_3 + 1/(R_1 R_2 C_3 C_4)} \quad (\text{P8.13.6})$$

let $R_{11} = R_{12} = R_{21} = R_{22} = 10 \text{ k}\Omega$ and find the values of C_{13} , C_{14} , C_{23} , and C_{24} such that the desired transfer function will be realized.

(b) Noting that the circuit of Figure P8.13.2(b) has the following transfer function:

$$G(s) = \frac{Z_f(s)}{Z_1(s)} - \frac{Z_f(s)}{Z_2(s)} = -\frac{(R_f/L_1)[s + 1/(R_1 C_1)]}{s^2 + s/(R_1 C_1) + 1/(L_1 C_1)} + \frac{(R_f/L_2)[s + 1/(R_2 C_2)]}{s^2 + s/(R_2 C_2) + 1/(L_2 C_2)} \quad (\text{P8.13.7})$$

let $L_1 = L_2 = L = 0.01 \text{ H}$ and find the values of R_1 , C_1 , R_2 , C_2 , and R_f such that the desired transfer function will be realized.

(c) Plot the magnitude of the frequency response of the filter having the transfer function (P8.13.3) for $f = 1 \sim 100 \text{ kHz}$. Is it smooth for the entire frequency band? Then perform

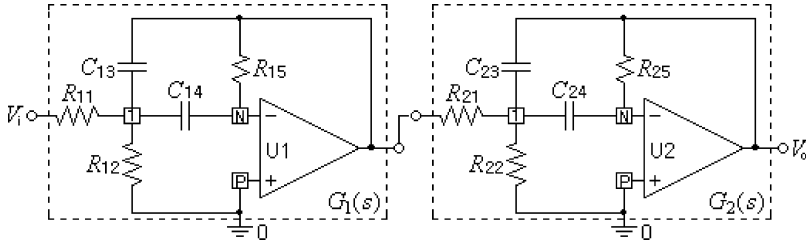


Figure P8.14

the PSpice simulation to see that the cutoff frequency of the two circuits, one realized in cascade form and the other in parallel form, is $f_c = 10\text{ kHz}$ at which the magnitude of the frequency response is $1/\sqrt{2} = 0.7071 = -3\text{ dB}$. The VdB marker (from PSpice/Markers/Advanced/) in the Schematic window can be used to obtain the frequency response in dB.

8.14 Design and Realization of High-Order Chebyshev I Bandpass Filter (BPF)

Given the filter order, say $N = 4$, and the lower/upper 3 dB frequencies, say $\omega_{l,3\text{dB}} = 2\pi \times 8000$ and $\omega_{u,3\text{dB}} = 2\pi \times 12500\text{ rad/s}$ of a BPF, the following MATLAB statements can be used to find the numerator (B) and denominator (A) polynomials in s of the transfer function of the desired fourth-order Chebyshev I BPF.

```
>>format short e; N=4; Rp=3; fl=8000; fu=12500; wl=2*pi*fl; wu=2*pi*fu;
>>[B,A]=cheby1(N/2,Rp,[ wl wu] , 's' )
B =
0 0 4.0067e+008 0 0
A = 1.0000e+000 1.8234e+004 8.4616e+009 7.1985e+013 1.5585e+019
```

This means the following transfer function:

$$G(s) = \frac{4.0067 \times 10^8 s^2}{s^4 + 1.8234 \times 10^4 s^3 + 8.4616 \times 10^9 s^2 + 7.1985 \times 10^{13} s + 1.5585 \times 10^{19}} \quad (\text{P8.14.1})$$

This transfer function can be expressed in the form of the product of two second-order transfer functions as

$$G(s) = G_{C1}(s)G_{C2}(s) = \frac{20000s}{s^2 + 7543s + 27852 \times 10^5} \frac{20000s}{s^2 + 10691s + 55958 \times 10^5} \quad (\text{P8.14.2})$$

In fact, this implies that the transfer function can be realized as a cascade connection of the two second-order sections, $G_{C1}(s)$ and $G_{C2}(s)$.

(a) Noting that each second-order section of the circuit in Figure P8.14 has the following transfer function:

$$G(s) \stackrel{(8.42)}{=} \frac{-(G_1/C_3)s}{s^2 + [G_5(C_3 + C_4)/(C_3C_4)]s + (G_1 + G_2)G_5/(C_3C_4)} \quad (\text{P8.14.3})$$

let $C_{13} = C_{14} = C_{23} = C_{24} = 10 \text{ nF}$ and find the values of R_{11} , R_{12} , R_{15} , R_{21} , R_{22} , and R_{25} such that the desired transfer function will be realized.

- (b) Plot the magnitude curve of the frequency response of the filter having the transfer function (P8.14.1) for $f = 1 \sim 100 \text{ kHz}$. Does it have any ripple in the passband or the stopband? Then perform the PSpice simulation to see that the lower and upper 3 dB frequencies of the circuit are $f_{l,3 \text{ dB}} = 8000$ and $f_{u,3 \text{ dB}} = 12\,500 \text{ Hz}$, respectively, at which the magnitudes of the frequency response are $1/\sqrt{2} = 0.7071 = -3 \text{ dB}$.

9

Circuit Analysis Using Fourier Series

While the preceding Chapters 6, 7, and 8 deal with the AC steady state response of a circuit excited by sinusoidal input sources, this chapter discusses the steady state response of a circuit excited by a nonsinusoidal, but periodic, input of various waveforms such as rectangular waves and half/full-wave rectified sinusoids. The well-known theory on Fourier series states that most periodic signals can be represented as a linear combination (weighted sum) of many sinusoids with different frequencies and phases, called the Fourier series representation. Therefore, by the superposition principle, the outputs to every individual sinusoid at each frequency component can be added to get the overall output as long as the circuit on the load side is linear (see Section 2.8). If the FFT (Fast Fourier Transform) analysis function of PSpice is used, the magnitude of any signal at major frequencies in a circuit can be obtained.

Joseph Fourier (1768–1830) was a French mathematician and physicist who initiated the investigation of Fourier series and its application. Born as the son of a tailor, he was orphaned at the age of 8. In 1784, at only 16 years of age, he became a mathematics teacher at the Ecole Royale Militaire of Auxerre, debarred from entering the army on account of his obscurity and poverty. In 1795, Fourier took a faculty position at the Ecole Normale (Polytechnique) in Paris, which is an elite institution training high school teachers, university professors, and researchers. In 1798, he joined Napoleon's army in its expedition to Egypt as scientific advisor to help establish educational facilities there and carry out archaeological explorations. Napoleon Bonaparte (1769–1821), not yet 30 years old, a great hero in the human history, and Joseph Fourier, a great scholar of about the same age, in their youth were on board the flagship *L'Orient* bound for Alexandria. What were they thinking of when walking around on the deck and looking up the stars twinkling in the sky above the Mediterranean Sea for several nights in May of 1798? One might have dreamed of Julius Caesar, who conquered Egypt about 1800 years before, falling in love with the Queen Cleopatra, or might have paid a tribute to the monumental achievements of the great king Alexander, who conquered one-third of the earth, opening the road between the West and the Orient. The other might have refreshed his memory on what he wrote in his diary on his 21st birthday, 'Yesterday was my 21st birthday, at that age Newton and Pascal had already acquired many claims to immortality', arranging his ideas on Fourier series and heat diffusion, or recollecting his political ideology, which had swept him, getting him very close to the guillotine in the vortex of the French Revolution.

9.1 Fourier Series

A function $x(t)$ of t is said to be *periodic* with period T if $x(t) = x(t + T) \forall t$, where T [s] and $\omega_0 = 2\pi/T$ [rad/s] are referred to as the *fundamental period* and the *fundamental (angular) frequency*, respectively, if T is the smallest positive real number to satisfy the equation for periodicity. According to the theory on Fourier series, any practical periodic function $x(t)$ can be represented as a sum of (infinitely) many

trigonometric (cosine/sine) functions or complex exponential functions, which is called the *Fourier series representation*. There are four forms of Fourier series representation as follows:

Trigonometric Form:

$$x(t) = a_0 + \sum_{k=1}^{\infty} a_k \cos k\omega_0 t + \sum_{k=1}^{\infty} b_k \sin k\omega_0 t \quad \text{with} \quad \omega_0 = \frac{2\pi}{T} \quad (T = \text{the period of } x(t)) \quad (9.1a)$$

where the Fourier coefficients a_0 , a_k , and b_k are

$$\begin{aligned} a_0 &= \frac{1}{T} \int_T x(t) dt \quad (\text{the integral over one period } T) \\ a_k &= \frac{2}{T} \int_T x(t) \cos k\omega_0 t \, dt \\ b_k &= \frac{2}{T} \int_T x(t) \sin k\omega_0 t \, dt \end{aligned} \quad (9.1b)$$

$$\text{Magnitude-and-Phase Form:} \quad x(t) = d_0 + \sum_{k=1}^{\infty} d_k \cos(k\omega_0 t + \phi_k) \quad (9.2a)$$

where the Fourier coefficients are

$$d_0 = a_0, \quad d_k = \sqrt{a_k^2 + b_k^2}, \quad \phi_k = \tan^{-1}(-b_k/a_k) \quad (9.2b)$$

$$\text{Sine-and-Phase Form:} \quad x(t) = d'_0 + \sum_{k=1}^{\infty} d'_k \sin(k\omega_0 t + \phi'_k) \quad (9.3a)$$

where the Fourier coefficients are

$$d'_0 = a_0 = d_0, \quad d'_k = \sqrt{a_k^2 + b_k^2} = d_k, \quad \phi'_k = \tan^{-1}(a_k/b_k) = \phi_k + \pi/2 \quad (9.3b)$$

$$\text{Complex Exponential Form:} \quad x(t) = \frac{1}{T} \sum_{k=-\infty}^{\infty} c_k e^{jk\omega_0 t} \quad (9.4a)$$

where the Fourier coefficients are

$$c_k = \int_T x(t) e^{-jk\omega_0 t} dt \quad (\text{the integral over one period } T) \quad (9.4b)$$

Here, the k th frequency $k\omega_0$ ($k > 1$) with the fundamental frequency $\omega_0 = 2\pi/T = 2\pi f_0$ [rad/s] (T = period) is referred to as the k th *harmonic*. The above four forms of Fourier representation are equivalent and their Fourier coefficients are related to each other as follows:

$$\begin{aligned} c_0 &= \int_T x(t) dt = Td_0 = Ta_0 \\ c_k &= \int_T x(t) e^{-jk\omega_0 t} dt = \int_T x(t) (\cos k\omega_0 t - j \sin k\omega_0 t) dt = \frac{T}{2}(a_k - jb_k) = \frac{T}{2} d_k \angle \phi_k \\ c_{-k} &= \int_T x(t) e^{jk\omega_0 t} dt = \int_T x(t) (\cos k\omega_0 t + j \sin k\omega_0 t) dt = \frac{T}{2}(a_k + jb_k) = \frac{T}{2} d_k \angle -\phi_k = c_k^* \\ a_0 &= \frac{c_0}{T}, \quad a_k = \frac{c_k + c_{-k}}{T} = \frac{2\operatorname{Re}\{c_k\}}{T}, \quad b_k = \frac{c_{-k} - c_k}{jT} = -\frac{2\operatorname{Im}\{c_k\}}{T} \end{aligned}$$

The plot of the Fourier coefficients (9.2b), (9.3b), or (9.4b) versus the frequency $k\omega_0$ is referred to as the *spectrum*, which can be used to describe the spectral contents of a signal, i.e. depict what frequency components are contained in the signal and how they are distributed over the low/medium/high-frequency range. The Fourier analysis performed by PSpice yields the Fourier coefficients (9.3b) and the corresponding spectra of chosen variables (see Example 9.3).

Effects of Vertical/Horizontal Translations of $x(t)$ on the Fourier Coefficients

Translating $x(t)$ along the vertical axis $\pm A$ (+: upward; -: downward) causes only the change of Fourier coefficient $d_0 = a_0$ for $k = 0$ (DC component or average value) by $\pm A$. Translating $x(t)$ along the horizontal (time) axis by $\pm t_1$ (+: rightward; -: leftward) causes only the change of phases (ϕ_k values) by $\mp k\omega_0 t_1$, not affecting the magnitudes d_k or d'_k or $|c_k|$:

$$\begin{aligned} c'_k &\stackrel{(9.4b)}{=} \int_T x(t - t_1) e^{-jk\omega_0 t} dt = \int_T x(t - t_1) e^{-jk\omega_0(t - t_1 + t_1)} dt \\ &= e^{-jk\omega_0 t_1} \int_T x(t - t_1) e^{-jk\omega_0(t - t_1)} dt \stackrel{(9.4b)}{=} c_k e^{-jk\omega_0 t_1} = |c_k| \angle(\phi_k - k\omega_0 t_1) \end{aligned} \quad (9.5)$$

Note that $x(t - t_1)$ is obtained by translating $x(t)$ by t_1 in the positive (rightward) direction for $t_1 > 0$ and by $-t_1$ in the negative (leftward) direction for $t_1 < 0$ along the horizontal (time) axis.

9.2 Computation of Fourier Coefficients Using Symmetry

If a function $x(t)$ has one of the following three symmetries, use can be made of it to reduce the computation for obtaining Fourier coefficients, as summarized in Table 9.1:

- (1) Even symmetry: $x(t) = x(-t)$
- (2) Odd symmetry: $x(t) = -x(-t)$
- (3) Half-wave symmetry: $x(t) = -x(t + T/2)$ (T = the period)

Note that an even (symmetric) function can become an odd (symmetric) one just by translation and vice versa, as illustrated in Fig. 9.1.

Table 9.1 Reduced computation of Fourier coefficients exploiting symmetries of $x(t)$

Fourier coefficients	Symmetry of a periodic function with period T		
	Even symmetry: $x(t) = x(-t)$	Odd symmetry: $x(t) = -x(-t)$	Half-wave symmetry: $x(t) = -x(t + T/2)$
a_0 (DC term)	$a_0 = \frac{2}{T} \int_0^{T/2} x(t) dt$	0	0
a_k (k = odd)	$a_k = \frac{4}{T} \int_0^{T/2} x(t) \cos k\omega_0 t dt$	0	$a_k = \frac{4}{T} \int_0^{T/2} x(t) \cos k\omega_0 t dt$
a_k (k = even)	$a_k = \frac{4}{T} \int_0^{T/2} x(t) \cos k\omega_0 t dt$	0	0
b_k (k = odd)	0	$b_k = \frac{4}{T} \int_0^{T/2} x(t) \sin k\omega_0 t dt$	$b_k = \frac{4}{T} \int_0^{T/2} x(t) \sin k\omega_0 t dt$
b_k (k = even)	0	$b_k = \frac{4}{T} \int_0^{T/2} x(t) \sin k\omega_0 t dt$	0

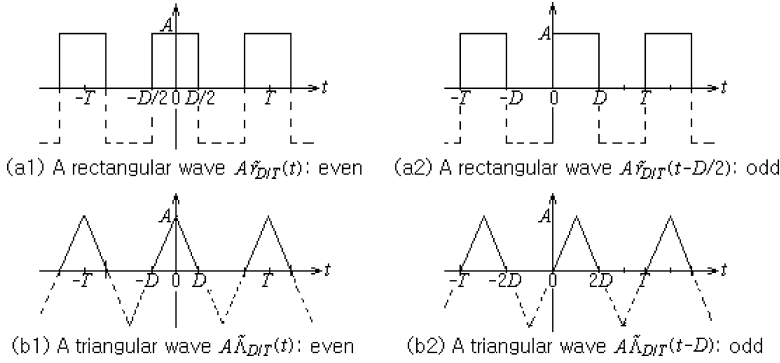


Figure 9.1 Example of rectangular and triangular waves

(Example 9.1) Fourier Spectra of a Rectangular (Square) Wave and a Triangular Wave

(a) Spectrum of a Rectangular (Square) Wave (Figure 9.1(a1))

Consider a rectangular wave $x(t)$ with the height A , duration D , and period T :

$$x(t) = A \tilde{r}_{D/T}(t) \quad \text{where} \quad \tilde{r}_{D/T}(t) = \begin{cases} 1 & \text{for } |t - mT| \leq D/2 \text{ (} m: \text{ an integer)} \\ 0 & \text{elsewhere} \end{cases} \quad (\text{E9.1.1})$$

Equation (9.1b) is used to obtain the Fourier coefficients as

$$a_0 = \frac{1}{T} \int_{-T/2}^{T/2} A \tilde{r}_{D/T}(t) dt = \frac{A}{T} \int_{-D/2}^{D/2} 1 dt = \frac{AD}{T} \quad (\text{E9.1.2})$$

$$a_k^{\text{even}} = \frac{4}{T} \int_0^{D/2} A r_{D/T}(t) \cos k\omega_0 t dt \stackrel{(\text{F.34})}{=} \frac{4A}{T} \left. \frac{\sin k\omega_0 t}{k\omega_0} \right|_0^{D/2} = \frac{2AD}{T} \frac{\sin(k\pi D/T)}{k\pi D/T} \quad (\text{E9.1.3})$$

$$b_k^{\text{even } x(t)} = \frac{2}{T} \int_{-T/2}^{T/2} x(t) \sin k\omega_0 t dt \stackrel{\text{odd}}{=} 0 \quad \text{with} \quad \omega_0 = \frac{2\pi}{T} \quad (\text{E9.1.4})$$

Thus the Fourier series representation of the rectangular wave $x(t)$ can be written as

$$A \tilde{r}_{D/T}(t) \stackrel{(9.1a)}{=} \frac{AD}{T} + \sum_{k=1}^{\infty} \frac{2AD \sin(k\pi D/T)}{k\pi D/T} \cos k\omega_0 t \quad (\text{E9.1.5})$$

In the case of $T = 2D$ as depicted in Figure 9.2(a1), the Fourier coefficients are

$$a_k = \begin{cases} A/2 & \text{for } k = 0 \\ (-1)^m 2A/(k\pi) & \text{for } k = 2m + 1 \\ 0 & \text{for } k = 2m > 0 \text{ (positive even number)} \end{cases} \quad \text{and } b_k = 0 \quad (\text{E9.1.6})$$

and the corresponding (magnitude) spectrum is depicted in Figure 9.2(b1).

(b) Spectrum of a Triangular Wave (Figure 9.1(b1))

Consider a triangular wave $x(t)$ with the maximum height A , duration $2D$, and period T :

$$x(t) = A \tilde{\lambda}_{D/T}(t) \quad \text{where} \quad \tilde{\lambda}_{D/T}(t) = \begin{cases} 1 - t/D & \text{for } |t - mT| \leq D \text{ (} m_0: \text{ an integer)} \\ 0 & \text{elsewhere} \end{cases} \quad (\text{E9.1.7})$$

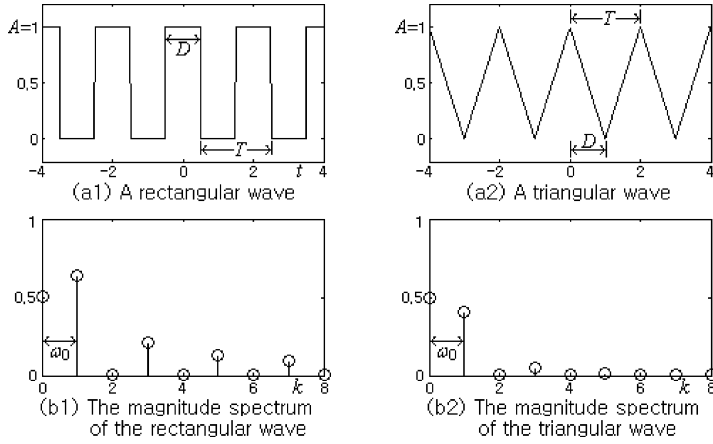


Figure 9.2 Rectangular/triangular waves and their magnitude spectra

Equation (9.1b) is used to obtain the Fourier coefficients as

$$a_0 = \frac{1}{T} \int_{-T/2}^{T/2} A \tilde{\lambda}_{D/T}(t) dt \stackrel{\text{even}}{=} \frac{2A}{T} \int_0^D \left(1 - \frac{t}{D}\right) dt \stackrel{(F.32)}{=} \frac{2A}{T} \left(t - \frac{t^2}{2D}\right) \Big|_0^D = \frac{AD}{T} \quad (E9.1.8)$$

$$\begin{aligned} a_k &\stackrel{\text{even}}{=} \frac{4}{T} \int_0^D A \left(1 - \frac{t}{D}\right) \cos k\omega_0 t dt \quad \text{with} \quad \omega_0 = \frac{2\pi}{T} \\ &\stackrel{(F.34),(F.36)}{=} \frac{4A}{T} \left[\frac{\sin k\omega_0 t}{k\omega_0} \Big|_0^D - \frac{1}{D} \left(\frac{\sin k\omega_0 t}{k\omega_0} t \Big|_0^D - \int_0^D \frac{\sin k\omega_0 t}{k\omega_0} dt \right) \right] \\ &\stackrel{(F.35)}{=} -\frac{4A}{TD} \frac{\cos k\omega_0 t}{(k\omega_0)^2} \Big|_0^D = \frac{4A}{TD} \frac{1 - \cos k\omega_0 D}{(k\omega_0)^2} \\ &\stackrel{(F.14)}{=} \frac{2AD \sin^2(k\omega_0 D/2)}{T (k\omega_0 D/2)^2} = \frac{2AD \sin^2(k\pi D/T)}{T (k\pi D/T)^2} \end{aligned} \quad (E9.1.9)$$

$$b_k \stackrel{\text{even } x(t)}{=} \frac{2}{T} \int_{-T/2}^{T/2} x(t) \sin k\omega_0 t dt \stackrel{\text{odd}}{=} 0 \quad (E9.1.10)$$

Thus the Fourier series representation of the triangular wave $x(t)$ can be written as

$$\tilde{\lambda}_{D/T}(t) \stackrel{(9.1a)}{\stackrel{(E9.1.8),(E9.1.9),(E9.1.10)}}{=} \frac{AD}{T} + \sum_{k=1}^{\infty} \frac{2AD}{T} \frac{\sin^2(k\pi D/T)}{(k\pi D/T)^2} \cos k\omega_0 t \quad (E9.1.11)$$

In the case of $T = 2D$ as depicted in Figure 9.2(a2), the Fourier coefficients are

$$a_k = \begin{cases} A/2 & \text{for } k = 0 \\ 4A/(k\pi)^2 & \text{for } k = 2m + 1 \\ 0 & \text{for } k = 2m > 0 (\text{positive even number}) \end{cases} \quad \text{and } b_k = 0 \quad (E9.1.12)$$

The corresponding (magnitude) spectrum is depicted in Figure 9.2(b2).

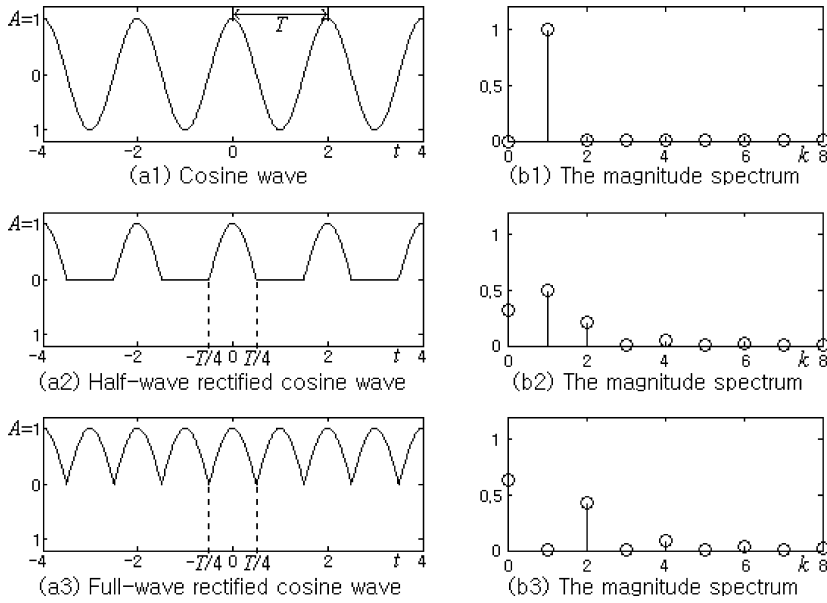


Figure 9.3 Half/full-wave rectified cosine waves and their magnitude spectra

(Example 9.2) Fourier Spectra of Rectified Cosine Waves

(a) Spectrum of a Cosine Wave (Figure 9.3(a1))

Consider a cosine wave $x(t) = A \cos(\omega_0 t)$ with $\omega_0 = 2\pi/T$. In the case of a single-tone periodic signal, the time function is the very Fourier series representation with a single nonzero coefficient

$$a_1 \stackrel{(9.1b)}{=} \frac{2A}{T} \int_{-T/2}^{T/2} \cos \omega_0 t \cos \omega_0 t dt \stackrel{(F.15)}{=} \frac{2A}{T} \int_{-T/2}^{T/2} \frac{1 + \cos 2\omega_0 t}{2} dt = A \quad (E9.2.1)$$

The (magnitude) spectrum is depicted in Figure 9.3(b1).

(b) Spectrum of a Half-Wave Rectified Cosine Wave (Figure 9.3(a2))

Consider a half-wave rectified cosine wave $x_h(t) = \max(A \cos \omega_0 t, 0)$. Equation (9.1b) is used to obtain the Fourier coefficients as

$$a_0 = \frac{1}{T} \int_{-T/4}^{T/4} A \cos \omega_0 t dt \stackrel{\text{even}}{=} \frac{2A}{T} \int_0^{T/4} \cos \omega_0 t dt \stackrel{(F.34)}{=} \frac{2A}{\omega_0 T} \sin \omega_0 t \Big|_0^{T/4} = \frac{A}{\pi} \quad (E9.2.2)$$

$$a_k \stackrel{\text{even}}{=} \frac{4}{T} \int_0^{T/4} A \cos \omega_0 t \cos k\omega_0 t dt \quad \text{with} \quad \omega_0 = \frac{2\pi}{T}$$

$$\stackrel{(F.11)}{=} \frac{2A}{T} \int_0^{T/4} [\cos(k+1)\omega_0 t + \cos(k-1)\omega_0 t] dt$$

$$\stackrel{(F.34)}{=} \frac{2A}{(k+1)\omega_0 T} \sin(k+1)\omega_0 t \Big|_0^{T/4} + \frac{2A}{(k-1)\omega_0 T} \sin(k-1)\omega_0 t \Big|_0^{T/4}$$

$$\stackrel{k \neq 1}{=} \frac{A}{(k+1)\pi} \sin(k+1)\frac{\pi}{2} + \frac{A}{(k-1)\pi} \sin(k-1)\frac{\pi}{2}$$

$$\stackrel{k=2m(\text{even})}{=} (-1)^m \frac{A}{\pi} \left(\frac{1}{k+1} - \frac{1}{k-1} \right) = \frac{(-1)^{m+1} 2A}{(k^2-1)\pi} \quad (E9.2.3)$$

$$a_1 \stackrel{\text{even}}{=} \frac{4}{T} \int_0^{T/4} A \cos^2 \omega_0 t dt \stackrel{(F.15)}{=} \frac{2A}{T} \int_0^{T/4} (1 + \cos 2\omega_0 t) dt = \frac{A}{2} \quad (\text{E9.2.4})$$

$$b_k \stackrel{\text{even } x(t)}{=} \frac{2}{T} \int_{-T/4}^{T/4} A \cos \omega_0 t \sin k\omega_0 t dt \stackrel{\text{odd}}{=} 0 \quad (\text{E9.2.5})$$

Thus we can write the Fourier series representation of the half-wave rectified cosine wave as

$$x_h(t) \stackrel{(9.1a)}{=} \frac{A}{\pi} + \frac{A}{2} \cos \omega_0 t - \sum_{k=2m}^{\infty} \frac{(-1)^m 2A}{(k-1)^2 \pi} \cos k\omega_0 t \quad (\text{E9.2.6})$$

The corresponding (magnitude) spectrum is depicted in Figure 9.3(b2).

(c) Spectrum of a Full-Wave Rectified Cosine Wave (Figure 9.3(a3))

Consider a full-wave rectified cosine wave $x_f(t) = |A \cos \omega_0 t|$, which can be regarded as the sum of a half-wave rectified cosine wave $x_h(t)$ and its $T/2$ -shifted version $x_h(t - T/2)$. Thus, instead of using Equation (9.1b), Equation (E9.2.6) is used to write its Fourier series representation as

$$\begin{aligned} x_f(t) &= x_h(t) + x_h\left(t - \frac{T}{2}\right) \stackrel{(E9.2.6)}{=} \frac{A}{\pi} + \frac{A}{2} \cos \omega_0 t - \sum_{k=2m}^{\infty} \frac{(-1)^m 2A}{(k-1)^2 \pi} \cos k\omega_0 t \\ &\quad + \frac{A}{\pi} + \frac{A}{2} \cos \omega_0 \left(t - \frac{T}{2}\right) - \sum_{k=2m}^{\infty} \frac{(-1)^m 2A}{(k-1)^2 \pi} \cos k\omega_0 \left(t - \frac{T}{2}\right) \quad \text{with } \omega_0 T = 2\pi \\ &\stackrel{(F.4)}{=} \frac{2A}{\pi} - \sum_{k=2m}^{\infty} \frac{(-1)^m 4A}{(k-1)^2 \pi} \cos k\omega_0 t \stackrel{k=2m}{=} \frac{2A}{\pi} - \sum_{m=1}^{\infty} \frac{(-1)^m 4A}{(2m-1)^2 \pi} \cos 2m\omega_0 t \end{aligned} \quad (\text{E9.2.7})$$

The corresponding (magnitude) spectrum is depicted in Figure 9.3(b3).

Note. In fact, the fundamental frequency of a full-wave rectified cosine wave is $2\omega_0 = 4\pi/T$, which is two times that of the original cosine wave.

(d) MATLAB Program to Get the Fourier Spectrum

Once you have saved a periodic function as an M-file, you can use the MATLAB routine `CtFS_trigonometric ( )` to find its trigonometric Fourier series coefficients, as illustrated below. Interested readers are invited to run the program `cir09e02.m` (see next page) to get the Fourier coefficients and the spectra for a pure cosine wave and half/full-wave rectified cosine waves in Figure 9.3.

9.3 Circuit Analysis Using Fourier Series

In the previous section we discussed how to represent a periodic function in Fourier series. In this section we study how to analyze a linear circuit excited by a source whose value can be described as a periodic function of time, which is not necessarily a sinusoidal function. The procedure which we take to use Fourier series for analyzing circuits is as follows:

1. Find the Fourier series representation of the periodic input function.
2. Based on the fact that a (linear) system having the transfer function $G(s)$ responds to a sinusoidal input of frequency $k\omega_0$ with the frequency response $G(jk\omega_0)$ (Section 4.6), apply the phasor method (Section 6.2) to find the individual responses to usually a few main terms, not necessarily every harmonic term, of the input represented in Fourier series.
3. If necessary, add the DC and the fundamental frequency and harmonic components of output based on the superposition principle to find the total response.

Note. The superposition principle holds only for linear systems (see Section 4.5.1).

```

%cir09e02.m: to plot Fig. 9.3
%plots the CtFS spectra of (half/full-rectified) cosine waves
clear, clf
global T, T=2; w0=2*pi/T; % Period and fundamental frequency
N=8; kk=1:N; % frequency indices
tt=[ -400:400]*T/200; % time interval of 4 periods
for i=1:3
    if i==1
        x='cos_wave'; % Fourier coefficients from analytical results
        a0_t=0; a_t=zeros(1,N); a_t(1)=1; b_t=zeros(1,N); % (E9.2.1)
    elseif i==2
        x='cos_wave_half_rectified';
        a0_t=1/pi; a_t=[1/2 zeros(1,N-1)]; b_t=zeros(1,N); % (E9.2.3)
        for m=1:floor(N/2), a_t(2*m)=-(-1)^m*2/(4*m^2-1)/pi; end
    else
        x='cos_wave_full_rectified';
        a0_t=2/pi; a_t=zeros(1,N); b_t=zeros(1,N); % (E9.2.7)
        for m=1:floor(N/2), a_t(2*m)=-(-1)^m*4/(4*m^2-1)/pi; end
    end
    xt=feval(x,tt); % original signal
    [a0,a,b]=CtFS_trigonometric(x,T,N);
    discrepancy_between_numeric_and_analytic=[a0 a b]-[a0_t a_t b_t]
    xht=a0;
    for k=1:N % restore x(t) from its Fourier series representation (9.1a)
        xht=xht+a(k)*cos(k*w0*tt)+b(k)*sin(k*w0*tt);
    end
    subplot(319+i*2), plot(tt,xt,'k-', tt,xht,'r:')
    d1=[a0 sqrt(a.^2+b.^2)]; philk=[0 atan2(a,b)]; % Eq. (9.3b)
    subplot(320+i*2), stem([0:length(d1)-1], d1)
end

```

```

function [a0,a,b]=CtFS_trigonometric(x,T,N)
% Find the Fourier coefficients a0, ak, bk for k=1:N
% x: A periodic function with period T
% T: Period, N: Maximum order of Fourier coefficients
w0=2*pi/T; % the fundamental frequency [rad/s]
xcoskw0t=[x '(t).*cos(k*w0*t)']; x_coskw0t=inline(xcoskw0t,'t','k','w0');
xsinkw0t=[x '(t).*sin(k*w0*t)']; x_sinkw0t=inline(xsinkw0t,'t','k','w0');
tol=1e-6; % toleration on numerical error
a0=quadl(x_coskw0t,-T/2,T/2,tol,[],0,w0)/T; % Eq. (9.1a)
for k=1:N
    a(k)=2/T*quadl(x_coskw0t,-T/2,T/2,tol,[],k,w0); % Eq. (9.1a)
    b(k)=2/T*quadl(x_sinkw0t,-T/2,T/2,tol,[],k,w0); % Eq. (9.1a)
end

```

```

function y=cos_wave(t)
global T, y=cos(2*pi/T*t);

```

```

function y=cos_wave_half_rectified(t)
global T, y=max(cos(2*pi/T*t),0);

```

```

function y=cos_wave_full_rectified(t)
global T, y=abs(cos(2*pi/T*t));

```

(Example 9.3) RC Circuit Excited by a Rectangular (Square)-Wave Voltage Source

Figure 9.4(a) shows the PSpice schematic of an RC circuit excited by a rectangular (square)-wave voltage source of height $\pm V_m = \pm \pi$, period $T = 2$ s, and duration (pulsewidth) $D = 1$ s, where the voltage across the capacitor is taken as the output. Figure 9.4(b) shows the input and output waveforms obtained from the PSpice simulation. Figure 9.4(c) shows the Fourier spectra of the input and output obtained by clicking the FFT button on the toolbar in the PSpice A/D (Probe) window. Figure 9.4(d) shows how to fill in the Simulation Settings dialog box to get the Fourier analysis results (for chosen variables) printed in the output file. Figure 9.4(e) shows the contents of the output file, which can be viewed by clicking PSpice/View_Output_File on the top menu bar and pulldown menu in the Capture window or View/Output_File on the top menu bar and pulldown menu in the Probe window.

- (a) Find the three leading frequency components of the input $v_i(t)$ and output $v_o(t)$. Rather than applying Equation (9.1b) to get the Fourier coefficients of the rectangular wave input $v_i(t)$, use will be made of the result obtained in Example 9.1. Note the following:

- A rectangular wave with height $A = 2\pi$, period $T = 2$, and duration $D = 1$ can be expressed by the Fourier series representation as follows:

$$x(t) \stackrel{(E9.1.5)}{=} \frac{AD}{T} + \sum_{k=1}^{\infty} \frac{2AD}{T} \frac{\sin(k\pi D/T)}{k\pi D/T} \cos k\omega_0 t = \pi + \sum_{k=1}^{\infty} 2\pi \frac{\sin(k\pi/2)}{k\pi/2} \cos k\pi t \quad (E9.3.1)$$

- The rectangular wave input $v_i(t)$ (in Figure 9.4(b)) can be obtained by translating this rectangular wave downward by π and rightward by $T/4 = 0.5$.

Thus $v_i(t)$ can be expressed as

$$\begin{aligned} v_i(t) &= x(t - 0.5) - \pi = \sum_{k=1}^{\infty} 2\pi \frac{\sin(k\pi/2)}{k\pi/2} \cos k\pi(t - 0.5) \\ &= \sum_{k=\text{odd}}^{\infty} \frac{4}{k} \sin k\pi t = 4 \sin \pi t + \frac{4}{3} \sin 3\pi t + \frac{4}{5} \sin 5\pi t + \dots \end{aligned} \quad (E9.3.2)$$

Since the RC circuit has the frequency response

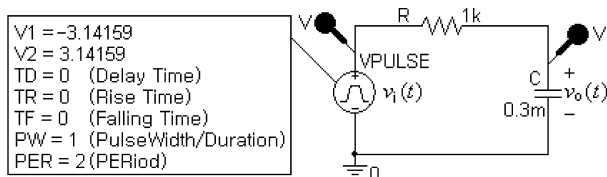
$$G(s) = \frac{1/(sC)}{R + 1/(sC)} = \frac{1}{1 + sRC}; \quad G(j\omega) = \frac{1}{1 + j\omega RC} = \frac{1}{\sqrt{1 + (\omega RC)^2}} \angle -\tan^{-1} \omega RC \quad (E9.3.3)$$

its output to the input $v_i(t)$ (described by Equation (E9.3.2)) can be written as

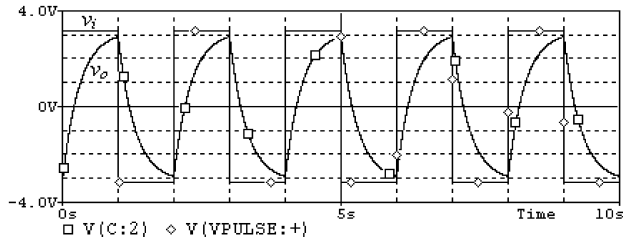
$$v_o(t) = \sum_{k=2m+1}^{\infty} \frac{4}{k\sqrt{1 + (k\omega_0 RC)^2}} \sin(k\omega_0 t - \tan^{-1} k\omega_0 RC) \quad (E9.3.4)$$

The three leading frequency components of the output phasor are

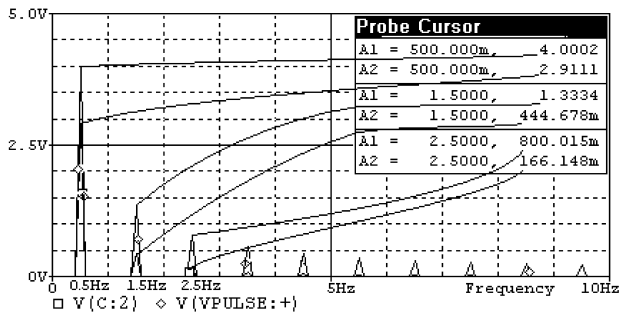
$$\begin{aligned} \mathbf{V}_o^{(1)} &\stackrel{k=1}{=} G(j\omega_0) \mathbf{V}_i^{(1)} = \frac{4}{\sqrt{1 + (\omega_0 RC)^2}} \angle -\tan^{-1} \omega_0 RC \\ &\stackrel{RC=0.3}{=} \frac{4}{\sqrt{1 + (0.3\pi)^2}} \angle -\tan^{-1} 0.3\pi = 2.91 \angle -43.3^\circ \end{aligned} \quad (E9.3.5)$$



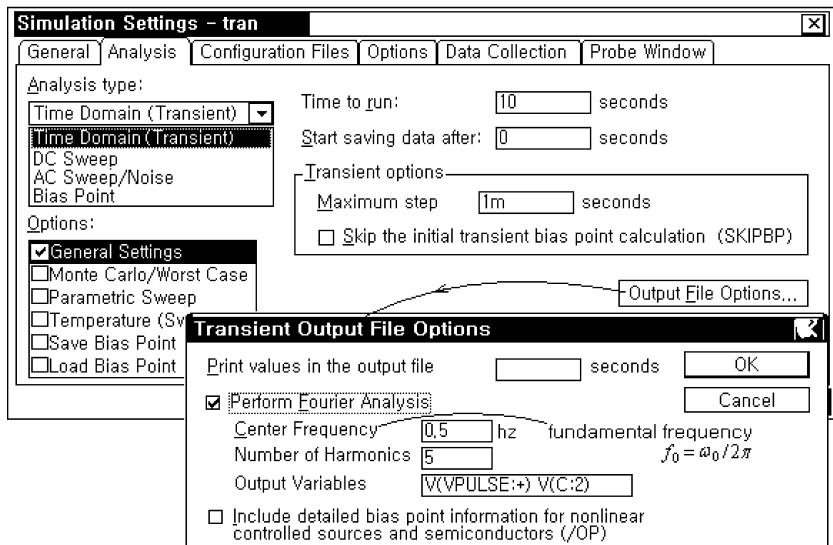
(a) PSpice schematic for the RC circuit excited by a square-wave source



(b) The input and output voltage waveforms



(c) The FFT spectra of the input and output voltages



(d) Simulation Settings window and Transient Output File Options dialog box

Figure 9.4 PSpice simulation and analysis for Example 9.3

```

C:\WPCAD\WYANG\cir09e03-PSpiceFiles\WSchematic1\Wtran\Wtran.out
** Profile: "SCHEMATIC1-tran" [C:\ORCAD\cir09e03-pspicefiles\tran.sim]
*Analysis directives:
.TRAN 0 10 0 1m
.FOUR 0.5 5 V([N00183]) V([N00070])

**** INCLUDING SCHEMATIC1.net ****
(* source CIR09E03
R_R      N00183 N00070 1k
C_C      0 N00070 0.3m
V_VPULSE N00183 0
+PULSE -3.14159 3.14159 0 0 0 1 2
.END)
VOLTAGE SOURCE CURRENTS
NAME      CURRENT
V_VPULSE  0.000E+00
TOTAL POWER DISSIPATION  0.00E+00 WATTS

****      FOURIER ANALYSIS      TEMPERATURE = 27.000 DEG C
FOURIER COMPONENTS OF TRANSIENT RESPONSE V(N00183)
DC COMPONENT = 3.140322E-03(d'_0)


| HARMONIC<br>NO(k) | FREQUENCY<br>(HZ) | FOURIER<br>COMPONENT | NORMALIZED<br>COMPONENT | PHASE<br>(DEG)     | NORMALIZED<br>PHASE (DEG) |
|-------------------|-------------------|----------------------|-------------------------|--------------------|---------------------------|
| 1                 | 5.000E-01(f_0)    | 4.000E+00(d'_1)      | 1.000E+00               | -1.800E-01(phi'_1) | 0.000E+00                 |
| 2                 | 1.000E+00(2f_0)   | 6.281E-03(d'_2)      | 1.570E-03               | 8.964E+01(phi'_2)  | 9.000E+01                 |
| 3                 | 1.500E+00(3f_0)   | 1.333E+00(d'_3)      | 3.333E-01               | -5.401E-01(phi'_3) | -7.176E-10                |
| 4                 | 2.000E+00(4f_0)   | 6.281E-03(d'_4)      | 1.570E-03               | 8.928E+01(phi'_4)  | 9.000E+01                 |
| 5                 | 2.500E+00(5f_0)   | 8.000E-01(d'_5)      | 2.000E-01               | -9.002E-01(phi'_5) | -3.588E-09                |


TOTAL HARMONIC DISTORTION = 3.887326E+01 PERCENT

FOURIER COMPONENTS OF TRANSIENT RESPONSE V(N00070)
DC COMPONENT = 3.141583E-03(d'_0)


| HARMONIC<br>NO(k) | FREQUENCY<br>(HZ) | FOURIER<br>COMPONENT | NORMALIZED<br>COMPONENT | PHASE<br>(DEG)     | NORMALIZED<br>PHASE (DEG) |
|-------------------|-------------------|----------------------|-------------------------|--------------------|---------------------------|
| 1                 | 5.000E-01(f_0)    | 2.911E+00(d'_1)      | 1.000E+00               | -4.348E+01(phi'_1) | 0.000E+00                 |
| 2                 | 1.000E+00(2f_0)   | 2.945E-03(d'_2)      | 1.012E-03               | 2.759E+01(phi'_2)  | 1.146E+02                 |
| 3                 | 1.500E+00(3f_0)   | 4.446E-01(d'_3)      | 1.527E-01               | -7.106E+01(phi'_3) | 5.939E+01                 |
| 4                 | 2.000E+00(4f_0)   | 1.611E-03(d'_4)      | 5.534E-04               | 1.414E+01(phi'_4)  | 1.881E+02                 |
| 5                 | 2.500E+00(5f_0)   | 1.661E-01(d'_5)      | 5.705E-02               | -7.892E+01(phi'_5) | 1.385E+02                 |


TOTAL HARMONIC DISTORTION = 1.630388E+01 PERCENT

JOB CONCLUDED

```

(e) The PSpice output file with FFT analysis

Figure 9.4 (Continued)

$$\begin{aligned}
 \mathbf{V}_o^{(3)} &\stackrel{k=3}{=} G(j3\omega_0) \mathbf{V}_i^{(3)} = \frac{4}{3\sqrt{1 + (3\omega_0 RC)^2}} \angle -\tan^{-1} 3\omega_0 RC \\
 &\stackrel{RC=0.3}{=} \frac{4}{3\sqrt{1 + (0.9\pi)^2}} \angle -\tan^{-1} 0.9\pi = 0.4446 \angle -70.5^\circ
 \end{aligned}
 \tag{E9.3.6}$$

$$\begin{aligned}
 \mathbf{V}_o^{(5)} &\stackrel{k=5}{=} G(j5\omega_0) \mathbf{V}_i^{(5)} = \frac{4}{5\sqrt{1 + (5\omega_0 RC)^2}} \angle -\tan^{-1} 5\omega_0 RC \\
 &\stackrel{RC=0.3}{=} \frac{4}{5\sqrt{1 + (1.5\pi)^2}} \angle -\tan^{-1} 1.5\pi = 0.166 \angle -78^\circ
 \end{aligned}
 \tag{E9.3.7}$$

where the phases are taken with that of $\sin \omega t$ as a reference as in Equation (9.3b) (called the sine-and-phase form of the Fourier series representation) in order to match this Fourier analysis result with that seen in the PSpice simulation output file (Figure 9.4(e)). The magnitude ratios among the leading three frequency components of the input and output are

$$\begin{aligned}\text{Input : } |V_1^{(1)}| : |V_1^{(3)}| : |V_1^{(5)}| &= 4 : \frac{4}{3} : \frac{5}{3} = 15 : 5 : 3 \\ \text{Output : } |V_0^{(1)}| : |V_0^{(3)}| : |V_0^{(5)}| &= 2.91 : 0.4446 : 0.166 = 15 : 2.3 : 0.86\end{aligned}$$

This implies that the relative magnitudes of high-frequency components to low ones become smaller after the input signal passes through the filter, resulting in a considerable decrease of the total harmonic distortion (THD) from 38.9 % to 16.3 %, as can be seen in the output file (Figure 9.4(e)). This is a piece of evidence that the RC circuit with the capacitor voltage taken as the output functions as a lowpass filter.

Note. See Section 9.5.1 for the definition of total harmonic distortion (THD).

- (b) Determine the minimum value of RC required to keep the relative magnitude of the second leading frequency ($3\omega_0$) component to the first one (ω_0) not larger than, say, 12.5 %:

$$\frac{4}{3\sqrt{1 + (3\omega_0 RC)^2}} \leq \frac{4}{\sqrt{1 + (\omega_0 RC)^2}} \times 0.125; \quad RC \geq \frac{\sqrt{3.2353}}{\omega_0} = \frac{1.8}{\pi} = 0.5725 \quad (\text{E9.3.8})$$

- (c) The Fourier analysis results may be obtained by running the following MATLAB program `cir09e03.m`, which uses the MATLAB function `Fourier_analysis()` declared as

```
function [yt, Y, X, THDy, THDx] = Fourier_analysis(n, d, x, T, N)
```

This function takes the numerator (n) and denominator (d) of the transfer function $G(s)$, the (periodic) input function (x) defined for at least one period $[-T/2, +T/2]$, the period (T), and the order (N) of Fourier analysis as the input arguments and produces the output (yt) for one period and the sine-and-phase form of Fourier coefficients Y and X of the output and input (for $k = 0, \dots, N$), and the THDs of the output and input as the output arguments.

```
%cir09e03.m: to solve Example 9.3
% Perform Fourier analysis to solve the RC circuit excited by a square wave
clear, clf
global T D Vm
T=2; w0=2*pi/T; D=1; Vm=pi; % Period, Frequency, Duration, Amplitude
N=5; kk=0:N; % Frequency indices to analyze using Fourier analysis
tt=[-300:300]*T/200; % Time interval of 3 periods
x= 'f_cir09e03'; % Bipolar square wave input function defined in an M-file
RC=0.3; n=1; d=[RC 1]; % Numerator/Denominator of transfer function (E9.3.3)
[Y, X, THDy, THDx] = Fourier_analysis(n, d, x, T, N);
Magnitude_and_phase_of_output_spectrum = ...
    [kk; abs(Y); angle(Y)*180/pi] . '
xt = feval(x, tt); % the input signal for tt
yt= Y(1); % DC component of the output signal
for k=1:N % Fourier series representation of the output signal
    yt = yt + abs(Y(k+1))*sin(k*w0*tt + angle(Y(k+1))); % Eq. (9.3a)
end
subplot(221), plot(tt, xt, tt, yt, 'r') % plot input/output signal waveform
subplot(222), stem(kk, abs(X)), hold on, stem(kk, abs(Y), 'r') % their spectra
```

```

function y=f_cir09e03(t)
% defines a bipolar square wave with period T, duration D, and amplitude Vm
global T D Vm
t=mod(t,T); y= ((t<=D) - (t>D))*Vm;

function[Y,X,THDy,THDx]=Fourier_analysis(n,d,x,T,N)
%Input:  n= the numerator polynomial of transfer function G(s)
%        d= the denominator polynomial of transfer function G(s)
%        x= the input periodic function
%        T= the period of the input function
%        N= the highest frequency index of Fourier analysis
%Output: Y= the Fourier coefficients[Y0,Y1,Y2,...] of the output
%        X= the Fourier coefficients[X0,X1,X2,...] of the input
%        THDy= the Total Harmonic Distortion factor of the output
%        THDx= the Total Harmonic Distortion factor of the input
% Copyleft: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only
if nargin<5, N=10; end
w0=2*pi/T; % Fundamental frequency
kk=0:N; % Frequency index vector
[a0,a,b]=CtFS_trigonometric(x,T,N); % trigonometric Fourier coefficients
Xmag=[a0 sqrt(a.^2+b.^2)]; Xph=[0 atan2(a,b)]; % Eq. (9.3b)
X=Xmag.*exp(j*Xph); % Input spectrum
Gw=freqs(n,d,kk*w0); % Frequency response
Y=X.*Gw; % Output spectrum
%Total Harmonic Distortion factors of the input and output
THDx= sqrt(Xmag(3:end)*Xmag(3:end)')/Xmag(2) % Eq. (9.15)
Ymag=abs(Y); % the magnitude of the output spectrum
THDy= sqrt(Ymag(3:end)*Ymag(3:end)')/Ymag(2)

```

(Example 9.4) Lowpass Filtering of a Full-Wave Rectified Cosine Wave

Figure 9.5(a) shows the PSpice schematic of a rectifier circuit in which a full-wave rectified voltage $v_i(t) = |10 \cos(2\pi 60t)|$ is made smooth by an LCR lowpass filter, as can be observed from the input and output signals and their spectra in Figures 9.5(b) and (c). We will try to find the condition on LC that should be satisfied to keep the relative magnitude of the major harmonic component to the DC component less than $r_{\max}$, say 5 %. To this end, we will find the two leading frequency components (including the DC term) of the input $v_i(t)$ and output $v_o(t)$.

Since the input voltage $v_i(t)$ to the LCR filter is a full-wave rectified cosine wave, Equation (E9.2.7) (in Example 9.2) can be used to write its Fourier series representation as

$$v_i(t) \stackrel{(E9.2.7)}{=} \frac{2A}{\pi} - \sum_{m=1}^{\infty} \frac{(-1)^m 4A}{(2m-1)^2 \pi} \cos 2m\omega_0 t$$

$$\stackrel{A=10}{\omega_0=2\pi 60=377} \frac{20}{\pi} + \frac{40}{3\pi} \cos 754t - \frac{40}{15\pi} \cos 1508t + \dots \quad (E9.4.1)$$

The frequency response of the LCR filter is

$$G(s) = \frac{R \parallel (1/sC)}{sL + [R \parallel (1/sC)]} = \frac{R/(sC)}{sL[R + 1/(sC)] + R/(sC)} = \frac{1}{s^2 LC + sL/R + 1}$$

$$G(j\omega) = \frac{1}{1 - \omega^2 LC + j\omega L/R} = \frac{1}{\sqrt{(1 - \omega^2 LC)^2 + (\omega L/R)^2}} \angle -\tan^{-1} \frac{\omega L/R}{1 - \omega^2 LC} \quad (E9.4.2)$$

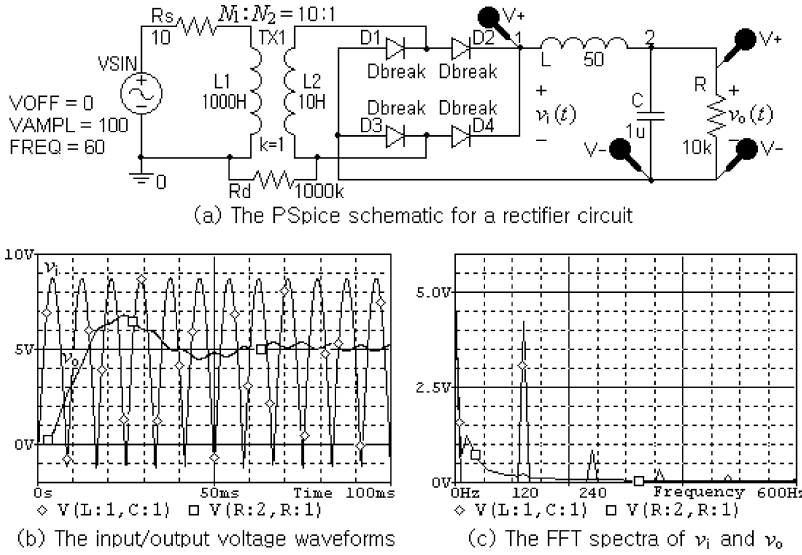


Figure 9.5 PSpice simulation for a rectifier (Example 9.4)

Thus the magnitudes of the DC component and the first harmonic in the output are

$$|V_o^{(0)}| = |G(j0)| |V_i^{(0)}| = \frac{1}{\sqrt{(1 - \omega^2 LC)^2 + (\omega L/R)^2}} \bigg|_{\substack{\omega=0 \\ R=10000}} \frac{20}{\pi} = \frac{20}{\pi} \quad (\text{E9.4.3})$$

$$|V_o^{(2)}| = |G(j2\omega_0)| |V_i^{(2)}| = \frac{1}{\sqrt{[1 - (2\omega_0)^2 LC]^2 + (2\omega_0 L/R)^2}} \bigg|_{\substack{2\omega_0=754 \\ R=10000}} \frac{40}{3\pi} \quad (\text{E9.4.4})$$

Consequently, the condition for the relative magnitude of the major harmonic to the DC component to be less than $r_{\max}$ can be written as

```
%cir09e04.m
clear, clf
global R w0
R= 10000; w0=2*pi*60;
CC= logspace(-4,-1,90); % set the range of C to 90 points on [10^-4, 10^-1]
TolX=1e-4; MaxIter=100; % tolerance on x and maximum iteration # for Newton
for ratio_max=0.05:0.05:0.1
    for m=1:length(CC)
        C= CC(m); L0=0.1; % Given the value of C and the starting value of L
        LL(m)= fsolve('f_cir09e04',L0,optimset('Display','off'),C,ratio_max);
        % LL(m)=Newtons('f_cir09e04',L0,TolX,MaxIter,C,ratio_max); % alternative
    end
    loglog(CC,LL), hold on
end
```

```
function y=f_cir09e04(L,C,r_max)
global R w0
w=2*w0; y= (1-w^2*L*C)^2 + (w*L/R)^2 - (2/3/r_max)^2; % Eq. (E9.4.5)
```

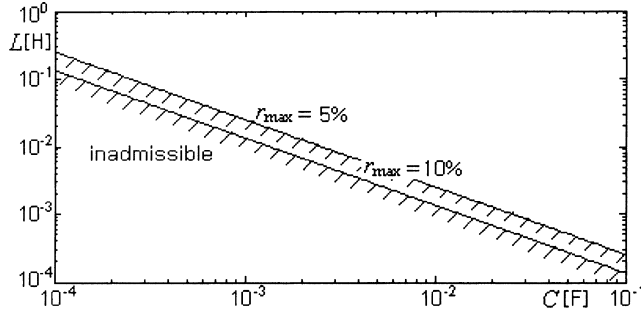



Figure 9.6 The admissible region for (C, L) to satisfy the design specification on the relative magnitude of the major harmonic to the DC component

$$\begin{aligned}
 & |\mathbf{V}_o^{(2)}| \leq r_{\max} |\mathbf{V}_o^{(0)}| \\
 & \frac{1}{\sqrt{[1 - (2\omega_0)^2 LC]^2 + (2\omega_0 L/R)^2}} \bigg|_{\substack{2\omega_0 = 754 \\ R = 10000}} \leq \frac{40}{3\pi} \leq \frac{20}{\pi} r_{\max} \\
 & [1 - (2\omega_0)^2 LC]^2 + \left(\frac{2\omega_0 L}{R}\right)^2 = (1 - 754^2 LC)^2 + \left(\frac{754L}{10000}\right)^2 \geq \left(\frac{2}{3r_{\max}}\right)^2 \quad (\text{E9.4.5})
 \end{aligned}$$

It seems to be good to plot the admissible region for (C, L) satisfying the design specification on filtering off harmonics with $r_{\max} = 5\%$ and 10% . This can be done by saving this equation in an M-file named `f_cir09e04.m` and running the MATLAB program `cir09e04.m` (see previous page) to get Figure 9.6, where the region below the graphs is not admissible in the sense that no positive values of (C, L) in the region can satisfy the above constraint (E9.4.5).

9.4 Fourier Series and Laplace Transform

Up until now in this chapter Fourier series have been used together with the phasor technique to analyze circuits excited by periodic wave sources. Where did the Laplace transform method go? Don't be concerned. It is always there, longing to be used by us. Not only to refresh our memory about the Laplace transform method, but also to compare it with the Fourier series-with-phasor technique, it will be applied to solve the RC circuit (in Figure 9.4(a)) that has been solved by using the Fourier series-with-phasor technique in Example 9.1.

Assuming that the rectangular wave voltage source is applied at $t = 0$, let us begin by writing it as the sum of many positive and negative scaled/delayed step functions and taking its Laplace transform:

$$v_i(t) = V_m[u_s(t) - 2u_s(t - D) + 2u_s(t - 2D) - \dots] \quad (9.6a)$$

$$V_i(s) = \lambda \{v_i(t)\} \stackrel{\text{Table A.1(3)}}{\underset{\text{Table A.2(2)}}{=}} V_m \left(\frac{1}{s} - \frac{2}{s} e^{-Ds} + \frac{2}{s} e^{-2Ds} - \frac{2}{s} e^{-3Ds} + \dots \right) \quad (9.6b)$$

The output of the RC circuit to this input can be obtained as

$$V_o(s) = G(s)V_i(s) \stackrel{(9.6)}{=} \frac{V_m/(RC)}{s[s + 1/(RC)]} (1 - 2e^{-Ds} + 2e^{-2Ds} - 2e^{-3Ds} + \dots) \quad (9.7a)$$

$$v_o(t) = \mathcal{L}^{-1}\{V_o(s)\}$$

$$\stackrel{\substack{\text{Table A.1(3),(5)} \\ \text{Table A.2(2)}}}{=} V_m[(1 - e^{-t/(RC)})u_s(t) - 2(1 - e^{-(t-D)/(RC)})u_s(t-D) \\ + 2(1 - e^{-(t-2D)/(RC)})u_s(t-2D) - \dots] \quad (9.7b)$$

In most cases, this is not the end of the work on the analysis of the RC circuit because the rectangular-wave voltage source is supposed to be applied much earlier than $t = 0$ so that the circuit can reach its steady state at $t = 0$. Referring to the sawtooth waveform of the output $v_o(t)$ in Figure 9.4(b), the steady state values of its upper/lower limits are found as follows:

$$v_o(2kD) \stackrel{(9.7b)}{=} V_m\{(1 - e^{-2kD/(RC)})u_s(2kD) - 2(1 - e^{-(2kD-D)/(RC)})u_s(2kD-D) \\ + \dots - 2(1 - e^{-(2kD-(2k-1)D)/(RC)})u_s[2kD - (2k-1)D] \\ + 2(1 - e^{-(2kD-2kD)/(RC)})u_s(2kD-2kD)\} \\ \stackrel{k \gg 1}{=} V_m\left(1 - \frac{2}{1 + e^{-D/(RC)}}\right) = V_m \frac{e^{-D/(RC)} - 1}{1 + e^{-D/(RC)}} = V_L \quad (9.8a)$$

$$v_o(2kD + D) \stackrel{(9.7b)}{=} V_m\{(1 - e^{-(2kD+D)/(RC)})u_s(2kD + D) - 2(1 - e^{-2kD/(RC)})u_s(2kD) \\ + \dots + 2(1 - e^{-(2kD+D-2kD)/(RC)})u_s(2kD + D - 2kD) \\ - 2(1 - e^{-(2kD+D-(2k+1)D)/(RC)})u_s[2kD + D - (2k+1)D]\} \\ \stackrel{k \gg 1}{=} V_m\left(-1 + \frac{2}{1 + e^{-D/(RC)}}\right) = V_m \frac{1 - e^{-D/(RC)}}{1 + e^{-D/(RC)}} = V_H = -V_L \quad (9.8b)$$

This can be supported by substituting $T_H = T_L = D$, $V_1 = -V_m$, $V_2 = V_m$, and $T = RC$ into Equation (P3.8.1), which can be found in Problem 3.8 of Chapter 3:

$$\begin{bmatrix} V_H \\ V_L \end{bmatrix} \stackrel{(P3.8.1)}{=} \frac{1}{1 - e^{-(T_H+T_L)/T}} \begin{bmatrix} (1 - e^{-T_H/T})V_2 + e^{-T_H/T}(1 - e^{-T_L/T})V_1 \\ (1 - e^{-T_L/T})V_1 + e^{-T_L/T}(1 - e^{-T_H/T})V_2 \end{bmatrix} \\ \stackrel{T_H=T_L=D}{=} \frac{1}{1 - e^{-2D/T}} \begin{bmatrix} (1 - e^{-D/T})V_m + e^{-D/T}(1 - e^{-D/T})(-V_m) \\ (1 - e^{-D/T})(-V_m) + e^{-D/T}(1 - e^{-D/T})V_m \end{bmatrix} \stackrel{T=RC}{=} \frac{V_m}{1 + e^{-D/(RC)}} \begin{bmatrix} 1 - e^{-D/(RC)} \\ -1 + e^{-D/(RC)} \end{bmatrix} \quad (9.9)$$

With V_L/V_H as the initial values and $V_m/-V_m$ as the final values for the rising/falling intervals, respectively, Equation (3.39) can be used to find the expression for the output $v_o(t)$ as

$$v_{o1}(t) \stackrel{(3.39)}{=} [v_{o1}(t_0) - v_{o1}(\infty)]e^{-(t-t_0)/T} + v_{o1}(\infty) \stackrel{v_{o1}(t_0)=V_L}{\stackrel{v_{o1}(\infty)=V_m}{=}} (V_L - V_m)e^{-(t-2kD)/T} + V_m$$

$$\stackrel{(9.9)}{\stackrel{T=RC}{=}} V_m \left(1 - \frac{2}{1 + e^{-D/(RC)}} e^{-(t-2kD)/(RC)} \right) \quad \text{for } 2kD \leq t < (2k+1)D \quad (9.10a)$$

$$v_{o2}(t) \stackrel{(3.39)}{=} [v_{o2}(t_0) - v_{o2}(\infty)]e^{-(t-t_0)/T} + v_{o1}(\infty) \stackrel{v_{o2}(t_0)=V_H}{\stackrel{v_{o2}(\infty)=-V_m}{=}} (V_H + V_m)e^{-[t-(2k+1)D]/T} - V_m$$

$$\stackrel{(9.9)}{\stackrel{T=RC}{=}} V_m \left(-1 + \frac{2}{1 + e^{-D/(RC)}} e^{-[t-(2k+1)D]/(RC)} \right) \quad \text{for } (2k+1)D \leq t < (2k+2)D \quad (9.10b)$$

At this point, it might be asked how this expression is related to the output equation (E9.3.4) that was obtained in Example 9.3:

$$v_o(t) \stackrel{(E9.3.4)}{=} \sum_{k=2m+1}^{\infty} \frac{4V_m}{k\pi\sqrt{1+(k\omega_0 RC)^2}} \sin(k\omega_0 t - \tan^{-1} k\omega_0 RC) \quad \text{with } \omega_0 = \frac{2\pi}{T} = \pi \quad (9.11)$$

```
%cir09f07.m to plot Fig. 9.7
clear, clf
T=2; w0=2*pi/T; D=1; Vm=pi; RC=0.3;
tt=[0:600]*T/200; % time interval of 3 periods
vit = ones(100,1)*(4*[1 -1 1 -1 1 -1]);
vit = [vit(:)'; vit(1)]; % input
ttD=[0:99]*D/100; % Laplace transform solution
volt= Vm*(1-2/(1+exp(-D/RC))*exp(-ttD/RC)); % Eq. (9.10a)
vo2t= Vm*(-1+2/(1+exp(-D/RC))*exp(-ttD/RC)); % Eq. (9.10b)
Laplace_transform_solution=[volt vo2t volt vo2t volt vo2t volt(1)];
MM= [1 2 5]; % the highest order of harmonic terms summed in (9.11)
for i=1:length(MM)
    if i==1, gs='k'; elseif i==2, gs='b'; else gs='r'; end
    vot= 0;
    for m=0:MM(i)
        k=2*m+1; kw0RC = k*w0*RC; % Fourier solution (9.11)
        vot = vot + 4/k/sqrt(1+kw0RC^2)*sin(k*w0*tt-atan(kw0RC));
    end
    subplot(410+i), plot(tt,vit), axis([tt(1) tt(end) -4.5 4.5])
    hold on, plot(tt,vot,gs)
end
subplot(414), plot(tt,vit), hold on
plot(tt,Laplace_transform_solution), axis([tt(1) tt(end) -4.5 4.5])
```

The Fourier series representation for Equations (9.10a) and (9.10b) can be found and compared with Equation (9.11) to confirm their equality (see Problem 9.3), which is very time-consuming. Instead, they are plotted for graphic comparison by running the program `cir09f07.m` (see above). It yields Figure 9.7, in which (a), (b), and (c) are the plots of Equation (9.11) approximated by a truncated sum for $m = 0, 1$, one for $m = 0, 1, 2$, and one for $m = 0, 1, \dots, 5$, respectively. It can be seen that the more terms are summed in the Fourier series representation (9.11), the closer its plot becomes to the Laplace transform solution, which is depicted in Figure 9.7(d).

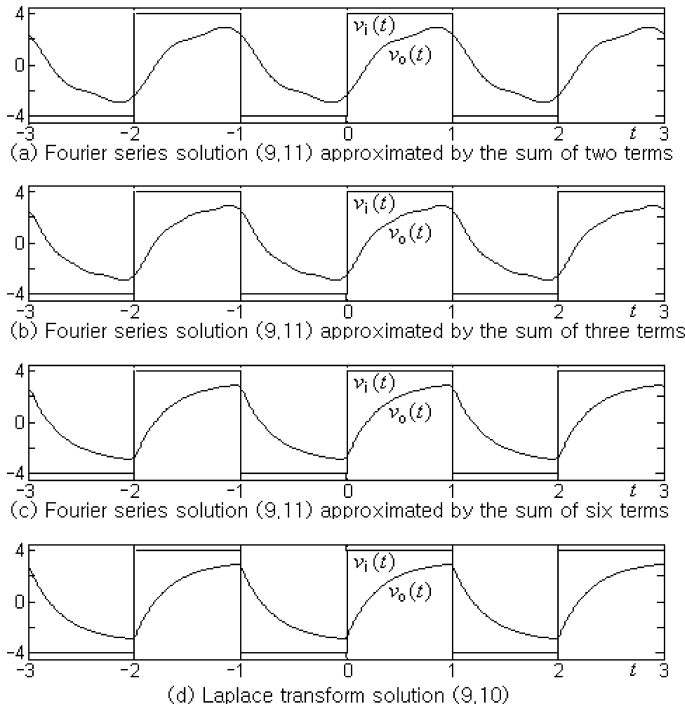


Figure 9.7 Fourier series solution and Laplace transform solution

[Remark 9.1] Fourier Series-with-Phasor Method versus Laplace Transform Method

For the Fourier series-with-phasor method, the strength is that it presents the frequency characteristics of the input and output(s). The weakness is that it does not produce a clear description of the output waveform, which is usually approximated by a finite sum of DC, fundamental, and harmonic terms. For the Laplace transform method, the strength is that it produces a clear description of the output waveform. The weakness is that it presents no information about the frequency characteristics of the input and output(s).

(Example 9.5) Fourier Series and Laplace Transform for a Narrow-Bandwidth Circuit

Figure 9.8 shows an RLC circuit with $R \gg (1/2)\sqrt{L/C}$ (for narrow bandwidth), which is excited by a square-wave voltage source with the amplitude $0 \sim V_m$, the period T , and the duration $D = T/2$.

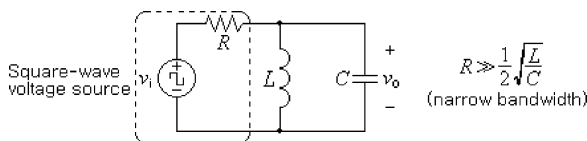


Figure 9.8 An RLC circuit excited by a square-wave source

Suppose this circuit is tuned so that its resonant frequency $\omega_r = 1/\sqrt{LC}$ is in accord with the fundamental frequency $\omega_0 = 2\pi/T$. To summarize, suppose that

$$(\text{fundamental frequency of input})\omega_0 = \frac{2\pi}{T} = \frac{1}{\sqrt{LC}} = \omega_r (\text{resonant frequency of circuit}) \quad (\text{E9.5.1})$$

where

$$R \gg \frac{1}{2}\sqrt{\frac{L}{C}}; \quad 2RC \gg \sqrt{LC}; \quad \frac{1}{2RC} \ll \sqrt{\frac{1}{LC}}; \quad \frac{1}{LC} \gg \left(\frac{1}{2RC}\right)^2 \quad (\text{E9.5.2})$$

Noting that a square wave is nothing but the sum of many scaled/delayed step functions (see Equation (9.6a)), work is first done on the step response, i.e. the output of the circuit to a step input $V_m u_s(t)$ with constant height V_m . Since the transfer function of this circuit is

$$G(s) = \frac{sL \parallel (1/sC)}{R + [sL \parallel (1/sC)]} = \frac{\frac{sL/(sC)}{sL + 1/(sC)}}{R + \frac{sL/(sC)}{sL + 1/(sC)}} = \frac{[1/(RC)]s}{s^2 + [1/(RC)]s + 1/(LC)} \quad (\text{E9.5.3})$$

the output to a step input $V_m u_s(t)$ with the Laplace transform of $\mathcal{L}\{V_m u_s(t)\} = V_m/s$ can be obtained as

$$\begin{aligned} V_{o1}(s) &= G(s) \frac{V_m}{s} = \frac{[1/(RC)]s}{s^2 + [1/(RC)]s + 1/(LC)} \frac{V_m}{s} = \frac{V_m/(RC)}{s^2 + [1/(RC)]s + 1/(LC)} \\ &= \frac{V_m}{\omega_d RC} \frac{\omega_d}{(s + \sigma)^2 + \omega_d^2} \\ &\quad \left(\text{with } \sigma = \frac{1}{2RC} \quad \text{and} \quad \omega_d = \sqrt{\frac{1}{LC} - \left(\frac{1}{2RC}\right)^2} \stackrel{(\text{E9.5.2})}{\simeq} \sqrt{\frac{1}{LC}} = \omega_r \right) \\ &\simeq \frac{V_m}{(1/\sqrt{LC})RC} \frac{\omega_r}{(s + \sigma)^2 + \omega_r^2} = \frac{V_m}{R} \sqrt{\frac{L}{C}} \frac{\omega_r}{(s + \sigma)^2 + \omega_r^2} \end{aligned} \quad (\text{E9.5.4a})$$

$$v_{o1}(t) \simeq \frac{V_m}{R} \sqrt{\frac{L}{C}} e^{-t/(2RC)} \sin(\omega_r t) u_s(t) \quad (\text{E9.5.4b})$$

Now the output to the square-wave input is going to be found twice, once by using the Laplace transform and once by using the Fourier series.

(a) Laplace Transform Approach

In the same manner as in Equation (9.6a), the square wave input with the amplitude $0 \sim V_m$, the period T , and the duration $D = T/2$ can be expressed as the sum of many scaled/time-shifted step functions as

$$v_i(t) = V_m \left[u_s(t) - u_s\left(t - \frac{T}{2}\right) + u_s(t - T) - \dots \right] \quad (\text{E9.5.5})$$

Thus, by the superposition principle, the steady state output $v_o(t)$ for $0 \leq t < T/2$ will be the sum of the responses to many scaled/time-shifted step inputs, i.e. $V_m u_s(t)$ (applied from $t = 0$), $-V_m u_s(t + T/2)$

(applied from $t = -T/2$), $+V_m u_s(t+T)$ (applied from $t = -T$), and so on:

$$v_o(t) = v_{o1}(t) - v_{o1}\left(t + \frac{T}{2}\right) + v_{o1}(t+T) - \dots \quad (\text{E9.5.6})$$

$$\stackrel{(\text{E9.5.4b})}{\simeq} \frac{V_m}{R} \sqrt{\frac{L}{C}} \left\{ e^{-t/(2RC)} \sin(\omega_r t) - e^{-(t+T/2)/(2RC)} \sin\left[\omega_r\left(t + \frac{T}{2}\right)\right] \right. \\ \left. + e^{-(t+T)/(2RC)} \sin[\omega_r(t+T)] - \dots \right\}$$

$$\text{with } \omega_r = \frac{2\pi}{T} \text{ and } \sin\left[\omega_r\left(t + \frac{T}{2}\right)\right] = \sin(\omega_r t + \pi) = -\sin(\omega_r t)$$

$$\stackrel{(\text{E9.5.1})}{=} \frac{V_m}{R} \sqrt{\frac{L}{C}} (1 + e^{-T/(4RC)} + e^{-2T/(4RC)} + \dots) e^{-t/(2RC)} \sin\left(\frac{2\pi}{T} t\right)$$

$$\stackrel{(\text{F.23})}{=} \frac{V_m}{R} \sqrt{\frac{L}{C}} \frac{1}{1 - e^{-T/(4RC)}} e^{-t/(2RC)} \sin(\omega_0 t) \quad (\text{E9.5.7})$$

$$\text{for } 0 \leq t < \frac{T}{2} \stackrel{(\text{E9.5.1})}{=} \pi \sqrt{LC} \stackrel{(\text{E9.5.2})}{\ll} 2RC; \quad \frac{T}{4RC} \ll 1$$

$$\stackrel{(\text{F.25})}{\simeq} \frac{V_m}{R} \sqrt{\frac{L}{C}} \frac{1}{1 - [T/(4RC)]} e^{-t/(2RC)} \sin(\omega_0 t) = \frac{4V_m}{T} \sqrt{LC} e^{-t/(2RC)} \sin(\omega_0 t)$$

$$v_o(t) \simeq \frac{2V_m}{\pi} \sin(\omega_0 t) \quad \text{for } 0 \leq t < \frac{T}{2} \quad (\text{E9.5.8})$$

Likewise for all successive intervals of length $T/2$,

$$v_o(t) \simeq -\frac{2V_m}{\pi} \sin\left[\omega_0\left(t - \frac{T}{2}\right)\right] = \frac{2V_m}{\pi} \sin(\omega_0 t) \quad \text{for } \frac{T}{2} \leq t < T$$

$$v_o(t) \simeq \frac{2V_m}{\pi} \sin[\omega_0(t-T)] = \frac{2V_m}{\pi} \sin(\omega_0 t) \quad \text{for } T \leq t < \frac{3T}{2}$$

.....

$$v_o(t) \simeq \frac{2V_m}{\pi} \sin(\omega_0 t) \quad \forall t \quad (\text{E9.5.9})$$

This implies that the output of the system (with a narrow bandwidth) to a square-wave input of the fundamental frequency $\omega_0 = 2\pi/T$ in tune with the resonant frequency $\omega_r = 1/\sqrt{LC}$ is just like a sinusoid of the resonant frequency.

(b) Fourier Series Approach

The first thing to do in the Fourier series approach is to find the Fourier series representation of the input function. Noting that the square wave of the input voltage source is a version of the standard square wave (depicted in Figure 9.1(a)) delayed by $T/4$, we can substitute $(t - T/4)$ for t into Equation (E9.1.5) with $A = V_m$ and $D = T/2$ to write the Fourier series representation of the input as

$$v_i(t) \stackrel{(\text{E9.1.5})}{=} \frac{AD}{T} + \sum_{k=1}^{\infty} \frac{2AD}{T} \frac{\sin(k\pi D/T)}{k\pi D/T} \cos k\omega_0 t = \frac{V_m}{2} + \sum_{k=1}^{\infty} V_m \frac{\sin(k\pi/2)}{k\pi/2} \cos k\omega_0 \left(t - \frac{T}{4}\right) \\ v_i(t) = \frac{V_m}{2} + \sum_{k=\text{odd}}^{\infty} \frac{2V_m}{k\pi} \sin k\omega_0 t \quad (\text{E9.5.10})$$

The frequency response

$$\begin{aligned}
 G(s) &\stackrel{(E9.5.3)}{=} \frac{s/(RC)}{s^2 + s/(RC) + 1/(LC)} = \frac{\omega_b s}{s^2 + \omega_b s + \omega_r^2} \\
 G(j\omega) &= \frac{j\omega\omega_b}{(\omega_r^2 - \omega^2) + j\omega\omega_b} \quad \text{with} \quad \omega_b = \frac{1}{RC} \ll \frac{1}{\sqrt{LC}} = \omega_r = \omega_0 = \frac{2\pi}{T} \\
 &= \frac{|\omega\omega_b|}{\sqrt{(\omega_r^2 - \omega^2)^2 + (\omega\omega_b)^2}} \angle \left(\frac{\pi}{2} - \tan^{-1} \frac{\omega\omega_b}{\omega_r^2 - \omega^2} \right) \quad (E9.5.11)
 \end{aligned}$$

implies that the circuit will function as a bandpass filter with a narrow bandwidth ω_b and so is likely to pass only the resonant frequency component with the unity frequency response $G(j\omega_r) = 1$ (see Section 8.3.1). Besides, the resonant frequency ω_r conforms with the fundamental frequency ω_0 . That is why a guess could be made that the output of the circuit is mainly the resonant frequency component contained in the input $v_i(t)$ (described by Equation (E9.5.10)):

$$v_o(t) \simeq \frac{2V_m}{\pi} \sin(\omega_0 t) \quad \text{for } k = 1 \quad (E9.5.12)$$

This agrees with Equation (E9.5.9), which has been obtained with the Laplace transform approach.

Note. In this example, it appears that the Fourier series approach produces the same solution as the Laplace transform approach, but with fewer steps. However, this statement can never be generalized. Instead it may be said that the Fourier series approach is useful for analyzing the frequency characteristic of signals, while the Laplace transform method is suitable for obtaining the analytical description of signals.

9.5 RMS Value and Power of a Nonsinusoidal Periodic Signal

The rms (root mean square) value of a sinusoidal voltage/current can easily be obtained by dividing its maximum value by $\sqrt{2}$ (Equations (6.4a)/(6.4b)). It is also easy to get the active/reactive power (Equations (6.26)/(6.27)) and the power factor (Equation (6.32)) if the AC voltage and current are purely sinusoidal. How can the rms value, the active/reactive powers, and the power factor be obtained for the nonsinusoidal periodic voltage and current? In this section the formulas will be derived in order to get those values from their Fourier series representations.

9.5.1 RMS Value and Distortion Factor of a Nonsinusoidal Periodic Signal

Let a nonsinusoidal periodic signal have the magnitude-and-phase or sine-and-phase form of Fourier series representation as

$$x(t) \stackrel{(9.2a)}{=} X_0 + \sum_{k=1}^{\infty} X_{km} \cos(k\omega_0 t + \phi_k) = X_0 + \sum_{k=1}^{\infty} X_{km} \sin(k\omega_0 t + \phi'_k) \quad (9.12)$$

Then its *rms* (root mean square) or *effective value* can be computed using Equation (6.3) as

$$\begin{aligned}
 X_{\text{rms}} &\stackrel{(6.3)}{=} \sqrt{\frac{1}{T} \int_T x^2(t) dt} = \sqrt{\frac{1}{T} \int_T \left[X_0^2 + \sum_{k=1}^{\infty} X_{km}^2 \cos^2(k\omega_0 t + \phi_k) \right] dt} \\
 &= \sqrt{X_0^2 + \sum_{k=1}^{\infty} \frac{1}{T} \int_T X_{km}^2 \cos^2(k\omega_0 t + \phi_k) dt} \stackrel{(F.15)}{=} \sqrt{X_0^2 + \sum_{k=1}^{\infty} X_k^2} \quad (9.13)
 \end{aligned}$$

which is the square root of the sum of the squared DC (average) value (X_0^2) and the squared rms values ($X_k^2 = X_{km}^2/2$) of every (fundamental and harmonic) frequency component contained in the signal. In this derivation, use has been made of the fact that the integral of a product of two sinusoids of different frequencies over their common period is zero:

$$\begin{aligned} & \int_T \cos(k\omega_0 t + \phi_k) \cos(m\omega_0 t + \phi_m) dt \\ & \stackrel{(F.11)}{=} \frac{1}{2} \int_T \{ \cos[(k+m)\omega_0 t + \phi_k + \phi_m] + \cos[(k-m)\omega_0 t + \phi_k - \phi_m] \} dt = 0 \end{aligned} \quad (9.14)$$

As a measure of how far a periodic signal is from its sinusoid of fundamental frequency, the *total harmonic distortion* (THD), also referred to as the *distortion factor*, is defined as the ratio of the rms value of harmonic components to that of fundamental frequency component:

$$\text{THD} = \frac{\sqrt{\sum_{k=2}^{\infty} X_k^2}}{X_1} = \frac{\sqrt{\sum_{k=2}^{\infty} X_{km}^2}}{X_{1m}} \quad (9.15)$$

9.5.2 Power and Power Factor of a Nonsinusoidal Periodic Signal

Suppose the terminal voltage and current of an element or a system are periodic with the same fundamental frequency ω_0 and each of them has the magnitude-and-phase form of Fourier series representation as

$$v(t) = V_0 + \sum_{k=1}^{\infty} V_{km} \cos(k\omega_0 t + \phi_k) \quad (9.16a)$$

$$i(t) = I_0 + \sum_{k=1}^{\infty} I_{km} \cos(k\omega_0 t + \psi_k) \quad (9.16b)$$

Then the *active* or *average power* of the element or system can be obtained as

$$\begin{aligned} P &= \frac{1}{T} \int_T p(t) dt \stackrel{(1.4a)}{=} \frac{1}{T} \int_T v(t)i(t) dt = \frac{1}{T} \int_T \left[V_0 I_0 + \sum_{k=1}^{\infty} V_{km} \cos(k\omega_0 t + \phi_k) I_{km} \cos(k\omega_0 t + \psi_k) \right] dt \\ & \stackrel{(F.11)}{=} V_0 I_0 + \sum_{k=1}^{\infty} \frac{V_{km} I_{km}}{2} \cos(\phi_k - \psi_k) \stackrel{(6.26)}{=} P_0 + \sum_{k=1}^{\infty} P_k \end{aligned} \quad (9.17)$$

This is nothing but the sum of all the active or average powers of every frequency component including the DC power. In this derivation, use has been made of the fact that the integral of a product of two sinusoids of different frequencies over their common period is zero, as shown in Equation (9.14).

The *apparent power*, defined as the product of the rms voltage and rms current in Equation (6.29), is

$$P_{\text{app}} = |S| \stackrel{(6.29)}{=} VI = \sqrt{\sum_{k=0}^{\infty} V_k^2} \sqrt{\sum_{k=0}^{\infty} I_k^2} \quad (9.18)$$

and the overall *power factor* is

$$\text{PF} \stackrel{(6.32)}{=} \frac{P}{P_{\text{app}}} \quad (9.19)$$

Problems

9.1 Fourier Series Representations of a Multistep Function and a Sawtooth Function

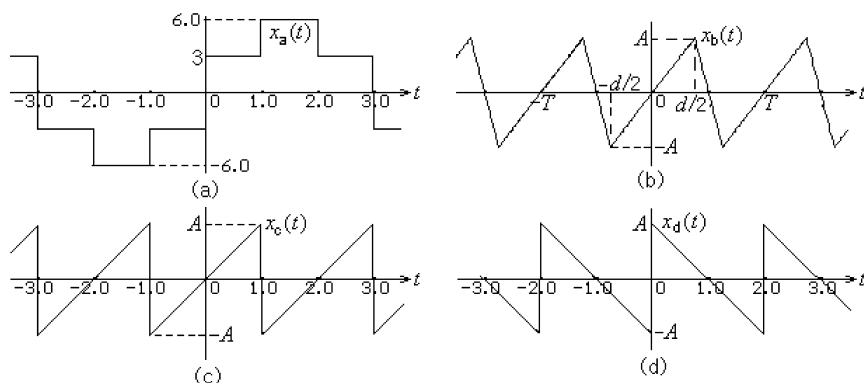


Figure P9.1

- (a) Noting that the multistep function in Figure P9.1(a) can be regarded as the sum of three scaled/time-shifted square waves with a common period of 6s and different durations

$$x_a(t) = 6 \tilde{r}_{3/6}(t - 1.5) - 3 + 3 \tilde{r}_{1/6}(t - 1.5) - 3 \tilde{r}_{1/6}(t - 4.5) \quad (\text{P9.1.1})$$

use Equation (E9.1.5) to find the Fourier series representation and then run the following MATLAB program `cir09p01a.m` to plot it to see if it becomes close to $x_a(t)$ as the number of terms in the summation increases, say, from 5 to 10.

(Answer)

$$x_a(t) \stackrel{(\text{E9.1.5})}{=} 12 \sum_{m=1}^{\infty} (-1)^{m+1} \frac{\sin[(2m-1)\pi/2] + \sin[(2m-1)\pi/6]}{(2m-1)\pi} \sin[(2m-1)\omega_0 t] \quad (\text{P9.1.2})$$

```
%cir09p01a.m
% to plot Fig.P9.1
clear, clf
T= 6; w0= 2*pi/T; % Period, Fundamental frequency
Ts= 0.01; % Sampling interval
x1= ones(1,1/Ts);
xt=[3*x1 -3*x1 -6*x1 -3*x1 0 3*x1 6*x1 3*x1 -3*x1]; % the original x(t)
tt=[-4: Ts: 4]; % Whole time interval
N= 10; % The number of terms in the following summation
xht= 0; sgn= 1;
for m=1:N % Restore from the Fourier series representation up to N terms
    kpi= (2*m-1)*pi; kw0t= (2*m-1)*w0*tt;
    xht= xht + sgn*12/kpi* (sin(kpi/2) + sin(kpi/6))*sin(kw0t); % (P9.1.2)
    sgn= -sgn;
end
subplot(221), plot(tt,xt, 'k', tt,xht, 'r')
```

(b) Noting that the sawtooth function in Figure P9.1(b) can be expressed as

$$x_b(t) = \begin{cases} \frac{A}{d/2}(t - kT) & \text{for } kT - \frac{d}{2} \leq t < kT + \frac{d}{2} \\ \frac{A}{d/2}[T - 2(t - kT)] & \text{for } kT + \frac{d}{2} \leq t < (k+1)T - \frac{d}{2} \end{cases} \quad (\text{P9.1.3})$$

use Equations (9.4a) and (9.4b) to find the complex exponential Fourier series representation with the Fourier coefficients $c_k = (T/2)(a_k - jb_k)$ and then plot it to see if it becomes close to $x_b(t)$ as the number of terms in the summation increases, say, from 3 to 6.

(Answer)

$$x_b(t) = \frac{8A}{d(T-d)} \sum_{k=1}^{\infty} \frac{\sin(k\omega_0 d/2)}{(k\omega_0)^2} \sin k\omega_0 t \quad (\text{P9.1.4})$$

(c) Make use of Equation (P9.1.4) to get the Fourier series representations of the waveforms in Figures P9.1(c) and (d) and plot them to check if they become close to $x_c(t)$ and $x_d(t)$, respectively, as the number of terms in the summation increases, say, from 5 to 10.

9.2 DC and Fundamental Frequency Components of a Periodic Short-Duration Noise

Consider a periodic noise $n(t)$ with period T and short duration $\delta \ll T$. Show that the magnitude of its fundamental frequency component is approximately two times that of its DC component. For convenience, assume that the noise is centered at $t = 0$.

9.3 Laplace Transform Solution and Fourier Series Solution of an RC Circuit Excited by a Square Wave

For the output of an RC circuit to the square-wave input, there is the Fourier series solution (E9.3.4) in Example 9.3 (Section 9.3) or Equation (9.11) and the Laplace transform solution (9.10a) and (9.10b) in Section 9.4:

$$v_{o,L}(t) \stackrel{(9.10a),(9.10b)}{=} \begin{cases} V_m \left(1 - \frac{2}{1+e^{-D/(RC)}} e^{-(t-2kD)/(RC)} \right) & \text{for } 2kD \leq t < (2k+1)D \\ V_m \left(-1 + \frac{2}{1+e^{-D/(RC)}} e^{-[t-(2k+1)D]/(RC)} \right) & \text{for } (2k+1)D \leq t < (2k+2)D \end{cases} \quad (\text{P9.3.1})$$

$$v_{o,F}(t) \stackrel{(9.11)}{=} \sum_{k=-\infty}^{\infty} \frac{4V_m}{k\pi\sqrt{1+(k\omega_0 RC)^2}} \sin(k\omega_0 t - \tan^{-1} k\omega_0 RC) \quad \text{with } \omega_0 = \frac{2\pi}{T} = \pi \quad (\text{P9.3.2})$$

To show that these two expressions are equivalent, take the following steps to find the Fourier series representation of Equation (P9.3.1) and verify that it conforms with Equation (P9.3.2).

(a) Check if the following equations are correct:

$$\begin{aligned} \int_0^{T/2} e^{-t/(RC)} e^{-jk\omega_0 t} dt &= \frac{1}{-jk\omega_0 - 1/(RC)} e^{-[jk\omega_0 + 1/(RC)]t} \Big|_0^{T/2} \\ &= \frac{RC}{-jk\omega_0 - 1/(RC)} (e^{-jk\pi - T/(2RC)} - 1) \\ &= \frac{RC(1 - jk\omega_0 RC)}{1 + (k\omega_0 RC)^2} [1 - (-1)^k e^{-T/(2RC)}] \\ &\stackrel{\text{for } k \text{ odd}}{=} \frac{RC(1 - jk\omega_0 RC)}{1 + (k\omega_0 RC)^2} (1 + e^{-T/(2RC)}) \end{aligned} \quad (\text{P9.3.3})$$

- (b) Noting the half-wave symmetry of the output waveform depicted in Figure 9.7, we see from Table 9.1 that the Fourier coefficients can be obtained by doubling the integration for a half-period but only for odd k values. Check if the exponential Fourier series coefficients are as follows:

$$\begin{aligned}
 c_k &\stackrel{(9.4b) \text{ for half-wave symmetry}}{=} 2 \int_0^{T/2} v_{o,L}(t) e^{-jk\omega_0 t} dt \quad \text{for } k \text{ odd, } T/2 = D \\
 &\stackrel{(P9.3.1a)}{=} 2 \int_0^D V_m \left(1 - \frac{2}{1 + e^{-D/(RC)}} e^{-t/(RC)} \right) e^{-jk\omega_0 t} dt \\
 &\stackrel{(P9.3.3)}{=} \frac{2V_m}{-jk\omega_0} e^{-jk\omega_0 t} \Big|_0^D - 4V_m \frac{RC(1 - jk\omega_0 RC)}{1 + (k\omega_0 RC)^2} \\
 &\stackrel{(F.33)}{=} \frac{2V_m}{-jk\omega_0} (e^{-jk\pi} - 1) - 4V_m \frac{RC(1 - jk\omega_0 RC)}{1 + (k\omega_0 RC)^2} \\
 &= -\frac{4V_m RC}{1 + (k\omega_0 RC)^2} - j\frac{4V_m}{k\omega_0} + j\frac{4V_m k\omega_0 (RC)^2}{1 + (k\omega_0 RC)^2} \\
 &= -\frac{4V_m RC}{1 + (k\omega_0 RC)^2} - j\frac{4V_m}{k\omega_0 [1 + (k\omega_0 RC)^2]} \\
 &= \frac{T}{2} (a_k - jb_k) = D(a_k - jb_k) \tag{P9.3.4}
 \end{aligned}$$

- (c) Based on (P9.3.4), find the Fourier series representation of the waveform $v_{o,L}(t)$ in Figure 9.7.
- (d) Referring to the MATLAB program `cir09e02.m` for Example 9.2, make use of the MATLAB routine `CtFS_trigonometric( )` to find the Fourier series coefficients for $k = 1, 3, 5, 7$ and check if they agree with the corresponding ones (P9.3.4) obtained in an analytical way. Plot the Fourier spectrum and the Fourier series representation (P9.3.2) together with the original waveform $v_{o,L}(t)$ to see if they become close as the number of terms in the summation increases, say, from 2 to 4.

Hint. It would be a good idea to store the function defining Equation (P9.3.1) as an M-file named, say, `RC_output.m`, complete the following program `cir09p03.m`, and run it.

```

%cir09p03.m
global T D Vm RC
T=2; w0=2*pi/T; D=1; Vm=pi; RC=0.3;
N=7; kk=0:N; % Frequency indices to analyze using Fourier analysis
tt=[-400:400]*T/200; % Time interval of 4 periods
x = 'RC_output'; % the name of the M-file function defining Eq. (P9.3.1)
[a0,a,b] = ??????????????????????????????; % get Fourier series coefficients
k=1:2:N; kw0= k*w0; kw0RC21= 1+(kw0*RC).^2;
a_t(k)=-4*Vm*RC/D./kw0RC21; b_t(k)=????????????????????; % Eq. (P9.3.4)
a0_ak_bk=[a0 a b; 0 a_t b_t] % the numerical/analytical FS coefficients
xht = a0; % starting from the DC component
for k=1:N % Fourier series representation (9.1a)
    xht = xht + a(k)*cos(k*w0*tt) + b(k)*????????????????;
end
subplot(221), xt = feval(x,tt); % original signal
plot(tt,xt,'k-', tt,xht,'b:') % plot original/reconstructed signals
d1 = [a0 ??????????????????]; % magnitude of Fourier spectrum (9.3b)
subplot(222), stem(kk, d1) % plot the magnitude spectrum

```

```

function y=RC_output(t)
global T D Vm RC
t= mod(t,T);
tmp= 2/(1+exp(-D/RC)); % Eq. (P9.3.1)
y= (t<=D)*Vm.*(1-tmp*exp(-t/RC)) + (t>D)*Vm.*(-1+tmp*exp(-(t-D)/RC));

```

9.4 Performance of a Lowpass Filter on the Suppression of Harmonics

Consider Example 9.3, where we determined the minimum value of RC required to keep the relative magnitude of the third harmonic ($3\omega_0$) component to the fundamental frequency (ω_0) component in the output not larger than $1/8$.

- Is it possible to make the ratio as low as $1/10$? If not, what is the lower bound of the ratio that can be achieved by adjusting the value of RC ?
- If the circuit in Figure P9.4 is used in place of the RC circuit, what will be the lower bound of the ratio that can be achieved by adjusting the values of R and C ?
- Let the circuit with $C = 5\mu\text{F}$ in Figure P9.4 be used in place of the RC circuit and find the minimum value of R such that the ratio is not larger than $1/15$. Plot the semi-log curve of the magnitude ratio of the frequency response (with $C = 5\mu\text{F}$) $G(3\omega_0)$ to $G(\omega_0)$ for $R = 10^4$ – 10^5 .

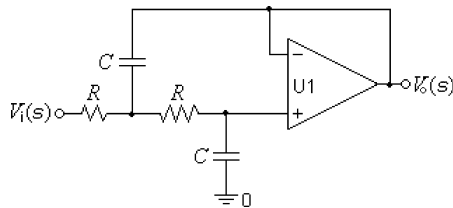


Figure P9.4

Note. Since the relative magnitude of the third harmonic ($3\omega_0$) component to the fundamental frequency (ω_0) one in the square-wave input is $1/3$, find the condition for

$$\frac{|G(3\omega_0)|}{|G(\omega_0)|} \leq \frac{1}{5} \quad (\text{P9.4.1})$$

9.5 Performance of a Lowpass Filter on the Suppression of Harmonics

Consider Example 9.4 in which the admissible regions of (C, L) were found to keep the relative magnitude of the second harmonic ($2\omega_0$) component to the DC one in the output not larger than 5 % or 10 %, as depicted in Figure 9.6.

- Are the values of $C = 3\text{ mF}$ and $L = 5\text{ mH}$ in the 5 % admissible region? Perform the Fourier analysis to check if the relative magnitude of the second harmonic ($2\omega_0$) component to the DC one in the output is less than 5 % or not.
- Do the same job for the values of $C = 3\text{ mF}$ and $L = 10\text{ mH}$ as in (a).

Hint. Complete the program `cir09p05.m` and save it together with the rectified sinusoidal input function `f_cir09p05( )` (presented over page) in the M-files named `cir09p05.m` and `f_cir09p05.m`, respectively, and run it by typing `cir09p05` (without the quotation marks) into the MATLAB command window.

```
%cir09p05.m: to solve Problem 9.5
% Perform Fourier analysis for a LPF with a rectified sine wave input
clear, clf
global f0 Vm
f0=60; T=1/f0; Vm= 10; % Frequency, Period, Amplitude
N=5; kk=0:N; % Frequency indices to analyze using Fourier analysis
x= '?????????'; % Rectified sine wave input function defined in an M-file
R=1e4; L=????; C=????; % admissible for  $|V_o(2\omega_0)|/|V_o(0)| < 5\%$ 
n=1, d=[L*C ??? 1] %Numerator/Denominator of G(s) Eq. (E9.4.2)
[Y,X,THDy,THDx]= Fourier_analysis(n,d,x,T,N);
Ratio_of_2nd_harmonic_to_DC= abs(Y(3))/abs(Y(1))

function y=f_cir09p05(t)
% defines a rectified sine wave with frequency f0 and amplitude Vm
global f0 Vm
y= abs(Vm*sin(2*pi*f0*t));
```

9.6 Fourier Series Solution of an Active Second-Order OP Amp Circuit Excited by a Triangular Wave

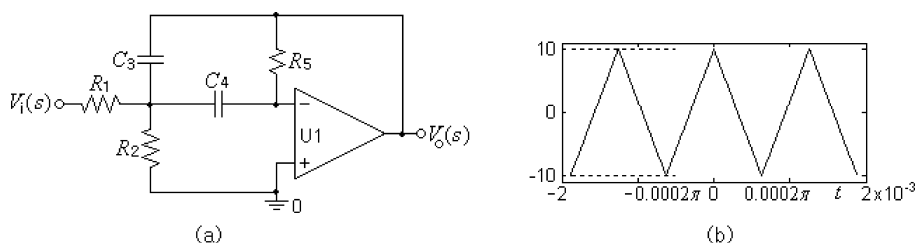


Figure P9.6

Consider the active second-order OP Amp circuit in Figure P9.6(a), which is excited by a triangular-wave voltage source depicted in Figure P9.6(b). Find the magnitudes of the leading three frequency components (up to three significant digits) of the input v_i and output v_o in two ways.

- (a) Use the MATLAB function `Fourier_analysis()` that was introduced in Section 9.3.
Hint. Complete the following program and save it together with `f_cir09p06()` (presented below) in the M-files named `cir09p06.m` and `f_cir09p06.m`, respectively, and run it.

```
%cir09p06.m: to solve Problem 9.6
% Perform Fourier analysis for a BPF with a triangular wave input
clear, clf
format short e
global T D Vm
T= 2*pi/5000, D= T/2, Vm= 10; % Period, Duration, Amplitude
N=5; kk=0:N; % Frequency indices to analyze using Fourier analysis
x= '?????????'; % Triangular wave input function defined in an M-file
R1=1e3; R2=20; C3=1e-6; C4=1e-6; R5=???;
n= -[1/R1/C3 ?] % Numerator of transfer function G(s) Eq. (8.42)
d=[(C3+C4)/R5/C3/C4 ?????????????????????] % Denominator of Eq. (8.42)
[Y,X,THDy,THDx]= ?????????????????? (n,d,x,T,N);
```

```
Magnitude_and_phase_of_input_and_output_spectrum=...
[kk; abs(X); phase(X)*180/pi; abs(Y); angle(Y)*180/pi] .'
```

```
function y=f_cir09p06(t)
% defines a triangular wave with period T, duration D, and amplitude Vm
global T D Vm
t=mod(t,T);
y=Vm* ((t<=D) .* (1-(2/D)*t) + (t>D) .* (-1+(2/D)*(t-D)));
```

- (b) Use the PSpice simulation, where the parameters of the periodic voltage source VPULS can be set as follows:

$$V_1 = -10, \quad V_2 = +10, \quad TD = 0, \quad TR = 0.000628, \\ TF = 0.000628, \quad PW = 0.000001, \quad PER = 0.001257$$

In the Simulation Settings dialog box, you can click the Output File Options button to open the Transient Output File Options dialog box and set the parameters as

$$\text{Center frequency} = 796 \text{ Hz}, \quad \text{Number of harmonics} = 5 \\ \text{Output variables} = V(U1 : OUT) \quad V(R1 : 1)$$

Note. Period = $0.0004\pi = 0.001257 \text{ s}$ and Frequency = $1/\text{period} = 1/0.0002\pi = 796 \text{ Hz}$.

10

Two-Port Networks

The concept of two-port networks helps us to have a general and systematic viewpoint for analyzing and designing complicated networks. The input–output relationship of a two-port network can be described in six different ways via the sets of z , y , h , g , a , and b -parameters. We will derive some formulas for converting a set of parameters into another one and implement them in a MATLAB routine. We will see how easy it is to obtain the overall set of parameters for various connections of two-port networks from their sets of two-port parameters. Once a set of parameters for a two-port network is determined, the voltage/current gain and the input/output impedances in terms of the two-port parameters can be found by using the formulas that are also implemented in a MATLAB routine. This chapter ends with the circuit models with a given parameter.

10.1 Definitions of Two-Port Parameters

You might feel like calling any pair of terminals in a circuit a port if only an external element can be connected between the two terminals (nodes). However, in order for a pair of terminals to be a *port*, it should satisfy the *port condition* or *current requirement* that the current flowing into the circuit through one terminal must equal the current flowing out of the circuit through the other terminal. In fact, for one-port (two-terminal) networks, there is no need to bother about the port condition since KCL does not allow the port condition to be violated for any pair of terminals. This is no longer true for two-port networks.

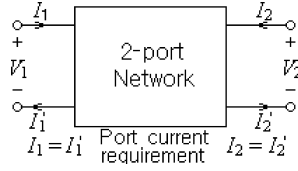
A circuit is called a *two-port network* if it has two ports available for external connection, each of which is supposed to satisfy the port condition. A two-port network is often depicted as in Figure 10.1, where the left/right port is somewhat arbitrarily referred to as the *input(source)/output(load) port* or the *primary/secondary port*, respectively, and the voltage/current are defined for each port.

The six sets of parameters characterizing the two-port network are defined via the following equations:

Impedance (Impedance/Admittance) Parameters

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \quad \text{with} \quad \begin{aligned} z_{11} &= \left. \frac{V_1}{I_1} \right|_{I_2=0}, & z_{12} &= \left. \frac{V_1}{I_2} \right|_{I_1=0} \\ z_{21} &= \left. \frac{V_2}{I_1} \right|_{I_2=0}, & z_{22} &= \left. \frac{V_2}{I_2} \right|_{I_1=0} \end{aligned} \quad (10.1)$$

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \quad \text{with} \quad \begin{aligned} y_{11} &= \left. \frac{I_1}{V_1} \right|_{V_2=0}, & y_{12} &= \left. \frac{I_1}{V_2} \right|_{V_1=0} \\ y_{21} &= \left. \frac{I_2}{V_1} \right|_{V_2=0}, & y_{22} &= \left. \frac{I_2}{V_2} \right|_{V_1=0} \end{aligned} \quad (10.2)$$

**Figure 10.1** A two-port network*Transmission (ABCD) Parameters*

$$\begin{bmatrix} V_1 \\ I_1 \end{bmatrix} = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix} \quad \text{with} \quad \begin{aligned} a_{11} &= \left. \frac{V_1}{V_2} \right|_{I_2=0}, & a_{12} &= \left. \frac{V_1}{-I_2} \right|_{V_2=0} \\ a_{21} &= \left. \frac{I_1}{V_2} \right|_{I_2=0}, & a_{22} &= \left. \frac{I_1}{-I_2} \right|_{V_2=0} \end{aligned} \quad (10.3)$$

$$\begin{bmatrix} V_2 \\ I_2 \end{bmatrix} = \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix} \begin{bmatrix} V_1 \\ -I_1 \end{bmatrix} \quad \text{with} \quad \begin{aligned} b_{11} &= \left. \frac{V_2}{V_1} \right|_{I_1=0}, & b_{12} &= \left. \frac{V_2}{-I_1} \right|_{V_1=0} \\ b_{21} &= \left. \frac{I_2}{V_1} \right|_{I_1=0}, & b_{22} &= \left. \frac{I_2}{-I_1} \right|_{V_1=0} \end{aligned} \quad (10.4)$$

Hybrid Parameters

$$\begin{bmatrix} V_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix} \quad \text{with} \quad \begin{aligned} h_{11} &= \left. \frac{V_1}{I_1} \right|_{V_2=0}, & h_{12} &= \left. \frac{V_1}{V_2} \right|_{I_1=0} \\ h_{21} &= \left. \frac{I_2}{I_1} \right|_{V_2=0}, & z_{22} &= \left. \frac{I_2}{V_2} \right|_{I_1=0} \end{aligned} \quad (10.5)$$

$$\begin{bmatrix} I_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} g_{11} & g_{12} \\ g_{21} & g_{22} \end{bmatrix} \begin{bmatrix} V_1 \\ I_2 \end{bmatrix} \quad \text{with} \quad \begin{aligned} g_{11} &= \left. \frac{I_1}{V_1} \right|_{I_2=0}, & g_{12} &= \left. \frac{I_1}{I_2} \right|_{V_1=0} \\ g_{21} &= \left. \frac{V_2}{V_1} \right|_{I_2=0}, & g_{22} &= \left. \frac{V_2}{I_2} \right|_{V_1=0} \end{aligned} \quad (10.6)$$

Note that, while the z and y -parameter matrices make the inverse of each other and so do the h and g parameter matrices, the a and b -parameter matrices do not have such an inverse relationship.

(Example 10.1) The z -Parameters

Find the z -parameters of the two-port network in Figure 10.2(a).

$$\begin{aligned} z_{11} &= \left. \frac{V_1}{I_1} \right|_{I_2=0} = \frac{11}{3} \parallel \left(\frac{11}{2} + 11 \right) = \frac{11/3 \times (11/2 + 11)}{11/3 + (11/2 + 11)} = 3 \\ z_{12} &= \left. \frac{V_1}{I_2} \right|_{I_1=0} = \frac{11}{(11/3 + 11/2) + 11} \times \frac{11}{3} = 2 \\ z_{21} &= \left. \frac{V_2}{I_1} \right|_{I_2=0} = \frac{11/3}{11/3 + (11/2 + 11)} \times 11 = 2 \\ z_{22} &= \left. \frac{V_2}{I_2} \right|_{I_1=0} = \left(\frac{11}{2} + \frac{11}{3} \right) \parallel 11 = \frac{(11/3 + 11/2) \times 11}{(11/3 + 11/2) + 11} = 5 \end{aligned} \quad (\text{E10.1.1})$$

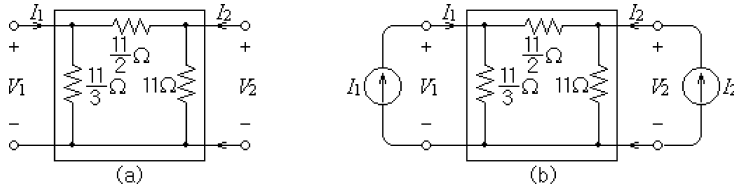


Figure 10.2 (Example 10.1)

Equation (10.1) has been used directly. Alternatively, noting that the two-port network is of the Π -type, we can suppose two current sources I_1 and I_2 are connected at each port, as depicted in Figure 10.2(b), set up the node equation, and solve it for V_1 and V_2 as

$$\begin{bmatrix} 3/11 + 2/11 & -2/11 \\ -2/11 & 1/11 + 2/11 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}; \quad YV = I; \quad V = Y^{-1}I \quad (\text{E10.1.2})$$

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \frac{1}{15/11^2 - 4/11^2} \begin{bmatrix} 3/11 & 2/11 \\ 2/11 & 5/11 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} 3 & 2 \\ 2 & 5 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \quad (\text{E10.1.3})$$

Then this equation is matched with Equation (10.1) to get the same z -parameters as above in matrix form:

$$\begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} = \begin{bmatrix} 3 & 2 \\ 2 & 5 \end{bmatrix} \quad (\text{E10.1.4})$$

(Example 10.2) the y -Parameters

Find the y -parameters of the two-port network in Figure 10.3(a):

$$\begin{aligned} y_{11} &= \left. \frac{I_1}{V_1} \right|_{V_2=0} = \frac{1}{1 + (2||3)} = \frac{1}{1 + 6/5} = \frac{5}{11} \\ y_{12} &= \left. \frac{I_1}{V_2} \right|_{V_2=0} = -\frac{1}{3 + (1||2)} \times \frac{2}{1 + 2} = -\frac{2}{11} \\ y_{21} &= \left. \frac{I_2}{V_1} \right|_{V_2=0} = -\frac{1}{1 + (2||3)} \times \frac{2}{2 + 3} = -\frac{2}{11} \\ y_{22} &= \left. \frac{I_2}{V_2} \right|_{V_2=0} = \frac{1}{3 + (1||2)} = \frac{3}{11} \end{aligned} \quad (\text{E10.2.1})$$

Equation (10.2) has been used directly. Alternatively, noting that the two-port network is of the T -type, we suppose two voltage sources V_1 and V_2 are connected at each port, as depicted in Figure 10.3(b), set up the mesh equation in I_1 and I_2 , and solve it as

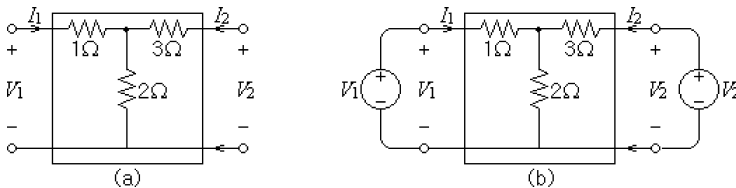


Figure 10.3 (Example 10.2)

$$\begin{bmatrix} 1+2 & 2 \\ 2 & 2+3 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} V_1 \\ V_2 \end{bmatrix}; \quad Z I = V; \quad I = Z^{-1} V \quad (\text{E10.2.2})$$

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \frac{1}{3 \times 5 - 2 \times 2} \begin{bmatrix} 5 & -2 \\ -2 & 3 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} 5/11 & -2/11 \\ -2/11 & 3/11 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \quad (\text{E10.2.3})$$

Then this equation is matched with Equation (10.2) to get the same y -parameter as above in matrix form:

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} = \begin{bmatrix} 5/11 & -2/11 \\ -2/11 & 3/11 \end{bmatrix} \quad (\text{E10.2.4})$$

(Example 10.3) The a -Parameter

The a -parameters of the two-port network in Figure 10.4(a) are obtained as

$$\begin{aligned} a_{11} &= \left. \frac{V_1}{V_2} \right|_{I_2=0} = 1, & a_{12} &= \left. \frac{V_1}{-I_2} \right|_{V_2=0} = 0 \\ a_{21} &= \left. \frac{I_1}{V_2} \right|_{I_2=0} = Y_a, & a_{22} &= \left. \frac{I_1}{-I_2} \right|_{V_2=0} = 1 \end{aligned}; \quad \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ Y_a & 1 \end{bmatrix} \quad (\text{E10.3.1})$$

The a -parameters of the two-port network in Figure 10.4(b) are obtained as

$$\begin{aligned} a_{11} &= \left. \frac{V_1}{V_2} \right|_{I_2=0} = 1, & a_{12} &= \left. \frac{V_1}{-I_2} \right|_{V_2=0} = Z_b \\ a_{21} &= \left. \frac{I_1}{V_2} \right|_{I_2=0} = 0, & a_{22} &= \left. \frac{I_1}{-I_2} \right|_{V_2=0} = 1 \end{aligned}; \quad \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = \begin{bmatrix} 1 & Z_b \\ 0 & 1 \end{bmatrix} \quad (\text{E10.3.2})$$

The a -parameters of the two-port network in Figure 10.4(c) are obtained as

$$\begin{aligned} a_{11} &= \left. \frac{V_1}{V_2} \right|_{I_2=0} = 1, & a_{12} &= \left. \frac{V_1}{-I_2} \right|_{V_2=0} = Z_b \\ a_{21} &= \left. \frac{I_1}{V_2} \right|_{I_2=0} = Y_a, & a_{22} &= \left. \frac{I_1}{-I_2} \right|_{V_2=0} = \frac{Y_a + Y_b}{Y_b} = 1 + Y_a Z_b \end{aligned}; \quad \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = \begin{bmatrix} 1 & Z_b \\ Y_a & 1 + Y_a Z_b \end{bmatrix} \quad (\text{E10.3.3})$$

Note. The two-port network in Figure 10.4(a) has the z -parameters, but its y -parameters do not exist

$$z = \begin{bmatrix} Z_a & Z_a \\ Z_a & Z_a \end{bmatrix}, \quad y = z^{-1} = ? \quad (\text{E10.3.4})$$

On the other hand, the two-port network in Figure 10.4(b) has the y -parameters, but its z -parameters do not exist:

$$y = \begin{bmatrix} Y_b & -Y_b \\ -Y_b & Y_b \end{bmatrix}, \quad z = y^{-1} = ? \quad (\text{E10.3.5})$$

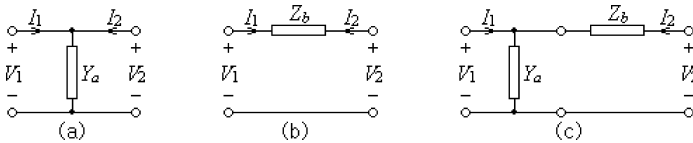


Figure 10.4 (Example 10.3)

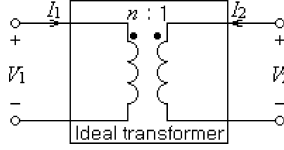


Figure 10.5 (Example 10.4)

(Example 10.4) The a -Parameters

The a -parameters of an ideal transformer with the turns ratio n in Figure 10.5 are obtained from its voltage-current relationship (Section 5.5) as follows:

$$\begin{aligned} V_1(s) &\stackrel{(5.21a)}{=} n V_2(s) ; & \begin{bmatrix} V_1 \\ I_1 \end{bmatrix} &= \begin{bmatrix} n & 0 \\ 0 & 1/n \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix} ; & a &= \begin{bmatrix} n & 0 \\ 0 & 1/n \end{bmatrix} \end{aligned} \quad (\text{E10.4.1})$$

Note. This two-port network has neither z -parameters nor y -parameters.

(Example 10.5) The h -Parameters

Find the h -parameters of the two-port network in Figure 10.6(a). Instead of using Equation (10.5) directly, an expression for V_1 and I_2 will be found in terms of I_1 and V_2 . With this objective in mind, it is supposed that a current source I_1 and a voltage source V_2 are connected at the primary (left) port and the secondary (right) port, respectively, as depicted in Figure 10.6(a). Noting that this circuit has more current sources than voltage sources, we convert the voltage source into a current source as shown in Figure 10.6(b), set up the node equation, and then solve it as

$$\begin{bmatrix} \frac{1}{r_b} & -\frac{1}{r_b} \\ -\frac{1}{r_b} & \frac{1}{r_b} + \frac{1}{r_c} + \frac{1}{r_e} \end{bmatrix} \begin{bmatrix} V_1 \\ V_d \end{bmatrix} = \begin{bmatrix} I_1 \\ \beta I_1 + \frac{V_2}{r_c} \end{bmatrix} \quad (\text{E10.5.1})$$

$$\begin{bmatrix} V_1 \\ V_d \end{bmatrix} = \frac{r_b r_c r_e}{r_c + r_e} \begin{bmatrix} \frac{1}{r_b} + \frac{1}{r_c} + \frac{1}{r_e} & \frac{1}{r_b} \\ \frac{1}{r_b} & \frac{1}{r_b} \end{bmatrix} \begin{bmatrix} I_1 \\ \beta I_1 + \frac{V_2}{r_c} \end{bmatrix} = \begin{bmatrix} \frac{r_c r_e (\beta + 1) + r_b}{r_c + r_e} & \frac{r_e}{r_c + r_e} \\ \frac{r_c r_e (\beta + 1)}{r_c + r_e} & \frac{r_e}{r_c + r_e} \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix} \quad (\text{E10.5.2})$$

This gives the expression of V_1 in terms of I_1 and V_2 . One more thing to find is the expression of $I_2 = i_c$ in terms of I_1 and V_2 , which is obtained by summing the current through r_c and the value of the dependent current source $\beta i_b = \beta I_1$ (in the original circuit of Figure 10.6(a)) as follows:

$$I_2 = \beta I_1 + \frac{1}{r_c} (V_2 - V_d) = \beta I_1 + \frac{1}{r_c} \left[V_2 - \frac{r_c r_e (\beta + 1)}{r_c + r_e} I_1 - \frac{r_e}{r_c + r_e} V_2 \right] = \frac{\beta r_c - r_e}{r_c + r_e} I_1 + \frac{1}{r_c + r_e} V_2 \quad (\text{E10.5.3})$$

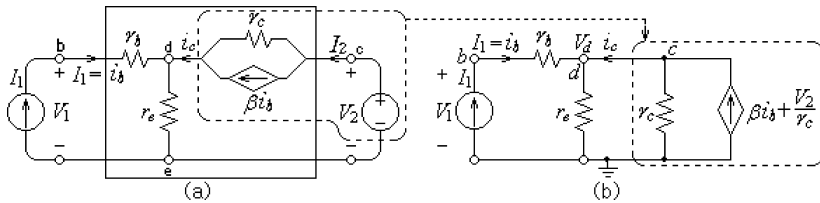


Figure 10.6 A model for a common-emitter transistor (Example 10.5)

Combining this equation with Equation (E10.5.2) yields

$$\begin{bmatrix} V_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} \frac{r_c r_e (\beta + 1)}{r_c + r_e} + r_b & \frac{r_e}{r_c + r_e} \\ \frac{\beta r_c - r_e}{r_c + r_e} & \frac{1}{r_c + r_e} \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix} \xrightarrow{(10.5)} \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} = \begin{bmatrix} \frac{r_c r_e (\beta + 1)}{r_c + r_e} + r_b & \frac{r_e}{r_c + r_e} \\ \frac{\beta r_c - r_e}{r_c + r_e} & \frac{1}{r_c + r_e} \end{bmatrix} \quad (\text{E10.5.4})$$

10.2 Relationships Among Two-Port Parameters

Let two or more sets of two-port parameters commonly describe the relationship among the input/output voltages and currents of a two-port network. Then they must be interrelated with each other. Accordingly, if two sets of two-port parameters exist, one set may be converted to the other set. In this section some formulas are derived that can be used to convert a set of two-port parameters into another one.

10.2.1 The z -Parameters and a -Parameters

Let us derive a formula for converting the z -parameters to the a -parameters:

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \xrightarrow{(10.1)} \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \rightarrow \begin{bmatrix} V_1 \\ I_1 \end{bmatrix} \xrightarrow{(10.3)} \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix}$$

Starting with the inverse relationship of Equation (10.1)

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \frac{1}{\Delta z} \begin{bmatrix} z_{22} & -z_{12} \\ -z_{21} & z_{11} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \quad \text{with } \Delta z = z_{11}z_{22} - z_{12}z_{21} \quad (10.7)$$

Equation (10.1-1) is substituted into Equation (10.7-1) to get the expression of I_1 in terms of V_2 and $-I_2$ as

$$\begin{aligned} I_1 &\stackrel{(10.7-1)}{=} \frac{z_{22}}{\Delta z} V_1 - \frac{z_{12}}{\Delta z} V_2 \stackrel{(10.1-1)}{=} \frac{z_{22}}{\Delta z} (z_{11}I_1 + z_{12}I_2) - \frac{z_{12}}{\Delta z} V_2 \\ I_1 &= \frac{z_{22}z_{12}I_2 - z_{12}V_2}{\Delta z - z_{11}z_{22}} = \frac{z_{22}z_{12}I_2 - z_{12}V_2}{-z_{12}z_{21}} = \frac{1}{z_{21}} V_2 + \frac{z_{22}}{z_{21}} (-I_2) \end{aligned}$$

Substituting this equation back into Equation (10.1-1) yields

$$V_1 = z_{11} \left(\frac{1}{z_{21}} V_2 - \frac{z_{22}}{z_{21}} I_2 \right) + z_{12} I_2 = \frac{z_{11}}{z_{21}} V_2 + \frac{\Delta z}{z_{21}} (-I_2)$$

Matching the combination of the above two equations with Equation (10.3) yields

$$\begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = \begin{bmatrix} \frac{z_{11}}{z_{21}} & \frac{\Delta z}{z_{21}} \\ \frac{1}{z_{21}} & \frac{z_{22}}{z_{21}} \end{bmatrix} \quad (10.8)$$

Now let us derive a formula for converting the a -parameters to the z -parameters:

$$\begin{bmatrix} V_1 \\ I_1 \end{bmatrix} \stackrel{(10.3)}{=} \begin{bmatrix} a_{11} & -a_{12} \\ a_{21} & -a_{22} \end{bmatrix} \begin{bmatrix} V_2 \\ I_2 \end{bmatrix} \rightarrow \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \stackrel{(10.1)}{=} \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

Starting with the inverse relationship of Equation (10.3)

$$\begin{bmatrix} V_2 \\ I_2 \end{bmatrix} = \frac{1}{\Delta a} \begin{bmatrix} a_{22} & -a_{12} \\ a_{21} & -a_{11} \end{bmatrix} \begin{bmatrix} V_1 \\ I_1 \end{bmatrix} \quad \text{with} \quad \Delta a = a_{11}a_{22} - a_{12}a_{21} \quad (10.9)$$

Equation (10.3-1) is substituted into Equation (10.9-1) to get the expression of V_2 in terms of I_1 and I_2 as

$$\begin{aligned} V_2 &\stackrel{(10.9-1)}{=} \frac{1}{\Delta a} (a_{22}V_1 - a_{12}I_1) \stackrel{(10.3-1)}{=} \frac{a_{22}}{\Delta a} (a_{11}V_2 - a_{12}I_2) - \frac{a_{12}}{\Delta a} I_1 \\ V_2 &= \frac{-a_{22}a_{12}I_2 - a_{12}I_1}{\Delta a - a_{11}a_{22}} = \frac{a_{22}a_{12}I_2 + a_{12}I_1}{a_{12}a_{21}} = \frac{1}{a_{21}} I_1 + \frac{a_{22}}{a_{21}} I_2 \end{aligned}$$

Substituting this equation back into Equation (10.3-1) yields

$$V_1 = a_{11} \left(\frac{1}{a_{21}} I_1 + \frac{a_{22}}{a_{21}} I_2 \right) - a_{12}I_2 = \frac{a_{11}}{a_{21}} I_1 + \frac{\Delta a}{a_{21}} I_2$$

Matching the combination of the above two equations with Equation (10.1) yields

$$\begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} = \begin{bmatrix} \frac{a_{11}}{a_{21}} & \frac{\Delta a}{a_{21}} \\ \frac{1}{a_{21}} & \frac{a_{22}}{a_{21}} \end{bmatrix} \quad (10.10)$$

10.2.2 The a -Parameters and h -Parameters

Let us derive a formula for converting the a -parameters to the h -parameters:

$$\begin{bmatrix} V_1 \\ I_1 \end{bmatrix} \stackrel{(10.3)}{=} \begin{bmatrix} a_{11} & -a_{12} \\ a_{21} & -a_{22} \end{bmatrix} \begin{bmatrix} V_2 \\ I_2 \end{bmatrix} \rightarrow \begin{bmatrix} V_1 \\ I_2 \end{bmatrix} \stackrel{(10.5)}{=} \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix}$$

Starting with the inverse relationship of Equation (10.3)

$$\begin{bmatrix} V_2 \\ I_2 \end{bmatrix} = \frac{1}{\Delta a} \begin{bmatrix} a_{22} & -a_{12} \\ a_{21} & -a_{11} \end{bmatrix} \begin{bmatrix} V_1 \\ I_1 \end{bmatrix} \quad \text{with} \quad \Delta a = a_{11}a_{22} - a_{12}a_{21} \quad (10.9)$$

Equation (10.3-1) is substituted into Equation (10.9-2) to get the expression of I_2 in terms of I_1 and V_2 as

$$\begin{aligned} I_2 &\stackrel{(10.9-2)}{=} \frac{a_{21}}{\Delta a} V_1 - \frac{a_{11}}{\Delta a} I_1 \stackrel{(10.3-1)}{=} \frac{a_{21}}{\Delta a} (a_{11}V_2 - a_{12}I_2) - \frac{a_{11}}{\Delta a} I_1 \\ I_2 &= \frac{a_{21}a_{11}V_2 - a_{11}I_1}{\Delta a + a_{21}a_{12}} = -\frac{1}{a_{22}} I_1 + \frac{a_{21}}{a_{22}} V_2 \end{aligned}$$

Substituting this equation back into Equation (10.3-1) yields

$$V_1 = a_{11}V_2 - a_{12}I_2 = a_{11}V_2 - a_{12}\left(-\frac{1}{a_{22}}I_1 + \frac{a_{21}}{a_{22}}V_2\right) = \frac{a_{12}}{a_{22}}I_1 + \frac{\Delta a}{a_{22}}V_2$$

Matching the combination of the above two equations with Equation (10.5) yields

$$\begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} = \begin{bmatrix} \frac{a_{12}}{a_{22}} & \frac{\Delta a}{a_{22}} \\ -\frac{1}{a_{22}} & \frac{a_{21}}{a_{22}} \end{bmatrix} \quad (10.11)$$

Now let us derive a formula for converting the h -parameters to the a -parameters:

$$\begin{bmatrix} V_1 \\ I_2 \end{bmatrix} \stackrel{(10.5)}{=} \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix} \rightarrow \begin{bmatrix} V_1 \\ I_1 \end{bmatrix} \stackrel{(10.3)}{=} \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix}$$

Starting with the inverse relationship of Equation (10.5)

$$\begin{bmatrix} I_1 \\ V_2 \end{bmatrix} = \frac{1}{\Delta h} \begin{bmatrix} h_{22} & -h_{12} \\ -h_{21} & h_{11} \end{bmatrix} \begin{bmatrix} V_1 \\ I_2 \end{bmatrix} \quad \text{with} \quad \Delta h = h_{11}h_{22} - h_{12}h_{21} \quad (10.12)$$

Equation (10.5-1) is substituted into Equation (10.12-1) to get the expression of I_1 in terms of V_2 and I_2 as

$$\begin{aligned} I_1 &\stackrel{(10.12-1)}{=} \frac{1}{\Delta h} (h_{22}V_1 - h_{12}I_2) \stackrel{(10.5-1)}{=} \frac{h_{22}}{\Delta h} (h_{11}I_1 + h_{12}V_2) - \frac{h_{12}}{\Delta h} I_2 \\ I_1 &= \frac{h_{22}h_{12}V_2 - h_{12}I_2}{\Delta h - h_{11}h_{22}} = \frac{-h_{22}h_{12}V_2 + h_{12}I_2}{h_{12}h_{21}} = -\frac{h_{22}}{h_{21}}V_2 + \frac{1}{h_{21}}I_2 \end{aligned}$$

Substituting this equation back into Equation (10.5-1) yields

$$V_1 = h_{11}\left(-\frac{h_{22}}{h_{21}}V_2 + \frac{1}{h_{21}}I_2\right) + h_{12}V_2 = -\frac{\Delta h}{h_{21}}V_2 + \frac{h_{11}}{h_{21}}I_2$$

Matching the combination of the above two equations with Equation (10.3) yields

$$\begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = \begin{bmatrix} -\frac{\Delta h}{h_{21}} & -\frac{h_{11}}{h_{21}} \\ -\frac{h_{22}}{h_{21}} & -\frac{1}{h_{21}} \end{bmatrix} \quad (10.13)$$

10.2.3 The z -Parameters and h -Parameters

Let us derive a formula for converting the z -parameters to the h -parameters:

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \stackrel{(10.1)}{=} \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \rightarrow \begin{bmatrix} V_1 \\ I_2 \end{bmatrix} \stackrel{(10.5)}{=} \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix}$$

Starting with the inverse relationship of Equation (10.1)

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \frac{1}{\Delta z} \begin{bmatrix} z_{22} & -z_{12} \\ -z_{21} & z_{11} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \quad \text{with} \quad \Delta z = z_{11}z_{22} - z_{12}z_{21} \quad (10.7)$$

Equation (10.1-1) is substituted into Equation (10.7-2) to get the expression of I_2 in terms of I_1 and V_2 as

$$\begin{aligned} I_2 &\stackrel{(10.7-2)}{=} -\frac{z_{21}}{\Delta z} V_1 + \frac{z_{11}}{\Delta z} V_2 \stackrel{(10.1-1)}{=} -\frac{z_{21}}{\Delta z} (z_{11}I_1 + z_{12}I_2) + \frac{z_{11}}{\Delta z} V_2 \\ I_2 &= \frac{-z_{21}z_{11}I_1 + z_{11}V_2}{\Delta z + z_{12}z_{21}} = -\frac{z_{21}}{z_{22}} I_1 + \frac{1}{z_{22}} V_2 \end{aligned}$$

Substituting this equation back into Equation (10.1-1) yields

$$V_1 = z_{11}I_1 + z_{12}I_2 = z_{11}I_1 + z_{12} \left(-\frac{z_{21}}{z_{22}} I_1 + \frac{1}{z_{22}} V_2 \right) = \frac{\Delta z}{z_{22}} I_1 + \frac{z_{12}}{z_{22}} V_2$$

Matching the combination of the above two equations with Equation (10.5) yields

$$\begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} = \begin{bmatrix} \frac{\Delta z}{z_{22}} & \frac{z_{12}}{z_{22}} \\ -\frac{z_{21}}{z_{22}} & \frac{1}{z_{22}} \end{bmatrix} \quad (10.14)$$

Now let us derive a formula for converting the h -parameters to the z -parameters:

$$\begin{bmatrix} V_1 \\ I_2 \end{bmatrix} \stackrel{(10.5)}{=} \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix} \rightarrow \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \stackrel{(10.1)}{=} \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

Starting with the inverse relationship of Equation (10.5)

$$\begin{bmatrix} I_1 \\ V_2 \end{bmatrix} = \frac{1}{\Delta h} \begin{bmatrix} h_{22} & -h_{12} \\ -h_{21} & h_{11} \end{bmatrix} \begin{bmatrix} V_1 \\ I_2 \end{bmatrix} \quad \text{with} \quad \Delta h = h_{11}h_{22} - h_{12}h_{21} \quad (10.12)$$

Equation (10.5-1) is substituted into Equation (10.12-2) to get the expression of V_2 in terms of I_1 and I_2 as

$$\begin{aligned} V_2 &\stackrel{(10.12-2)}{=} \frac{1}{\Delta h} (-h_{21}V_1 + h_{11}I_2) \stackrel{(10.5-1)}{=} -\frac{h_{21}}{\Delta h} (h_{11}I_1 + h_{12}V_2) + \frac{h_{11}}{\Delta h} I_2 \\ V_2 &= \frac{-h_{21}h_{11}I_1 + h_{11}I_2}{\Delta h + h_{12}h_{21}} = \frac{-h_{21}h_{11}I_1 + h_{11}I_2}{h_{11}h_{22}} = -\frac{h_{21}}{h_{22}} I_1 + \frac{1}{h_{22}} I_2 \end{aligned}$$

Substituting the equation back into Equation (10.5-1) yields

$$V_1 = h_{11}I_1 + h_{12}V_2 = h_{11}I_1 + h_{12} \left(-\frac{h_{21}}{h_{22}} I_1 + \frac{1}{h_{22}} I_2 \right) = \frac{\Delta h}{h_{22}} I_1 + \frac{h_{12}}{h_{22}} I_2$$

Matching the combination of the above two equations with Equation (10.1) yields

$$\begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} = \begin{bmatrix} \frac{\Delta h}{h_{22}} & \frac{h_{12}}{h_{22}} \\ -\frac{h_{21}}{h_{22}} & \frac{1}{h_{22}} \end{bmatrix} \quad (10.15)$$

Similarly, the conversion formulas involving the y , b , and g -parameters can be derived, all of which are listed in Table 10.1 and cast into the MATLAB routine `port_conversion()`.

Table 10.1 The conversion among two-port parameters

	z	y	a	b	h	g
z	$\begin{bmatrix} z_{11} & z_{12} \\ z_{11} & z_{12} \end{bmatrix}$	$\begin{bmatrix} \frac{y_{22}}{\Delta y} & -\frac{y_{12}}{\Delta y} \\ -\frac{y_{21}}{\Delta y} & \frac{y_{11}}{\Delta y} \end{bmatrix}$	$\begin{bmatrix} \frac{a_{11}}{a_{21}} & \frac{\Delta a}{a_{21}} \\ \frac{a_{21}}{a_{21}} & \frac{a_{22}}{a_{21}} \end{bmatrix}$	$\begin{bmatrix} \frac{b_{22}}{b_{21}} & \frac{1}{b_{21}} \\ \frac{b_{21}}{b_{21}} & \frac{b_{11}}{b_{21}} \end{bmatrix}$	$\begin{bmatrix} \frac{\Delta h}{h_{22}} & \frac{h_{12}}{h_{22}} \\ -\frac{h_{21}}{h_{22}} & \frac{1}{h_{22}} \end{bmatrix}$	$\begin{bmatrix} \frac{1}{g_{11}} & -\frac{g_{12}}{g_{11}} \\ \frac{g_{21}}{g_{11}} & \frac{\Delta g}{g_{11}} \end{bmatrix}$
y	$\begin{bmatrix} \frac{z_{22}}{\Delta z} & -\frac{z_{12}}{\Delta z} \\ -\frac{z_{21}}{\Delta z} & \frac{z_{11}}{\Delta z} \end{bmatrix}$	$\begin{bmatrix} y_{21} & y_{22} \\ y_{11} & y_{12} \end{bmatrix}$	$\begin{bmatrix} \frac{a_{22}}{a_{12}} & -\frac{\Delta a}{a_{12}} \\ \frac{1}{a_{12}} & \frac{a_{11}}{a_{12}} \end{bmatrix}$	$\begin{bmatrix} \frac{b_{11}}{b_{12}} & -\frac{1}{b_{12}} \\ -\frac{\Delta b}{b_{12}} & \frac{b_{22}}{b_{12}} \end{bmatrix}$	$\begin{bmatrix} \frac{1}{h_{11}} & -\frac{h_{12}}{h_{11}} \\ \frac{h_{21}}{h_{11}} & \frac{\Delta h}{h_{11}} \end{bmatrix}$	$\begin{bmatrix} \frac{\Delta g}{g_{22}} & \frac{g_{12}}{g_{22}} \\ -\frac{g_{21}}{g_{22}} & \frac{1}{g_{22}} \end{bmatrix}$
a	$\begin{bmatrix} \frac{z_{11}}{\Delta z} & \frac{\Delta z}{z_{21}} \\ \frac{1}{z_{21}} & \frac{z_{22}}{z_{21}} \end{bmatrix}$	$\begin{bmatrix} -\frac{y_{22}}{\Delta y} & -\frac{1}{\Delta y} \\ \frac{y_{21}}{\Delta y} & -\frac{y_{11}}{\Delta y} \end{bmatrix}$	$\begin{bmatrix} \frac{a_{21}}{a_{11}} & \frac{a_{22}}{a_{11}} \\ \frac{\Delta a}{a_{11}} & \frac{\Delta a}{a_{11}} \end{bmatrix}$	$\begin{bmatrix} \frac{b_{22}}{\Delta b} & \frac{b_{12}}{\Delta b} \\ \frac{b_{21}}{\Delta b} & \frac{b_{11}}{\Delta b} \end{bmatrix}$	$\begin{bmatrix} -\frac{\Delta h}{h_{21}} & -\frac{h_{11}}{h_{21}} \\ \frac{h_{22}}{h_{21}} & -\frac{1}{h_{21}} \end{bmatrix}$	$\begin{bmatrix} \frac{1}{g_{21}} & \frac{g_{22}}{g_{21}} \\ \frac{g_{11}}{g_{21}} & \frac{\Delta g}{g_{21}} \end{bmatrix}$
b	$\begin{bmatrix} \frac{z_{22}}{\Delta z} & \frac{\Delta z}{z_{12}} \\ \frac{1}{z_{12}} & \frac{z_{11}}{z_{12}} \end{bmatrix}$	$\begin{bmatrix} -\frac{y_{11}}{\Delta y} & \frac{1}{\Delta y} \\ -\frac{y_{12}}{\Delta y} & \frac{y_{22}}{\Delta y} \end{bmatrix}$	$\begin{bmatrix} \frac{a_{22}}{\Delta a} & \frac{a_{12}}{\Delta a} \\ \frac{a_{21}}{\Delta a} & \frac{a_{11}}{\Delta a} \end{bmatrix}$	$\begin{bmatrix} \frac{b_{21}}{b_{11}} & \frac{b_{22}}{b_{11}} \\ \frac{b_{11}}{b_{11}} & \frac{b_{12}}{b_{11}} \end{bmatrix}$	$\begin{bmatrix} \frac{1}{h_{12}} & \frac{h_{11}}{h_{12}} \\ \frac{h_{22}}{h_{12}} & \frac{\Delta h}{h_{12}} \end{bmatrix}$	$\begin{bmatrix} -\frac{\Delta g}{g_{12}} & -\frac{g_{22}}{g_{12}} \\ -\frac{g_{11}}{g_{12}} & -\frac{1}{g_{12}} \end{bmatrix}$
h	$\begin{bmatrix} \frac{\Delta z}{z_{22}} & \frac{z_{12}}{z_{22}} \\ -\frac{z_{21}}{z_{22}} & \frac{1}{z_{22}} \end{bmatrix}$	$\begin{bmatrix} \frac{1}{y_{11}} & -\frac{y_{12}}{y_{11}} \\ \frac{y_{21}}{y_{11}} & \frac{\Delta y}{y_{11}} \end{bmatrix}$	$\begin{bmatrix} \frac{a_{12}}{a_{22}} & \frac{\Delta a}{a_{22}} \\ -\frac{1}{a_{22}} & \frac{a_{21}}{a_{22}} \end{bmatrix}$	$\begin{bmatrix} \frac{b_{12}}{b_{11}} & \frac{1}{b_{11}} \\ \frac{\Delta b}{b_{11}} & \frac{b_{21}}{b_{11}} \end{bmatrix}$	$\begin{bmatrix} \frac{h_{21}}{h_{11}} & \frac{h_{22}}{h_{11}} \\ \frac{h_{11}}{h_{11}} & \frac{h_{12}}{h_{11}} \end{bmatrix}$	$\begin{bmatrix} \frac{g_{22}}{\Delta g} & -\frac{g_{12}}{\Delta g} \\ -\frac{g_{21}}{\Delta g} & \frac{g_{11}}{\Delta g} \end{bmatrix}$
g	$\begin{bmatrix} \frac{1}{z_{11}} & -\frac{z_{12}}{z_{11}} \\ \frac{z_{21}}{z_{11}} & \frac{\Delta z}{z_{11}} \end{bmatrix}$	$\begin{bmatrix} \frac{\Delta y}{y_{22}} & \frac{y_{12}}{y_{22}} \\ -\frac{y_{21}}{y_{22}} & \frac{1}{y_{22}} \end{bmatrix}$	$\begin{bmatrix} \frac{a_{21}}{a_{11}} & -\frac{\Delta a}{a_{11}} \\ \frac{1}{a_{11}} & \frac{a_{12}}{a_{11}} \end{bmatrix}$	$\begin{bmatrix} \frac{b_{21}}{b_{22}} & -\frac{1}{b_{22}} \\ \frac{\Delta b}{b_{22}} & \frac{b_{12}}{b_{22}} \end{bmatrix}$	$\begin{bmatrix} \frac{h_{22}}{\Delta h} & -\frac{h_{12}}{\Delta h} \\ -\frac{h_{21}}{\Delta h} & \frac{h_{11}}{\Delta h} \end{bmatrix}$	$\begin{bmatrix} \frac{g_{21}}{g_{11}} & \frac{g_{22}}{g_{11}} \\ \frac{g_{11}}{g_{11}} & \frac{g_{12}}{g_{11}} \end{bmatrix}$

```
function P2=port_conversion(P1,which)
%Conversion Formulas among Two-port network parameters (Table 10.1)
% Copyright: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only
switch lower(which)
case 'z2y', P2=P1^-1;
case 'z2a', P2=[P1(1,1) det(P1); 1 P1(2,2)]/P1(2,1);
case 'z2b', P2=[P1(2,2) det(P1); 1 P1(1,1)]/P1(1,2);
case 'z2h', P2=[det(P1) P1(1,2); -P1(2,1) 1]/P1(2,2);
case 'z2g', P2=[1 -P1(1,2); P1(2,1) det(P1)]/P1(1,1);
case 'y2z', P2=P1^-1;
case 'y2a', P2=[P1(2,2) 1; det(P1) P1(1,1)]/P1(2,1);
```



```

case 'y2b', P2=[-P1(1,1) 1; -det(P1) P1(2,2)]/P1(1,2);
case 'y2h', P2=[1 -P1(1,2); P1(2,1) det(P1)]/P1(1,1);
case 'y2g', P2=[det(P1) P1(1,2); -P1(2,1) 1]/P1(2,2);
case 'a2z', P2=[P1(1,1) det(P1); 1 P1(2,2)]/P1(2,1);
case 'a2y', P2=[P1(2,2) -det(P1); -1 P1(1,1)]/P1(1,2);
case 'a2b', P2=[P1(2,2) P1(1,2); P1(2,1) P1(1,1)]/det(P1);
case 'a2h', P2=[P1(1,2) det(P1); -1 P1(2,1)]/P1(2,2);
case 'a2g', P2=[P1(2,1) -det(P1); 1 P1(1,2)]/P1(1,1);
case 'b2z', P2=[P1(2,2) 1; det(P1) P1(1,1)]/P1(2,1);
case 'b2y', P2=[P1(1,1) -1; -det(P1) P1(2,2)]/P1(1,2);
case 'b2a', P2=[P1(2,2) P1(1,2); P1(2,1) P1(1,1)]/det(P1);
case 'b2h', P2=[P1(1,2) 1; det(P1) P1(2,1)]/P1(1,1);
case 'b2g', P2=[P1(2,1) -1; det(P1) P1(1,2)]/P1(2,2);
case 'h2z', P2=[det(P1) P1(1,2); -P1(2,1) 1]/P1(2,2);
case 'h2y', P2=[1 -P1(1,2); P1(2,1) det(P1)]/P1(1,1);
case 'h2a', P2=[-det(P1) P1(1,1); P1(2,2) 1]/P1(2,1);
case 'h2b', P2=[1 P1(1,1); P1(2,2) det(P1)]/P1(1,2);
case 'h2g', P2=P1^-1;
case 'g2z', P2=[1 -P1(1,2); P1(2,1) det(P1)]/P1(1,1);
case 'g2y', P2=[det(P1) P1(1,2); -P1(2,1) 1]/P1(2,2);
case 'g2a', P2=[1 P1(2,2); P1(1,1) det(P1)]/P1(2,1);
case 'g2b', P2=[-det(P1) P1(2,2); P1(1,1) 1]/P1(1,2);
case 'g2h', P2=P1^-1;
otherwise error('What do you want to do by port_conversion()?')
end

```

10.3 Reciprocity of a Two-Port Network

Before studying the reciprocity of a two-port network, we will expose ourselves to several network functions that are the ratios of the Laplace/phasor transform of a response function to that of an excitation function with all the initial conditions set to zero. For example, the *driving-point impedances* at port 1 and port 2 are defined respectively as

$$\left. \frac{V_1(s)}{I_1(s)} \right|_{I_2=0} \quad (\text{with port 2 open}), \quad \left. \frac{V_2(s)}{I_2(s)} \right|_{V_1=0} \quad (\text{with port 1 closed})$$

where port 2 (on the load side) is supposed to be open with no load and port 1 (on the source side) is supposed to be short with zero source impedance. The *transfer impedance* from port 1 to port 2 and the transfer impedance from port 2 to port 1 are defined respectively as

$$z_{21} = \left. \frac{V_2(s)}{I_1(s)} \right|_{I_2=0} \quad (\text{with port 2 open}), \quad z_{12} = \left. \frac{V_1(s)}{I_2(s)} \right|_{I_1=0} \quad (\text{with port 1 open})$$

These impedances can be obtained by applying a current source I_1/I_2 at port 1/2 and measuring the (open-circuit) voltage at port 2/1, respectively. The forward transfer *voltage/current ratios* are defined as

$$g_{21} = \left. \frac{V_2}{V_1} \right|_{I_2=0} \quad (\text{with port 2 open}), \quad h_{21} = \left. \frac{I_2}{I_1} \right|_{V_2=0} \quad (\text{with port 2 closed})$$

where port 2 is supposed to be open/short for voltage/current ratio, respectively. The two-port network functions are listed together with the related parameters in Table 10.2.

The reciprocity of a two-port network is defined as follows:

Table 10.2 Network functions for a two-port network

Driving-point functions	Driving-point impedance:	$z_{11} = \left. \frac{V_1}{I_1} \right _{I_2=0}, \quad g_{22} = \left. \frac{V_2}{I_2} \right _{V_1=0}$	(10.16)
	Driving-point admittance:	$g_{11} = \left. \frac{I_1}{V_1} \right _{I_2=0}, \quad y_{22} = \left. \frac{I_2}{V_2} \right _{V_1=0}$	(10.17)
Transfer functions	Transfer impedances:	$z_{21} = \left. \frac{V_2}{I_1} \right _{I_2=0}, \quad z_{12} = \left. \frac{V_1}{I_2} \right _{I_1=0}$	(10.18)
	Transfer admittances:	$y_{21} = \left. \frac{I_2}{V_1} \right _{V_2=0}, \quad y_{12} = \left. \frac{I_1}{V_2} \right _{V_1=0}$	(10.19)
	Forward/reverse voltage ratio:	$g_{21} = \left. \frac{V_2}{V_1} \right _{I_2=0}, \quad a_{11} = \left. \frac{V_1}{V_2} \right _{I_2=0}$	(10.20)
	Forward/reverse current ratio:	$h_{21} = \left. \frac{I_2}{I_1} \right _{V_2=0}, \quad -a_{22} = \left. \frac{I_1}{I_2} \right _{V_2=0}$	(10.21)

[Remark 10.1] Reciprocity and Symmetry of a Two-Port Network

A two-port network is said to be *reciprocal* if the transfer impedance/admittance from port 1 to port 2 and those from port 2 to port 1 are the same or, equivalently, if one of the following conditions hold:

$$z_{12} = z_{21}, \quad y_{12} = y_{21}, \quad h_{12} = -h_{21}, \quad g_{12} = -g_{21}, \quad \Delta a = 1, \quad \Delta b = 1 \quad (10.22)$$

In general, the networks composed of two-terminal elements such as resistors, inductors, and capacitors with no dependent (controlled) source turn out to be reciprocal.

A reciprocal two-port network is also said to be *symmetrical* if interchanging the two ports does not alter the voltages/currents at the two ports or, equivalently, if one of the following conditions holds:

$$z_{11} = z_{22}, \quad y_{11} = y_{22}, \quad a_{11} = a_{22}, \quad b_{11} = b_{22}, \quad \Delta h = 1, \quad \Delta g = 1 \quad (10.23)$$

(Example 10.6) A Reciprocal Network

In order to test for the reciprocity of the two-port network of Figure 10.7, its z -parameters can be found by applying two current sources I_1/I_2 to port 1/2, respectively, setting up the node equations in the unknowns V_1 , V_2 , and V_3 , and obtaining the expressions of V_1/V_2 in terms of I_1/I_2 :

$$\begin{bmatrix} 1+s & -1 & -s \\ -1 & 1+1/s & -1/s \\ -s & -1/s & s+1/s+2 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \end{bmatrix} = \begin{bmatrix} I_1 \\ I_2 \\ 0 \end{bmatrix} \quad (\text{E10.6.1})$$

Since this seems to be a bit complicated to solve, let us borrow the power of MATLAB for symbolic computations.

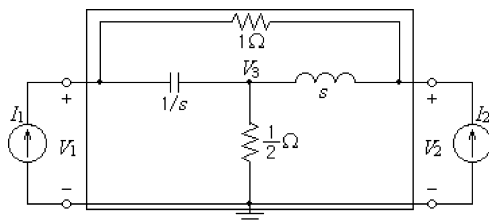


Figure 10.7 A reciprocal two-port network

```
%cir10e06.m: Symbolic computation to solve a 3x3 node equation
clear
syms s
Y=[1+s -1 -s;-1 1+1/s -1/s;-s -1/s s+1/s+2]; %Node admittance matrix
V1=Y\[1 0 0]'; V2=Y\[0 1 0]'; %Node voltage vectors for I1 and I2
z=[V1(1:2) V2(1:2)] % the z-parameter matrix
y=z^-1 % the y-parameter matrix if necessary
```

```
>>cir10e06
```

```
z=[1/2*(3*s+3+s^2)/(s^2+1+s), 1/2*(s^2+1+3*s)/(s^2+1+s)]
[1/2*(s^2+1+3*s)/(s^2+1+s), 1/2*(3*s^2+1+3*s)/(s^2+1+s)]
y=[(3*s^2+1+3*s)/(s^2+1+2*s), -(s^2+1+3*s)/(s^2+1+2*s)]
[-(s^2+1+3*s)/(s^2+1+2*s), (3*s+3+s^2)/(s^2+1+2*s)]
```

From this result of running the MATLAB program `cir10e06.m`, the z and y -parameters are obtained as

$$\begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} = \begin{bmatrix} \frac{s^2 + 3s + 3}{2(s^2 + s + 1)} & \frac{s^2 + 3s + 1}{2(s^2 + s + 1)} \\ \frac{s^2 + 3s + 1}{2(s^2 + s + 1)} & \frac{3s^2 + 3s + 1}{2(s^2 + s + 1)} \end{bmatrix}, \quad \begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} = \begin{bmatrix} \frac{3s^2 + 3s + 1}{(s + 1)^2} & -\frac{s^2 + 3s + 1}{(s + 1)^2} \\ -\frac{s^2 + 3s + 1}{(s + 1)^2} & \frac{s^2 + 3s + 3}{(s + 1)^2} \end{bmatrix} \quad (\text{E10.6.2})$$

The symmetry of these z and y -parameter matrices ($z_{12} = z_{21}$, $y_{12} = y_{21}$) together with Equation (10.22) implies that the two-port network is reciprocal, which can be guessed from the fact that it is composed of two-terminal elements such as resistors, inductors, and capacitors with no dependent source.

10.4 Interconnection of Two-Port Networks

For resistors connected in series/parallel, it is convenient to use the impedance/admittance for obtaining their equivalent parameter. Likewise, depending on the type of connection of two two-port networks, there are some sets of parameters that are more convenient than other sets of parameters for obtaining their equivalent parameters.

10.4.1 Series Connection and z -Parameters

Consider the connection of two two-port networks in Figure 10.8. Since the currents of the two two-port networks are the same ($I_1^a = I_1^b = I_1$, $I_2^a = I_2^b = I_2$) and their voltages are added to

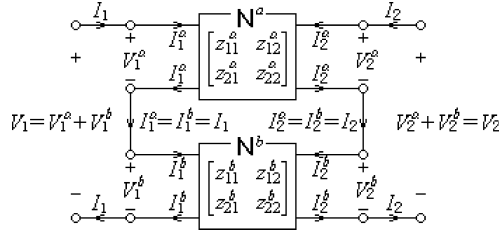


Figure 10.8 Series connection of two two-port networks

yield the voltages of the overall network ($V_1^a + V_1^b = V_1$, $V_2^a + V_2^b = V_2$), it is called a *series connection* and its overall z parameter matrix equals the sum of the z -parameter matrices of the two networks:

$$\begin{aligned} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} &= \begin{bmatrix} V_1^a \\ V_2^a \end{bmatrix} + \begin{bmatrix} V_1^b \\ V_2^b \end{bmatrix} = \begin{bmatrix} z_{11}^a & z_{12}^a \\ z_{21}^a & z_{22}^a \end{bmatrix} \begin{bmatrix} I_1^a \\ I_2^a \end{bmatrix} + \begin{bmatrix} z_{11}^b & z_{12}^b \\ z_{21}^b & z_{22}^b \end{bmatrix} \begin{bmatrix} I_1^b \\ I_2^b \end{bmatrix} \\ &= \begin{bmatrix} z_{11}^a + z_{11}^b & z_{12}^a + z_{12}^b \\ z_{21}^a + z_{21}^b & z_{22}^a + z_{22}^b \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \end{aligned} \quad (10.24)$$

10.4.2 Parallel (Shunt) Connection and y -Parameters

Consider the connection of two two-port networks in Figure 10.9. Since the voltages of the two two-port networks are the same ($V_1^a = V_1^b = V_1$, $V_2^a = V_2^b = V_2$) and their currents are added to yield the currents of the overall network ($I_1^a + I_1^b = I_1$, $I_2^a + I_2^b = I_2$), it is called a *parallel connection* and its overall y -parameter matrix equals the sum of the y -parameter matrices of the two networks:

$$\begin{aligned} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} &= \begin{bmatrix} I_1^a \\ I_2^a \end{bmatrix} + \begin{bmatrix} I_1^b \\ I_2^b \end{bmatrix} = \begin{bmatrix} y_{11}^a & y_{12}^a \\ y_{21}^a & y_{22}^a \end{bmatrix} \begin{bmatrix} V_1^a \\ V_2^a \end{bmatrix} + \begin{bmatrix} y_{11}^b & y_{12}^b \\ y_{21}^b & y_{22}^b \end{bmatrix} \begin{bmatrix} V_1^b \\ V_2^b \end{bmatrix} \\ &= \begin{bmatrix} y_{11}^a + y_{11}^b & y_{12}^a + y_{12}^b \\ y_{21}^a + y_{21}^b & y_{22}^a + y_{22}^b \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \end{aligned} \quad (10.25)$$

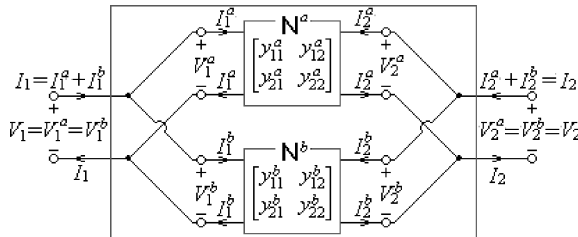


Figure 10.9 Parallel connection of two two-port networks

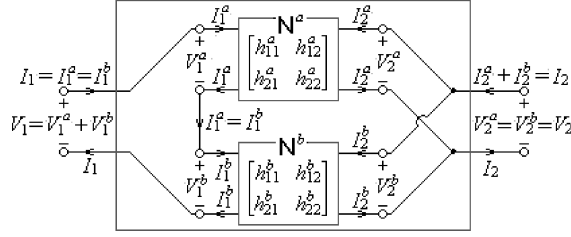


Figure 10.10 Series-parallel connection of two two-port networks

10.4.3 Series-Parallel (Shunt) Connection and h -Parameters

Consider the connection of two two-port networks in Figure 10.10. Since the input ports of the two networks are connected in series ($I_1^a = I_1^b = I_1$, $V_1^a + V_1^b = V_1$) and their output ports are connected in parallel ($V_2^a = V_2^b = V_2$, $I_2^a + I_2^b = I_2$), it is called a *series-parallel connection* and its overall h -parameter matrix equals the sum of the h -parameter matrices of the two networks:

$$\begin{aligned} \begin{bmatrix} V_1 \\ I_2 \end{bmatrix} &= \begin{bmatrix} V_1^a \\ I_2^a \end{bmatrix} + \begin{bmatrix} V_1^b \\ I_2^b \end{bmatrix} = \begin{bmatrix} h_{11}^a & h_{12}^a \\ h_{21}^a & h_{22}^a \end{bmatrix} \begin{bmatrix} I_1^a \\ V_2^a \end{bmatrix} + \begin{bmatrix} h_{11}^b & h_{12}^b \\ h_{21}^b & h_{22}^b \end{bmatrix} \begin{bmatrix} I_1^b \\ V_2^b \end{bmatrix} \\ &= \begin{bmatrix} h_{11}^a + h_{11}^b & h_{12}^a + h_{12}^b \\ h_{21}^a + h_{21}^b & h_{22}^a + h_{22}^b \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix} \end{aligned} \quad (10.26)$$

10.4.4 Parallel (Shunt)-Series Connection and g -Parameters

Consider the connection of two two-port networks in Figure 10.11. Since the input ports of the two networks are connected in parallel ($V_1^a = V_1^b = V_1$, $I_1^a + I_1^b = I_1$) and their output ports are connected in series ($V_2^a + V_2^b = V_2$, $I_2^a = I_2^b = I_2$), it is called a *parallel-series connection* and its overall g -parameter matrix equals the sum of the g -parameter matrices of the two networks:

$$\begin{aligned} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix} &= \begin{bmatrix} I_1^a \\ V_2^a \end{bmatrix} + \begin{bmatrix} I_1^b \\ V_2^b \end{bmatrix} = \begin{bmatrix} g_{11}^a & g_{12}^a \\ g_{21}^a & g_{22}^a \end{bmatrix} \begin{bmatrix} V_1^a \\ I_2^a \end{bmatrix} + \begin{bmatrix} g_{11}^b & g_{12}^b \\ g_{21}^b & g_{22}^b \end{bmatrix} \begin{bmatrix} V_1^b \\ I_2^b \end{bmatrix} \\ &= \begin{bmatrix} g_{11}^a + g_{11}^b & g_{12}^a + g_{12}^b \\ g_{21}^a + g_{21}^b & g_{22}^a + g_{22}^b \end{bmatrix} \begin{bmatrix} V_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} g_{11} & g_{12} \\ g_{21} & g_{22} \end{bmatrix} \begin{bmatrix} V_1 \\ I_2 \end{bmatrix} \end{aligned} \quad (10.27)$$

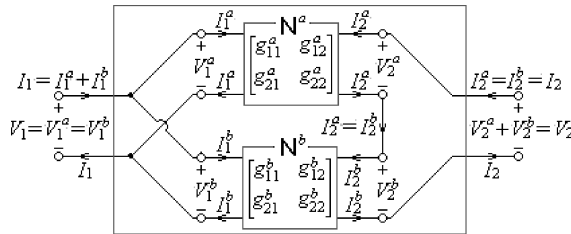


Figure 10.11 Parallel-series connection of two two-port networks

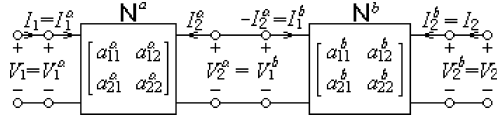


Figure 10.12 Cascade connection of two two-port networks

10.4.5 Cascade Connection and a -Parameters

Consider the *cascade* connection of two two-port networks in Figure 10.12, where the output port of a network N^a is connected to the input port of another network N^b . Since the voltage and current at the input port of the network N^b are the same as those at the output port of the network N^a ($-I_2^a = I_1^b$, $V_2^a = V_1^b$), its overall a -parameter matrix equals the product of the a -parameter matrices of the two networks:

$$\begin{aligned} \begin{bmatrix} V_1 \\ I_1 \end{bmatrix} &= \begin{bmatrix} V_1^a \\ I_1^a \end{bmatrix} = \begin{bmatrix} a_{11}^a & a_{12}^a \\ a_{21}^a & a_{22}^a \end{bmatrix} \begin{bmatrix} V_2^a \\ -I_2^a \end{bmatrix} = \begin{bmatrix} a_{11}^a & a_{12}^a \\ a_{21}^a & a_{22}^a \end{bmatrix} \begin{bmatrix} V_1^b \\ I_1^b \end{bmatrix} \\ &= \begin{bmatrix} a_{11}^a & a_{12}^a \\ a_{21}^a & a_{22}^a \end{bmatrix} \begin{bmatrix} a_{11}^b & a_{12}^b \\ a_{21}^b & a_{22}^b \end{bmatrix} \begin{bmatrix} V_2^b \\ -I_2^b \end{bmatrix} = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix} \end{aligned} \quad (10.28)$$

10.4.6 Curse of the Port Condition (Current Requirement)

In order for the overall parameters of an interconnection of two-port networks to be obtained by one of the above formulas (10.24 to 10.28), each port of the two-port networks should satisfy the *port condition* requiring that the current flowing into the network through one terminal should equal the current flowing out of the network through the other terminal.

Although the port condition is satisfied for each of two two-port networks alone, it may not be satisfied after the two two-port networks are interconnected. Let us see the following example.

(Example 10.7) The Curse of the Port Condition Violated by Interconnection

- (a) Consider the two-port network in Figure 10.13(a). Its y -parameters can be found by applying two current sources I_1 and I_2 to the ports 1 and 2, respectively, obtaining the expression of V_1 and

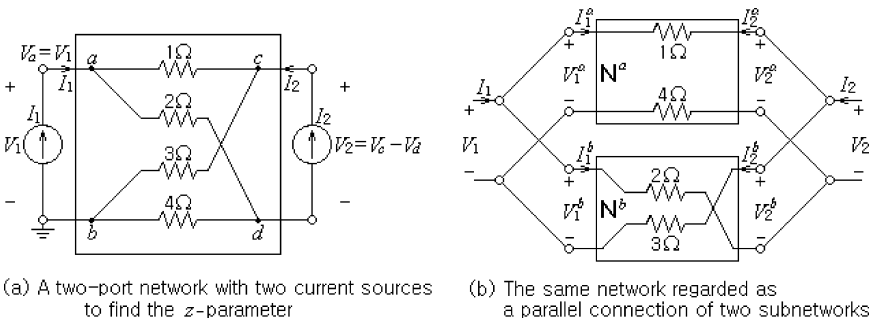


Figure 10.13 An example of two-port networks violating the port condition after connection

$V_2 = V_c - V_d$ in terms of I_1 and I_2 to get the z -parameters, and then taking the inverse of the z -parameter matrix to get the y -parameter matrix. According to this scheme, the node equation is written and MATLAB is used to solve it as

$$\begin{bmatrix} 1+1/2 & -1 & -1/2 \\ -1 & 1+1/3 & 0 \\ -1/2 & 0 & 1/2+1/4 \end{bmatrix} \begin{bmatrix} V_1 \\ V_c \\ V_d \end{bmatrix} = \begin{bmatrix} I_1 \\ I_2 \\ -I_2 \end{bmatrix} \quad (\text{E10.7.1})$$

```
>>Y=[1+1/2 -1 -1/2;-1 1+1/3 0;-1/2 0 1/2+1/4]; Y^-1
ans=[12/5 9/5 8/5
     9/5 21/10 6/5
     8/5 6/5 12/5]
```

$$\begin{bmatrix} V_1 \\ V_c \\ V_d \end{bmatrix} = \begin{bmatrix} 12/5 & 9/5 & 8/5 \\ 9/5 & 21/10 & 6/5 \\ 8/5 & 6/5 & 12/5 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \\ -I_2 \end{bmatrix} = \begin{bmatrix} (12/5)I_1 + (1/5)I_2 \\ (9/5)I_1 + (9/10)I_2 \\ (8/5)I_1 - (6/5)I_2 \end{bmatrix}$$

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} V_1 \\ V_c - V_d \end{bmatrix} = \begin{bmatrix} 12/5 & 1/5 \\ 1/5 & 21/10 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \quad (\text{E10.7.2})$$

Thus the z -parameter matrix is obtained and its inverse taken to get the y -parameter matrix as

$$z = \begin{bmatrix} 12/5 & 1/5 \\ 1/5 & 21/10 \end{bmatrix}; \quad y = z^{-1} = \frac{50}{12 \times 21 - 2} \begin{bmatrix} 21/10 & -1/5 \\ -1/5 & 12/5 \end{bmatrix} = \begin{bmatrix} 21/50 & -1/25 \\ -1/25 & 12/25 \end{bmatrix} \quad (\text{E10.7.3})$$

Note. You can use the following MATLAB program `cir10_04_06.m` to get this result.

```
%cir10_04_06.m: Symbolic computation to solve a 3x3 node equation
clear
format rat
Y=[1+1/2 -1 -1/2;-1 1+1/3 0;-1/2 0 1/2+1/4]; % Node admittance matrix
Y^-1
V1=Y\[1 0 0]'; V2=Y\[0 1 -1]'; % Node Voltage vectors for I1 and I2
z=[V1(1) V2(1); V1(2)-V1(3) V2(2)-V2(3)]; % the z-parameter matrix
y=z^-1, format short % the y-parameter matrix
```

Now, regarding the two-port network as the parallel interconnection of two subnetworks as depicted in Figure 10.13(b), let us add the y -parameter matrices of the subnetworks to get the overall y -parameter matrix as

$$\begin{bmatrix} I_1^a \\ I_2^a \end{bmatrix} = \begin{bmatrix} 1/(1+4) & -1/(1+4) \\ -1/(1+4) & 1/(1+4) \end{bmatrix} \begin{bmatrix} V_1^a \\ V_2^a \end{bmatrix}, \quad \begin{bmatrix} I_1^b \\ I_2^b \end{bmatrix} = \begin{bmatrix} 1/(2+3) & 1/(2+3) \\ 1/(2+3) & 1/(2+3) \end{bmatrix} \begin{bmatrix} V_1^b \\ V_2^b \end{bmatrix} \quad (\text{E10.7.4})$$

$$y \stackrel{?}{=} y^a + y^b = \begin{bmatrix} 1/5 & -1/5 \\ -1/5 & 1/5 \end{bmatrix} + \begin{bmatrix} 1/5 & 1/5 \\ 1/5 & 1/5 \end{bmatrix} = \begin{bmatrix} 2/5 & 0 \\ 0 & 2/5 \end{bmatrix} \quad (\text{E10.7.5})$$

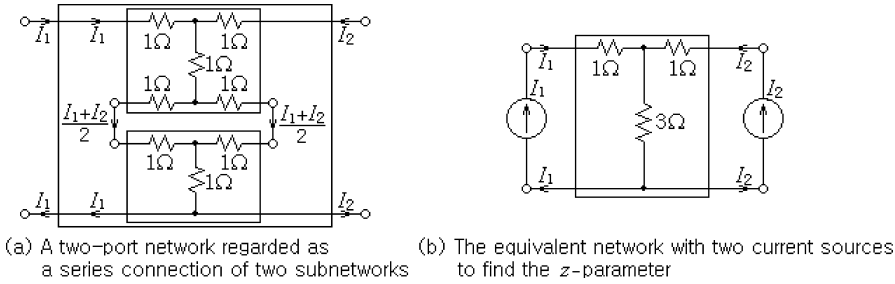


Figure 10.14 Examples of two-port networks violating the port condition after connection

To our surprise, this does not agree with the previous result (E10.7.3). What is happening? This is the case where the subnetworks violate the port condition, which can be verified by finding the currents flowing through the resistors $1\ \Omega$ and $4\ \Omega$ (belonging to subnetwork N^a):

$$I_{1\Omega} = \frac{V_1 - V_c}{1} \stackrel{(E10.7.2)}{=} \frac{3}{5}I_1 - \frac{7}{10}I_2 \neq I_{4\Omega} = \frac{V_d - 0}{4} \stackrel{(E10.7.2)}{=} \frac{2}{5}I_1 - \frac{3}{10}I_2 \quad (E10.7.6)$$

- (b) Consider the two-port network in Figure 10.14(a). First, regarding the network as the series connection of two subnetworks, the z -parameter matrices of the two subnetworks are added to get its z -parameter matrix. Second, the whole network could be simplified as depicted in Figure 10.14(b) and its z -parameter matrix found as

$$z^a + z^b = \begin{bmatrix} 1 + 1 + 1 & 1 \\ 1 & 1 + 1 + 1 \end{bmatrix} + \begin{bmatrix} 1 + 1 & 1 \\ 1 & 1 + 1 \end{bmatrix} \neq z = \begin{bmatrix} 4 & 3 \\ 3 & 4 \end{bmatrix} \quad (E10.7.7)$$

Why are these results different? It is because the two subnetworks in the series interconnection (Figure 10.14(a)) do not satisfy the port condition; i.e. the currents flowing into each subnetwork through one terminal of the ports are not the same as the currents flowing out of it through the other terminal of the ports ($I_1 \neq (I_1 + I_2)/2 \neq I_2$) as long as $I_1 \neq I_2$.

Note. This example implies that we should be careful in regarding a two-port network as an interconnection of subnetworks for the purpose of finding its parameters more easily.

(Example 10.8) Examples of Interconnection Not Irritating the Port Condition

- (a) Consider the two-port network in Figure 10.15(a). Its z -parameters will be found by regarding it as a series connection of two subnetworks as depicted in Figure 10.15(b) and adding the z -parameter

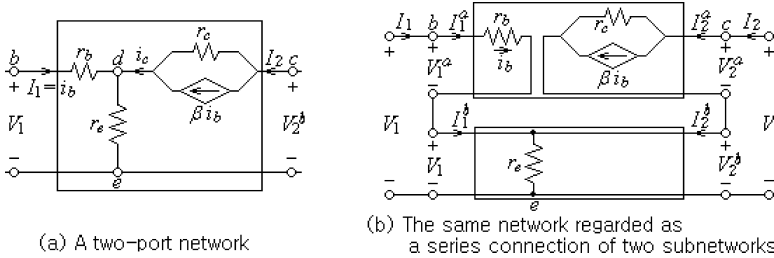


Figure 10.15 A circuit model for a common-emitter transistor

matrices of the two subnetworks (see Equation (10.24)). Noting that $I_1^a = i_b$, the input/output port voltages are expressed in terms of the input/output port currents as

$$\begin{bmatrix} V_1^a \\ V_2^a \end{bmatrix} = \begin{bmatrix} r_b & 0 \\ -\beta r_c & r_c \end{bmatrix} \begin{bmatrix} I_1^a \\ I_2^a \end{bmatrix}, \quad \begin{bmatrix} V_1^b \\ V_2^b \end{bmatrix} = \begin{bmatrix} r_e & r_e \\ r_e & r_e \end{bmatrix} \begin{bmatrix} I_1^b \\ I_2^b \end{bmatrix} \quad (\text{E10.8.1})$$

Thus the two z -parameter matrices are added to get the overall z -parameter matrix as

$$z = z^a + z^b = \begin{bmatrix} r_b & 0 \\ -\beta r_c & r_c \end{bmatrix} + \begin{bmatrix} r_e & r_e \\ r_e & r_e \end{bmatrix} = \begin{bmatrix} r_b + r_e & r_e \\ -\beta r_c + r_e & r_c + r_e \end{bmatrix} \quad (\text{E10.8.2})$$

In order to check if this is correct, this z -parameter is converted into the h -parameter by using the MATLAB routine `port_conversion()` listed with Table 10.1 in Section 10.2:

```
>>syms rb rc re beta
>>z=[ rb+re re; re-beta*rc rc+re];
>>h= port_conversion(z,'z2h') %convert z-parameter to h-parameter
h= [(rb+rc+rb*re+re*rc+re*beta*rc)/(rc+re), re/(rc+re)]
    [ (-re+beta*rc)/(rc+re), 1/(rc+re) ]
```

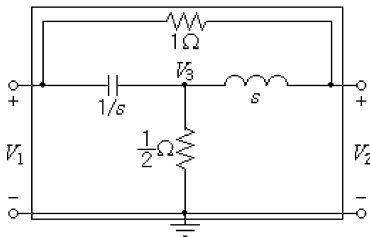
This agrees with Equation (E10.5.4), which was obtained in Example 10.5.

- (b) Consider the two-port network in Figure 10.16(a). Its y -parameters will be found by regarding it as a parallel connection of two subnetworks as depicted in Figure 10.16(b) and adding the y -parameter matrices of the two subnetworks (see Equation (10.25)). From Equation (E10.3.5), the y -parameter matrix of the upper subnetwork can be obtained as

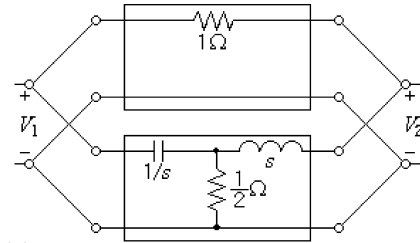
$$y^a = \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad (\text{E10.8.3})$$

and that of the lower subnetwork can be obtained by taking the inverse of the z -parameter matrix as

$$\begin{aligned} y^b = [z^b]^{-1} &= \begin{bmatrix} 1/s + 1/2 & 1/2 \\ 1/2 & s + 1/2 \end{bmatrix}^{-1} = \frac{2}{2 + s + 1/s} \begin{bmatrix} s + 1/2 & -1/2 \\ -1/2 & 1/s + 1/2 \end{bmatrix} \\ &= \frac{1}{(s + 1)^2} \begin{bmatrix} s(2s + 1) & -s \\ -s & s + 2 \end{bmatrix} \end{aligned} \quad (\text{E10.8.4})$$



(a) A two-port network



(b) The same network regarded as a series connection of two subnetworks

Figure 10.16 A two-port network and its parallel decomposition

These two y -parameter matrices are added to get the y -parameters of the overall network as

$$y = y^a + y^b = \begin{bmatrix} \frac{3s^2 + 3s + 1}{(s+1)^2} & -\frac{s^2 + 3s + 1}{(s+1)^2} \\ -\frac{s^2 + 3s + 1}{(s+1)^2} & \frac{s^2 + 3s + 3}{(s+1)^2} \end{bmatrix} \quad (\text{E10.8.5})$$

This agrees with Equation (E10.6.2), which is the result obtained in Example 10.6.

- (c) Consider the two-port network in Figure 10.4(c). Regarding it as a cascade connection of the two two-port networks depicted in Figures 10.4(a) and (b) and using Equation (10.28), the a -parameter matrices of the two networks, i.e. Equations (E10.3.1) and (E10.3.2), are multiplied to get its overall a -parameters as

$$\begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ Y_a & 1 \end{bmatrix} \begin{bmatrix} 1 & Z_b \\ 0 & 1 \end{bmatrix} = \begin{bmatrix} 1 & Z_b \\ Y_a & 1 + Y_a Z_b \end{bmatrix} \quad (\text{E10.8.6})$$

This agrees with Equation (E10.3.3), which is the result obtained in Example 10.3.

10.5 Two-Port Networks Having Source/Load

In this section we will explore a systematic way of finding the following properties for a two-port network connected with a source and a load, as depicted in Figure 10.17(a):

1. Input impedance
2. Current gain $-I_2/I_1$
3. Voltage gain V_2/V_1 and V_2/V_s
4. Thevenin equivalent circuit seen from the output port

As an example, an expression will be found for the above properties in terms of the z -parameters. With this objective in mind, the two-port network is regarded as a series connection of two subnetworks as

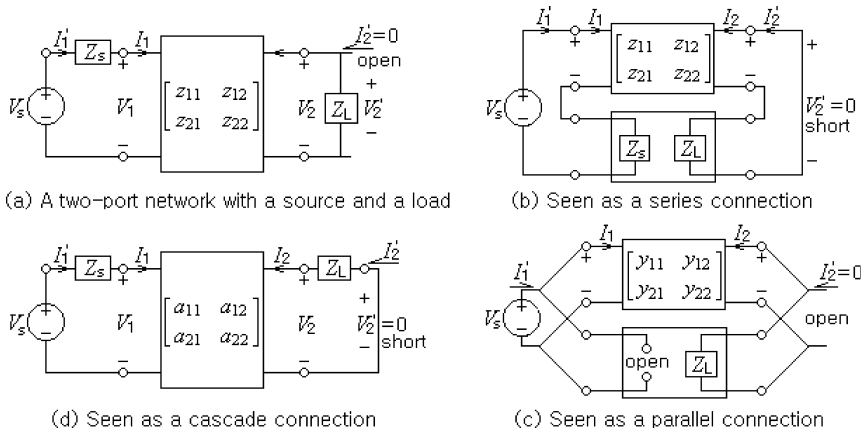


Figure 10.17 A two-port network with a source and a load and its equivalent circuits

depicted in Figure 10.17(b) and the voltage–current relationships of the two-port network itself and the overall network including the source/load are written as

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \quad (10.29)$$

$$\begin{bmatrix} V_s \\ V'_2 \end{bmatrix} = \begin{bmatrix} z_{11} + Z_s & z_{12} \\ z_{21} & z_{22} + Z_L \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \quad (10.30)$$

From Equation (10.30-2) with $V'_2 = 0$,

$$V'_2 = z_{21}I_1 + (z_{22} + Z_L)I_2 = 0, \quad \frac{-I_2}{I_1} = \frac{z_{21}}{z_{22} + Z_L} \text{ (current gain)} \quad (10.31)$$

Substituting this into Equation (10.29-1) yields the input impedance of the network including the load impedance as

$$\begin{aligned} V_1 &\stackrel{(10.29-1)}{=} z_{11}I_1 + z_{12}I_2 \stackrel{(10.31)}{=} z_{11}I_1 - z_{12} \frac{z_{21}}{z_{22} + Z_L} I_1 \\ Z_{in} = \frac{V_1}{I_1} &= z_{11} - \frac{z_{12}z_{21}}{z_{22} + Z_L} = \frac{\Delta z + z_{11}Z_L}{z_{22} + Z_L} \text{ (input impedance)} \end{aligned} \quad (10.32)$$

Substituting Equation (10.31) into Equation (10.30-1) yields the overall input impedance of the network including the source/load impedances as

$$\begin{aligned} V_s &\stackrel{(10.30-1)}{=} (z_{11} + Z_s)I_1 + z_{12}I_2 \stackrel{(10.31)}{=} (z_{11} + Z_s)I_1 - z_{12} \frac{z_{21}}{z_{22} + Z_L} I_1 \\ Z'_{in} = \frac{V_s}{I_1} &= z_{11} + Z_s - \frac{z_{12}z_{21}}{z_{22} + Z_L} = Z_s + Z_{in} \text{ (overall input impedance)} \end{aligned} \quad (10.33)$$

Equation (10.31) can be substituted into Equations (10.29) and (10.30-1) to get

$$\begin{aligned} V_2 &\stackrel{(10.29-2)}{=} z_{21}I_1 + z_{22}I_2 \stackrel{(10.31)}{=} \left(z_{21} - \frac{z_{22}z_{21}}{z_{22} + Z_L} \right) I_1 = \frac{z_{21}Z_L}{z_{22} + Z_L} I_1 \\ V_1 &\stackrel{(10.29-1)}{=} z_{11}I_1 + z_{12}I_2 \stackrel{(10.31)}{=} \left(z_{11} - \frac{z_{12}z_{21}}{z_{22} + Z_L} \right) I_1 = \frac{\Delta z + z_{11}Z_L}{z_{22} + Z_L} I_1 \\ V_s &\stackrel{(10.30-1)}{=} (z_{11} + Z_s)I_1 + z_{12}I_2 \stackrel{(10.31)}{=} \left(z_{11} + Z_s - \frac{z_{12}z_{21}}{z_{22} + Z_L} \right) I_1 \end{aligned}$$

which yields the voltage gains

$$\frac{V_2}{V_1} = \frac{z_{21}Z_L}{\Delta z + z_{11}Z_L} \text{ (voltage gain)} \quad (10.34)$$

$$\frac{V_2}{V_s} = \frac{z_{21}Z_L}{(z_{11} + Z_s)(z_{22} + Z_L) - z_{12}z_{21}} \text{ (overall voltage gain)} \quad (10.35)$$

Now, in order to find the Thevenin equivalent circuit of the network seen from the load (at the output port), Equation (10.30-1) can be written as

$$V_s = (z_{11} + Z_s)I_1 + z_{12}I_2; \quad I_1 = \frac{1}{z_{11} + Z_s} V_s - \frac{z_{12}}{z_{11} + Z_s} I_2$$

and substituting this into Equation (10.29-2) yields

$$\begin{aligned}
 V_2 &= z_{21}I_1 + z_{22}I_2 = z_{21}\left(\frac{1}{z_{11} + Z_s}V_s - \frac{z_{12}}{z_{11} + Z_s}I_2\right) + z_{22}I_2 \\
 &= \frac{z_{21}}{z_{11} + Z_s}V_s + \left(z_{22} - \frac{z_{12}z_{21}}{z_{11} + Z_s}\right)I_2 \stackrel{(2.14)}{=} V_{Th} + Z_{Th}I_2 \\
 V_{Th} &= \frac{z_{21}}{z_{11} + Z_s}V_s, \quad Z_{Th} = z_{22} - \frac{z_{12}z_{21}}{z_{11} + Z_s} = \frac{\Delta z + z_{22}Z_s}{z_{11} + Z_s}
 \end{aligned} \tag{10.36}$$

Note. It is no wonder that this output impedance Z_{Th} and the input impedance Z_{in} (Equation (10.32)) are symmetric.

Now, in order to derive a more systematic way for the formulas to obtain the input/output impedances and the voltage gain of the overall network (having the source/load) from other parameters than the z -parameters, let us find the expression of the input–output characteristics of the two-port network (with no source/load) in terms of the various parameters.

10.5.1 Input Impedance

In order to get the input impedance of a two-port network, the output port is opened to cut off any load effect so that $I_2 = 0$.

(1) In terms of the z -parameters

$$\begin{aligned}
 V_1 &\stackrel{(10.1-1)}{=} z_{11}I_1 + z_{12}I_2 \Big|_{I_2=0} = z_{11}I_1 \\
 Z_{in} &= \frac{V_1}{I_1} \Big|_{I_2=0} = z_{11}
 \end{aligned} \tag{10.37}$$

(2) In terms of the h -parameters

$$\begin{aligned}
 \begin{bmatrix} V_1 \\ I_2 \end{bmatrix} &\stackrel{(10.5)}{=} \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix} \quad \text{with } I_2 = 0 \\
 I_2 &\stackrel{(10.5-2)}{=} h_{21}I_1 + h_{22}V_2 = 0; \quad V_2 = -\frac{h_{21}}{h_{22}}I_1 \\
 V_1 &\stackrel{(10.5-1)}{=} h_{11}I_1 + h_{12}V_2 = h_{11}I_1 - h_{12}\frac{h_{21}}{h_{22}}I_1 = \frac{\Delta h}{h_{22}}I_1 \\
 Z_{in} &= \frac{V_1}{I_1} \Big|_{I_2=0} = \frac{\Delta h}{h_{22}}
 \end{aligned} \tag{10.38}$$

(3) In terms of the a -parameters

$$\begin{aligned}
 \begin{bmatrix} V_1 \\ I_1 \end{bmatrix} &\stackrel{(10.3)}{=} \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix} \Big|_{I_2=0} = \begin{bmatrix} a_{11} \\ a_{21} \end{bmatrix} V_2 \\
 Z_{in} &= \frac{V_1}{I_1} \Big|_{I_2=0} = \frac{a_{11}}{a_{21}}
 \end{aligned} \tag{10.39}$$

10.5.2 Voltage Gain

In order to get the *voltage gain*, i.e. the ratio of the output voltage to the input voltage of a two-port network, the output port is opened to cut off any load effect so that $I_2 = 0$.

(1) In terms of the z -parameters

$$\begin{aligned} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} &\stackrel{(10.1)}{=} \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \Big|_{I_2=0} = \begin{bmatrix} z_{11} \\ z_{21} \end{bmatrix} I_1 \\ A_v &= \frac{V_2}{V_1} \Big|_{I_2=0} = \frac{z_{21}}{z_{11}} \end{aligned} \quad (10.40)$$

(2) In terms of the h -parameters

$$\begin{aligned} I_2 &\stackrel{(10.5-2)}{=} h_{21}I_1 + h_{22}V_2 = 0; \quad I_1 = -\frac{h_{22}}{h_{21}}V_2 \\ V_1 &\stackrel{(10.5-1)}{=} h_{11}I_1 + h_{12}V_2 = h_{12}V_2 - h_{11}\frac{h_{22}}{h_{21}}V_2 = -\frac{\Delta h}{h_{21}}V_2 \\ A_v &= \frac{V_2}{V_1} \Big|_{I_2=0} = -\frac{h_{21}}{\Delta h} \end{aligned} \quad (10.41)$$

(3) In terms of the a -parameters

$$V_1 \stackrel{(10.3-1)}{=} a_{11}V_2 - a_{12}I_2 \Big|_{I_2=0} = a_{11}V_2; \quad A_v = \frac{V_2}{V_1} \Big|_{I_2=0} = \frac{1}{a_{11}} \quad (10.42)$$

10.5.3 Current Gain

In order to get the *current gain*, i.e. the ratio of the output current to the input current of a two-port network, the output port is shorted to make the output current unblocked so that $V_2 = 0$.

(1) In terms of the z -parameters

$$V_2 \stackrel{(10.1-2)}{=} z_{21}I_1 + z_{22}I_2 = 0; \quad A_i = \frac{-I_2}{I_1} \Big|_{V_2=0} = \frac{z_{21}}{z_{22}} \quad (10.43)$$

(2) In terms of the h -parameters

$$I_2 \stackrel{(10.5-2)}{=} h_{21}I_1 + h_{22}V_2 \Big|_{V_2=0} = h_{21}I_1; \quad A_i = \frac{-I_2}{I_1} \Big|_{V_2=0} = -h_{21} \quad (10.44)$$

(3) In terms of the a -parameters

$$I_1 \stackrel{(10.3-2)}{=} a_{21}V_2 - a_{22}I_2 \Big|_{V_2=0} = -a_{22}I_2; \quad A_i = \frac{-I_2}{I_1} \Big|_{V_2=0} = \frac{1}{a_{22}} \quad (10.45)$$

10.5.4 (Thevenin) Equivalent Impedance Seen from the Output

In order to get the output impedance of a two-port network, the input port is shorted to remove the effect of the source so that $V_1 = 0$.

(1) In terms of the z -parameters

$$\begin{aligned} V_1 &\stackrel{(10.1-1)}{=} z_{11}I_1 + z_{12}I_2 = 0; & I_1 &= -\frac{z_{12}}{z_{11}}I_2 \\ V_2 &\stackrel{(10.1-2)}{=} z_{21}I_1 + z_{22}I_2 = \left(z_{22} - z_{21}\frac{z_{12}}{z_{11}}\right)I_2 \\ Z_{\text{Th}} &= \frac{V_2}{I_2} \Big|_{V_1=0} = \frac{\Delta z}{z_{11}} \end{aligned} \quad (10.46)$$

(2) In terms of the h -parameters

$$\begin{aligned} V_1 &\stackrel{(10.5-1)}{=} h_{11}I_1 + h_{12}V_2 = 0; & I_1 &= -\frac{h_{12}}{h_{11}}V_2 \\ I_2 &\stackrel{(10.5-2)}{=} h_{21}I_1 + h_{22}V_2 = \left(h_{22} - h_{21}\frac{h_{12}}{h_{11}}\right)V_2 \\ Z_{\text{Th}} &= \frac{V_2}{I_2} \Big|_{V_1=0} = \frac{h_{11}}{\Delta h} \end{aligned} \quad (10.47)$$

(3) In terms of the a -parameters

$$V_1 \stackrel{(10.3-1)}{=} a_{11}V_2 - a_{12}I_2 = 0; \quad Z_{\text{Th}} = \frac{V_2}{I_2} \Big|_{V_1=0} = \frac{a_{12}}{a_{11}} \quad (10.48)$$

10.5.5 (Thevenin) Equivalent Source Seen from the Output

In order to get the Thevenin equivalent voltage source of a two-port network seen from the output port, the output port is opened to cut off any load effect so that $I_2 = 0$.

(1) In terms of the z -parameters

$$V_{\text{Th}} = A_v V_1 \stackrel{(10.40)}{=} \frac{z_{21}}{z_{11}} V_1 \quad (10.49)$$

(2) In terms of the h -parameters

$$V_{\text{Th}} = A_v V_1 \stackrel{(10.41)}{=} -\frac{h_{21}}{\Delta h} V_1 \quad (10.50)$$

(3) In terms of the a -parameters

$$V_{\text{Th}} = A_v V_1 \stackrel{(10.42)}{=} \frac{1}{a_{11}} V_1 \quad (10.51)$$

We might also derive the formulas for obtaining the input/output impedances and the voltage gain of the two-port network (with no source/load) in terms of y , g and b -parameters, which are listed in Table 10.3.

Table 10.3 The properties of a two-port network in terms of its parameters

	z	y	a	b	h	g
Z_{in} (port 2 open)	(10.37) z_{11}	$\frac{y_{22}}{\Delta y}$	(10.39) $\frac{a_{11}}{a_{21}}$	$\frac{b_{22}}{b_{21}}$	(10.38) $\frac{\Delta h}{h_{22}}$	$\frac{1}{g_{11}}$
$A_v = V_2/V_1$ (port 2 open)	(10.40) $\frac{z_{21}}{z_{11}}$	$-\frac{y_{21}}{y_{22}}$	(10.42) $\frac{1}{a_{11}}$	$\frac{\Delta b}{b_{22}}$	(10.41) $-\frac{h_{21}}{\Delta h}$	g_{21}
$A_i = -I_2/I_1$ (port 2 short)	(10.43) $\frac{z_{21}}{z_{22}}$	$-\frac{y_{21}}{y_{11}}$	(10.45) $\frac{1}{a_{22}}$	$\frac{\Delta b}{b_{11}}$	(10.44) $-h_{21}$	$\frac{g_{21}}{\Delta g}$
$Z_{\text{out}} = Z_{\text{Th}}$ (port 1 short)	(10.46) $\frac{\Delta z}{z_{11}}$	$\frac{1}{y_{22}}$	(10.48) $\frac{a_{12}}{a_{11}}$	$\frac{b_{12}}{b_{22}}$	(10.47) $\frac{h_{11}}{\Delta h}$	g_{22}

10.5.6 The Parameters of an Overall Two-Port Network

In order to use the formulas for the input/output impedances and the voltage/current gains of the two-port network (with no source/load) in obtaining those properties of the overall two-port network containing the source/load, the overall parameters are required.

10.5.6.1 Overall Parameters for Input Impedance and Voltage Gain

The overall network is regarded as a parallel or series-parallel or cascade interconnection of two subnetworks depending on the base parameter. Considering that the overall parameters will be used to find the input impedance and the voltage gain, the source impedance is excluded and the output port of the overall network is opened (see Sections 10.5.1 and 10.5.2 and Figures 10.17(a) and (c)).

(1) The overall z and y -parameters

To obtain the overall z and y -parameters, the overall network is regarded as a parallel interconnection (with the output port open), as depicted in Figure 10.18, and Equation (10.25) is used:

$$\begin{aligned}
 \begin{bmatrix} y'_{11} & y'_{12} \\ y'_{21} & y'_{22} \end{bmatrix} &= \begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & Y_L \end{bmatrix} = \begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} + Y_L \end{bmatrix} \\
 \begin{bmatrix} z'_{11} & z'_{12} \\ z'_{21} & z'_{22} \end{bmatrix} &= \begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} + Y_L \end{bmatrix}^{-1} = \frac{1}{\Delta y + y_{11}Y_L} \begin{bmatrix} y_{22} + Y_L & -y_{12} \\ -y_{21} & y_{11} \end{bmatrix} \\
 &= \frac{1}{1/\Delta z + [z_{22}/(\Delta z Z_L)]} \begin{bmatrix} z_{11}/\Delta z + 1/Z_L & z_{12}/\Delta z \\ z_{21}/\Delta z & z_{22}/\Delta z \end{bmatrix} \\
 &= \frac{1}{z_{22} + Z_L} \begin{bmatrix} z_{11}Z_L + \Delta z & z_{12}Z_L \\ z_{21}Z_L & z_{22}Z_L \end{bmatrix} \quad (10.52)
 \end{aligned}$$

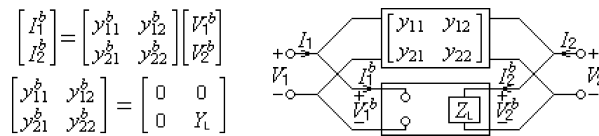


Figure 10.18 The y -parameter of a two-port network including a load impedance, regarded as a parallel interconnection with the overall output port open

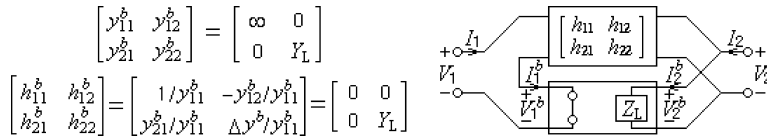


Figure 10.19 The h -parameter of a two-port network including a load impedance, regarded as a series-parallel interconnection with the overall output port open

(2) The overall h and g -parameters

To obtain the overall h and g -parameters, the overall network is regarded as a series-parallel interconnection (with the output port open), as depicted in Figure 10.19, and Equation (10.26) is used:

$$\begin{aligned} \begin{bmatrix} h'_{11} & h'_{12} \\ h'_{21} & h'_{22} \end{bmatrix} &= \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & Y_L \end{bmatrix} = \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} + Y_L \end{bmatrix} \\ \begin{bmatrix} g'_{11} & g'_{12} \\ g'_{21} & g'_{22} \end{bmatrix} &= \frac{1}{g_{22} + Z_L} \begin{bmatrix} g_{11}Z_L + \Delta g & g_{12}Z_L \\ g_{21}Z_L & g_{22}Z_L \end{bmatrix} \end{aligned} \quad (10.53)$$

(3) The overall a and b -parameters

To obtain the overall a and b -parameters, the overall network is regarded as a cascade interconnection (with the output port open), as depicted in Figure 10.20, and Equation (10.28) is used:

$$\begin{aligned} \begin{bmatrix} a'_{11} & a'_{12} \\ a'_{21} & a'_{22} \end{bmatrix} &= \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} \begin{bmatrix} 1 & 0 \\ Y_L & 1 \end{bmatrix} = \begin{bmatrix} a_{11} + a_{12}Y_L & a_{12} \\ a_{21} + a_{22}Y_L & a_{22} \end{bmatrix} \\ \begin{bmatrix} b'_{11} & b'_{12} \\ b'_{21} & b'_{22} \end{bmatrix} &= \begin{bmatrix} 1 & 0 \\ Y_L & 1 \end{bmatrix} \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix} = \begin{bmatrix} b_{11} & b_{12} \\ b_{11}Y_L + b_{21} & b_{12}Y_L + b_{22} \end{bmatrix} \end{aligned} \quad (10.54)$$

Now, these overall parameters can be substituted into the corresponding formulas (listed in Table 10.3 in Section 10.5.5) to get the input impedance and the voltage gain of the two-port network with the load in Figure 10.17(a). For example, Equation (10.32) for the overall input impedance can be obtained by substituting $z'_{11} = (\Delta z + z_{11}Z_L)/(z_{22} + Z_L)$ (from Equation (10.52)) for z_{11} into Equation (10.37). The results are listed in Table 10.4.

Note. The Thevenin equivalent voltage seen from the output port can be obtained as follows:

$$V_{Th} = V_2|_{I_2=0} = \frac{V_1}{V_s} \frac{V_2}{V_1} V_s = \frac{Z_{in}}{Z_s + Z_{in}} A_v V_s \quad (10.55)$$

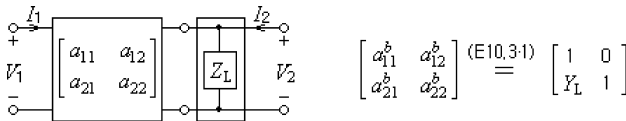


Figure 10.20 The a -parameter of a two-port network including a load impedance, regarded as a cascade interconnection with the overall output port open

Table 10.4 The properties of a two-port network having a source/load in terms of its parameters

	z	y	a	b	h	g
Z_{in} (port 2 open)	(10.32) $\frac{\Delta z Y_L + z_{11}}{z_{22} Y_L + 1}$	$\frac{y_{22} + Y_L}{\Delta y + y_{11} Y_L}$	$\frac{a_{11} + a_{12} Y_L}{a_{21} + a_{22} Y_L}$	$\frac{b_{22} + b_{12} Y_L}{b_{21} + b_{11} Y_L}$	$\frac{\Delta h + h_{11} Y_L}{h_{22} + Y_L}$	$\frac{g_{22} Y_L + 1}{\Delta g Y_L + g_{11}}$
$A_v = V_2/V_1$ (port 2 open)	(10.34) $\frac{z_{21}}{\Delta z Y_L + z_{11}}$	$\frac{-y_{21}}{y_{22} + Y_L}$	$\frac{1}{a_{11} + a_{12} Y_L}$	$\frac{\Delta b}{b_{22} + b_{12} Y_L}$	$\frac{-h_{21}}{\Delta h + h_{11} Y_L}$	$\frac{g_{21}}{g_{22} Y_L + 1}$
$A_i = -I_2/I_1$ (port 2 short)	(10.31) $\frac{z_{21}}{z_{22} + Z_L}$	$\frac{-y_{21}}{\Delta y Z_L + y_{11}}$	$\frac{1}{a_{22} + a_{21} Z_L}$	$\frac{\Delta b}{b_{11} + b_{21} Z_L}$	$\frac{-h_{21}}{h_{22} Z_L + 1}$	$\frac{g_{21}}{\Delta g + g_{11} Z_L}$
$Z_{out} = Z_{Th}$ (port 1 short)	(10.36) $\frac{\Delta z + z_{22} Z_s}{z_{11} + Z_s}$	$\frac{y_{11} + Y_s}{\Delta y + y_{22} Y_s}$	$\frac{a_{12} + a_{22} Z_s}{a_{11} + a_{21} Z_s}$	$\frac{b_{12} + b_{11} Z_s}{b_{22} + b_{21} Z_s}$	$\frac{h_{11} + Z_s}{\Delta h + h_{22} Z_s}$	$\frac{\Delta g + g_{22} Y_s}{g_{11} + Y_s}$

```

function p=port_property(P,Z,which) %Table 10.4
%Input: P= Base parameter matrix
%       Z= ZL(load impedance) or Zs(source impedance)
%       which= specifies the property and the base parameter
%       e.g., to find z-parameter-based current gain (Ai) ,
%       you put 'Aiz' as the 3rd input argument 'which' .
%Output: p=Zi (input impedance) /Av (voltage gain) /Ai (current gain) ,
%        Zo (output impedance) depending on the 1st two chs of which
%        based on the parameter specified by the 3rd character
%Copyright: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only
switch lower (which)
    case 'ziz' , Y=1/Z; p=(det (P)*Y+P(1,1)) / (P(2,2)*Y+1) ;
    case 'ziy' , Y=1/Z; p=(P(2,2)+Y) / (det (P)+P(1,1)*Y) ;
    case 'zia' , Y=1/Z; p=(P(1,1)+P(1,2)*Y) / (P(2,1)+P(2,2)*Y) ;
    case 'zib' , Y=1/Z; p=(P(2,2)+P(1,2)*Y) / (P(2,1)+P(1,1)*Y) ;
    case 'zih' , Y=1/Z; p=(det (P)+P(1,1)*Y) / (P(2,2)+Y) ;
    case 'zig' , Y=1/Z; p=(P(2,2)*Y+1) / (det (P)*Y+P(1,1)) ;
    case 'avz' , Y=1/Z; p=P(2,1) / (det (P)*Y+P(1,1)) ;
    case 'avy' , Y=1/Z; p=-P(2,1) / (P(2,2)+Y) ;
    case 'ava' , Y=1/Z; p=1/ (P(1,1)+P(1,2)*Y) ;
    case 'avb' , Y=1/Z; p=det (P) / (P(2,2)+P(1,2)*Y) ;
    case 'avh' , Y=1/Z; p=-P(2,1) / (det (P)+P(1,1)*Y) ;
    case 'avg' , Y=1/Z; p=P(2,1) / (P(2,2)*Y+1) ;
    case 'aiz' , p=P(2,1) / (P(2,2)+Z) ;
    case 'aiy' , p=-P(2,1) / (det (P)*Z+P(1,1)) ;
    case 'aia' , p=1/ (P(2,2)+P(2,1)*Z) ;
    case 'aib' , p=det (P) / (P(1,1)+P(2,1)*Z) ;
    case 'aih' , p=-P(2,1) / (P(2,2)*Z+1) ;
    case 'aig' , p=P(2,1) / (det (P)+P(1,1)*Z) ;
    case 'zoz' , p=(det (P)+P(2,2)*Z) / (P(1,1)+Z) ;
    case 'zoy' , p=(P(1,1)*Z+1) / (det (P)*Z+P(2,2)) ;
    case 'zoa' , p=(P(1,2)+P(2,2)*Z) / (P(1,1)+P(2,1)*Z) ;
    case 'zob' , p=(P(1,2)+P(1,1)*Z) / (P(2,2)+P(2,1)*Z) ;
    case 'zoh' , p=(P(1,1)+Z) / (det (P)+P(2,2)*Z) ;
    case 'zog' , p=(det (P)*Z+P(2,2)) / (P(1,1)*Z+1) ;
    otherwise error ('What do you want to do by using port_property()?' )
end

```

10.5.6.2 Overall Parameters for Current Gain

The overall network is regarded as a series or parallel-series or cascade interconnection of two subnetworks depending on the base parameter. Noting that the overall parameters will be used to find the current gain, the source impedance is excluded and the output port of the overall network is shorted (see Section 10.5.3 and Figures 10.17(b) and (d)).

(1) The overall z and y -parameters

To obtain the overall z and y -parameters, the overall network is regarded as a series interconnection (with the output port shorted), as depicted in Figure 10.21, and Equation (10.24) is used:

$$\begin{aligned} \begin{bmatrix} z'_{11} & z'_{12} \\ z'_{21} & z'_{22} \end{bmatrix} &= \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & Z_L \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} + Z_L \end{bmatrix} \\ \begin{bmatrix} y'_{11} & y'_{12} \\ y'_{21} & y'_{22} \end{bmatrix} &= \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} + Z_L \end{bmatrix}^{-1} = \frac{1}{y_{22} + Y_L} \begin{bmatrix} y_{11}Y_L + \Delta y & y_{12}Y_L \\ y_{21}Y_L & y_{22}Y_L \end{bmatrix} \end{aligned} \quad (10.56)$$

(2) The overall h and g -parameters

To obtain the overall h and g -parameters, the overall network is regarded as a parallel-series interconnection (with the output port shorted), as depicted in Figure 10.22, and Equation (10.27) is used:

$$\begin{aligned} \begin{bmatrix} g'_{11} & g'_{12} \\ g'_{21} & g'_{22} \end{bmatrix} &= \begin{bmatrix} g_{11} & g_{12} \\ g_{21} & g_{22} \end{bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & Z_L \end{bmatrix} = \begin{bmatrix} g_{11} & g_{12} \\ g_{21} & g_{22} + Z_L \end{bmatrix} \\ \begin{bmatrix} h'_{11} & h'_{12} \\ h'_{21} & h'_{22} \end{bmatrix} &= \frac{1}{h_{22} + Y_L} \begin{bmatrix} h_{11}Y_L + \Delta h & h_{12}Y_L \\ h_{21}Y_L & h_{22}Y_L \end{bmatrix} \end{aligned} \quad (10.57)$$

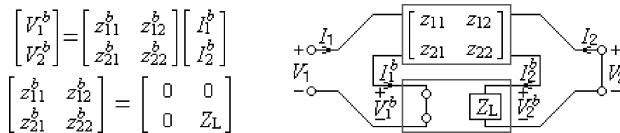


Figure 10.21 The z -parameter of a two-port network including a load impedance, regarded as a series interconnection with the overall output port shorted

$$\begin{aligned} \begin{bmatrix} z'_{11} & z'_{12} \\ z'_{21} & z'_{22} \end{bmatrix} &= \begin{bmatrix} \infty & 0 \\ 0 & Z_L \end{bmatrix} \\ \begin{bmatrix} g'_{11} & g'_{12} \\ g'_{21} & g'_{22} \end{bmatrix} &= \begin{bmatrix} 1/z'_{11} & -z'_{12}/z'_{11} \\ z'_{21}/z'_{11} & \Delta z'/z'_{11} \end{bmatrix} = \begin{bmatrix} 0 & 0 \\ 0 & Z_L \end{bmatrix} \end{aligned}$$

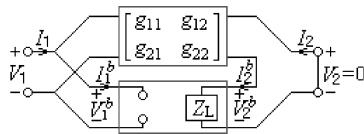


Figure 10.22 The g -parameter of a two-port network including a load impedance, regarded as a parallel-series interconnection with the overall output port shorted

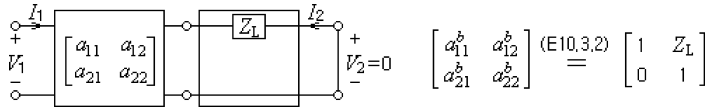


Figure 10.23 The a -parameter of a two-port network including a load impedance, regarded as a cascade interconnection with the overall output port shorted

(3) The overall a and b -parameters

To obtain the overall a and b -parameters, the overall network is regarded as a cascade interconnection (with the output port shorted), as depicted in Figure 10.23, and Equation (10.28) is used:

$$\begin{aligned} \begin{bmatrix} a'_{11} & a'_{12} \\ a'_{21} & a'_{22} \end{bmatrix} &= \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} \begin{bmatrix} 1 & Z_L \\ 0 & 1 \end{bmatrix} = \begin{bmatrix} a_{11} & a_{12} + a_{11}Z_L \\ a_{21} & a_{22} + a_{21}Z_L \end{bmatrix} \\ \begin{bmatrix} b'_{11} & b'_{12} \\ b'_{21} & b'_{22} \end{bmatrix} &= \begin{bmatrix} 1 & Z_L \\ 0 & 1 \end{bmatrix} \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix} = \begin{bmatrix} b_{11} + b_{21}Z_L & b_{12} + b_{22}Z_L \\ b_{21} & b_{22} \end{bmatrix} \end{aligned} \quad (10.58)$$

Now, these overall parameters can be substituted into the corresponding formulas (listed in Table 10.3) to get the current gain of the two-port network with the load in Figure 10.17(a). For example, Equation (10.31) for the current gain can be obtained by substituting $z_{21} = z_{21}$ and $z_{22} = z_{22} + Z_L$ (from Equation (10.56)) for z_{21} and z_{22} into Equation (10.43). The results are listed in Table 10.4.

10.5.6.3 Overall Parameters for Output Impedance

The overall network is regarded as a series or series-parallel or cascade interconnection of two subnetworks depending on the base parameter. Noting that the overall parameters will be used to find the output impedance, the load impedance is excluded and the input port of the overall network is shorted (see Section 10.5.4 and Figures 10.17(b) and (d)).

(1) The overall z and y -parameters

To obtain the overall z and y -parameters, the overall network is regarded as a series interconnection, as depicted in Figure 10.24, which is symmetric with the network in Figure 10.21:

$$\begin{aligned} \begin{bmatrix} z'_{11} & z'_{12} \\ z'_{21} & z'_{22} \end{bmatrix} &= \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} + \begin{bmatrix} Z_s & 0 \\ 0 & 0 \end{bmatrix} = \begin{bmatrix} z_{11} + Z_s & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \\ \begin{bmatrix} y'_{11} & y'_{12} \\ y'_{21} & y'_{22} \end{bmatrix} &= \begin{bmatrix} z_{11} + Z_s & z_{12} \\ z_{21} & z_{22} \end{bmatrix}^{-1} = \frac{1}{y_{11} + Y_s} \begin{bmatrix} y_{11}Y_s & y_{12}Y_s \\ y_{21}Y_s & y_{22}Y_s + \Delta y \end{bmatrix} \end{aligned} \quad (10.59)$$

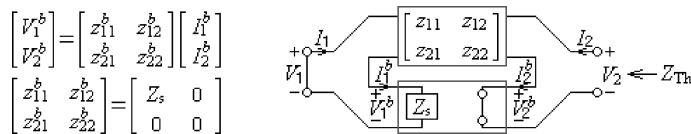


Figure 10.24 The z -parameter of a two-port network including a source impedance, regarded as a series interconnection with the overall input port shorted

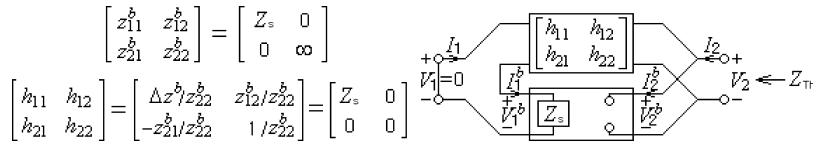


Figure 10.25 The h -parameter of a two-port network including a source impedance regarded as a series-parallel interconnection with the overall input port shorted

(2) The overall h and g -parameters

To obtain the overall h and g -parameters, the overall network is regarded as a series-parallel interconnection, as depicted in Figure 10.25, which is symmetric with the network in Figure 10.22:

$$\begin{aligned} \begin{bmatrix} h'_{11} & h'_{12} \\ h'_{21} & h'_{22} \end{bmatrix} &= \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} + \begin{bmatrix} Z_s & 0 \\ 0 & 0 \end{bmatrix} = \begin{bmatrix} h_{11} + Z_s & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \\ \begin{bmatrix} g'_{11} & g'_{12} \\ g'_{21} & g'_{22} \end{bmatrix} &= \begin{bmatrix} h_{11} + Z_s & h_{12} \\ h_{21} & h_{22} \end{bmatrix}^{-1} = \frac{1}{g_{11} + Y_s} \begin{bmatrix} g_{11} Y_s & g_{12} Y_s \\ g_{21} Y_s & g_{22} Y_s + \Delta g \end{bmatrix} \end{aligned} \quad (10.60)$$

(3) The overall a and b -parameters

To obtain the overall a and b -parameters, the overall network is regarded as a cascade interconnection, as depicted in Figure 10.26, which is symmetric with the network in Figure 10.23:

$$\begin{aligned} \begin{bmatrix} a'_{11} & a'_{12} \\ a'_{21} & a'_{22} \end{bmatrix} &= \begin{bmatrix} 1 & Z_s \\ 0 & 1 \end{bmatrix} \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = \begin{bmatrix} a_{11} + a_{21} Z_s & a_{12} + a_{22} Z_s \\ a_{21} & a_{22} \end{bmatrix} \\ \begin{bmatrix} b'_{11} & b'_{12} \\ b'_{21} & b'_{22} \end{bmatrix} &= \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix} \begin{bmatrix} 1 & Z_s \\ 0 & 1 \end{bmatrix} = \begin{bmatrix} b_{11} & b_{12} + b_{11} Z_s \\ b_{21} & b_{22} + b_{21} Z_s \end{bmatrix} \end{aligned} \quad (10.61)$$

Now, these overall parameters can be substituted into the corresponding formulas (listed in Table 10.3) to get the output impedance of the two-port network with the source impedance in Figure 10.17(a). For example, Equation (10.36) for the output impedance can be obtained by substituting $z'_{11} = z_{11} + Z_s$ and $\Delta z' = \Delta z + z_{22} Z_s$ (from Equation (10.59)) for z_{11} and Δz into Equation (10.46). The results are listed in Table 10.4.

10.6 Feedback Amplifiers as Two-Port Networks

In this section we will study various structures of feedback amplifiers and find their input-output properties based on their two-port parameters.

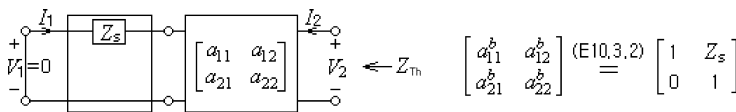


Figure 10.26 The a -parameter of a two-port network including a source impedance, regarded as a cascade interconnection with the overall input port shorted

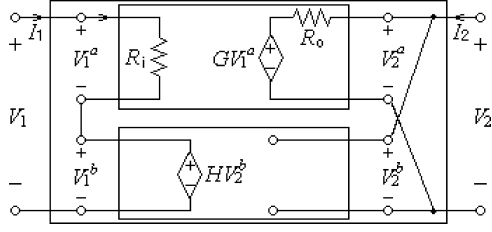


Figure 10.27 The series–parallel (shunt) feedback amplifier

10.6.1 Series–Parallel (Shunt) Feedback Amplifier

Figure 10.27 shows the basic structure of the series–parallel feedback amplifier, which consists of an amplifier with the open-loop gain G and a VCVS (voltage-controlled voltage source) with the gain H in the feedback path. For this series–parallel interconnection of two subnetworks it would be good to use the h -parameters. The h -parameter matrix of the lower subnetwork is obtained as

$$\begin{bmatrix} V_1^b \\ I_2^b \end{bmatrix} = \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix} \begin{bmatrix} I_1^b \\ V_2^b \end{bmatrix} \rightarrow \begin{bmatrix} h_{11}^b & h_{12}^b \\ h_{21}^b & h_{22}^b \end{bmatrix} = \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix}$$

Just because the z -parameters are easy to obtain for the upper subnetwork, first its z -parameter matrix is found and it is then converted to the h -parameter matrix as

$$\begin{aligned} \begin{bmatrix} V_1^a \\ V_2^a \end{bmatrix} &= \begin{bmatrix} R_i & 0 \\ GR_i & R_o \end{bmatrix} \begin{bmatrix} I_1^a \\ I_2^a \end{bmatrix} \rightarrow \begin{bmatrix} z_{11}^a & z_{12}^a \\ z_{21}^a & z_{22}^a \end{bmatrix} = \begin{bmatrix} R_i & 0 \\ GR_i & R_o \end{bmatrix} \\ &\begin{bmatrix} h_{11}^a & h_{12}^a \\ h_{21}^a & h_{22}^a \end{bmatrix} \stackrel{(10.14)}{\underset{\text{Table 10.1}}{=}} \begin{bmatrix} R_i & 0 \\ -GR_i/R_o & 1/R_o \end{bmatrix} \end{aligned}$$

These two h -parameter matrices can be added to get the h -parameters of the whole network as

$$\begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} = \begin{bmatrix} R_i & 0 \\ -GR_i/R_o & 1/R_o \end{bmatrix} + \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix} = \begin{bmatrix} R_i & H \\ -GR_i/R_o & 1/R_o \end{bmatrix} \quad (10.62)$$

and the corresponding formulas used in Table 10.4 to find the input–output properties as

$$Z_{\text{in}} = \left. \frac{V_1}{I_1} \right|_{I_2=0} = \left. \frac{\Delta h + h_{11}Y_L}{h_{22} + Y_L} \right|_{Y_L=0} = \frac{\Delta h}{h_{22}} = (1 + GH) \frac{R_i}{R_o} = (1 + GH)R_i \quad (10.63a)$$

$$G_c = \left. \frac{V_2}{V_1} \right|_{I_2=0} = \left. \frac{-h_{21}}{\Delta h + h_{11}Y_L} \right|_{Y_L=0} = \frac{GR_i/R_o}{(1 + GH)R_i/R_o} = \frac{G}{1 + GH} \quad (10.63b)$$

$$Z_{\text{out}} = \left. \frac{V_2}{I_2} \right|_{V_1=0} = \left. \frac{h_{11} + Z_s}{\Delta h + h_{22}Z_s} \right|_{Z_s=0} = \frac{h_{11}}{\Delta h} = \frac{R_i}{(1 + GH)R_i/R_o} = \frac{R_o}{1 + GH} \quad (10.63c)$$

10.6.2 Series–Series Feedback Amplifier

Figure 10.28 shows the basic structure of the series–series feedback amplifier, which consists of an amplifier with the open-loop gain G and an ICVS (current-controlled voltage source) with the gain H in

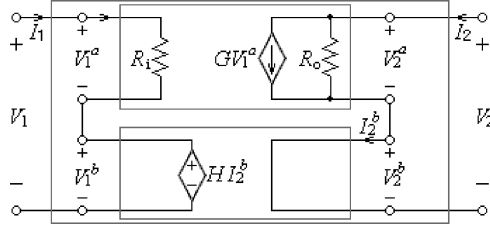


Figure 10.28 The series-series feedback amplifier

the feedback path. For this series-series interconnection of two subnetworks it would be good to use the z -parameters. The z -parameter matrices of the upper/lower subnetworks are

$$\begin{aligned} \begin{bmatrix} V_1^a \\ V_2^a \end{bmatrix} &= \begin{bmatrix} R_i & 0 \\ -GR_iR_o & R_o \end{bmatrix} \begin{bmatrix} I_1^a \\ I_2^a \end{bmatrix} \rightarrow \begin{bmatrix} z_{11}^a & z_{12}^a \\ z_{21}^a & z_{22}^a \end{bmatrix} = \begin{bmatrix} R_i & 0 \\ -GR_iR_o & R_o \end{bmatrix} \\ \begin{bmatrix} V_1^b \\ V_2^b \end{bmatrix} &= \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix} \begin{bmatrix} I_1^b \\ I_2^b \end{bmatrix} \rightarrow \begin{bmatrix} z_{11}^b & z_{12}^b \\ z_{21}^b & z_{22}^b \end{bmatrix} = \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix} \end{aligned}$$

We can add these two z -parameter matrices to get the z -parameters of the whole network as

$$\begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} = \begin{bmatrix} R_i & H \\ -GR_iR_o & R_o \end{bmatrix} + \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix} = \begin{bmatrix} R_i & H \\ -GR_iR_o & R_o \end{bmatrix} \quad (10.64)$$

and use the corresponding formulas in Table 10.4 to find the input-output properties as

$$Z_{in} = \left. \frac{V_1}{I_1} \right|_{V_2=0} = \left. \frac{\Delta z + z_{11}Z_L}{z_{22} + Z_L} \right|_{Z_L=0} = \frac{\Delta z}{z_{22}} = \frac{(1 + GH)R_iR_o}{R_o} = (1 + GH)R_i \quad (10.65a)$$

$$G_c = \left. \frac{I_2}{V_1} \right|_{V_2=0} = \left. \frac{-1}{V_1/I_1} \frac{-I_2}{I_1} \right|_{Z_L=0} = \left. \frac{-1}{Z'_{in}} A'_i \right|_{Z_L=0} \stackrel{(10.31)}{=} \frac{z_{21}/z_{22}}{\Delta z/z_{22}} \stackrel{(10.32)}{=} \frac{G}{1 + GH} \quad (10.65b)$$

$$Z_{out} = \left. \frac{V_2}{I_2} \right|_{V_1=0} = \left. \frac{\Delta z + z_{22}Z_s}{z_{11} + Z_s} \right|_{Z_s=0} = \frac{\Delta z}{z_{11}} = \frac{(1 + GH)R_iR_o}{R_i} = (1 + GH)R_o \quad (10.65c)$$

10.6.3 Parallel-Parallel Feedback Amplifier

Figure 10.29 shows the basic structure of the parallel-parallel feedback amplifier, which consists of an amplifier with the open-loop gain G and a VCIS (voltage-controlled current source) with the gain H in

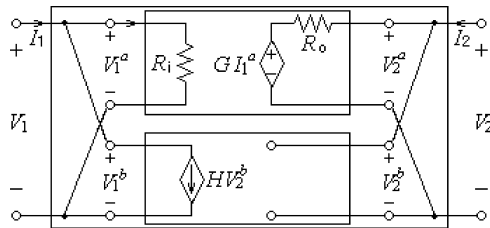


Figure 10.29 The parallel (shunt)-parallel (shunt) feedback amplifier

the feedback path. For this parallel–parallel interconnection of two subnetworks it would be good to use the y -parameters. The y -parameter matrix of the lower subnetwork is

$$\begin{bmatrix} I_1^b \\ I_2^b \end{bmatrix} = \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix} \begin{bmatrix} V_1^b \\ V_2^b \end{bmatrix} \rightarrow \begin{bmatrix} y_{11}^b & y_{12}^b \\ y_{21}^b & y_{22}^b \end{bmatrix} = \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix}$$

Just because the z -parameter is easy to obtain for the upper subnetwork, first its z -parameter matrix is found and then its inverse is taken to get the y -parameter matrix as

$$\begin{bmatrix} z_{11}^a & z_{12}^a \\ z_{21}^a & z_{22}^a \end{bmatrix} = \begin{bmatrix} R_i & 0 \\ G & R_o \end{bmatrix} \rightarrow \begin{bmatrix} y_{11}^a & y_{12}^a \\ y_{21}^a & y_{22}^a \end{bmatrix} = \begin{bmatrix} z_{11}^a & z_{12}^a \\ z_{21}^a & z_{22}^a \end{bmatrix}^{-1} = \begin{bmatrix} 1/R_i & 0 \\ -G/(R_i R_o) & 1/R_o \end{bmatrix}$$

These two y -parameter matrices can be added to get the y -parameters of the whole network as

$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} = \begin{bmatrix} 1/R_i & 0 \\ -G/(R_i R_o) & 1/R_o \end{bmatrix} + \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix} = \begin{bmatrix} 1/R_i & H \\ -G/(R_i R_o) & 1/R_o \end{bmatrix} \quad (10.66)$$

and the corresponding formulas in Table 10.4 used to find the input–output properties as

$$Z_{in} = \frac{V_1}{I_1} \Big|_{I_2=0} = \frac{y_{22} + Y_L}{\Delta y + y_{11} Y_L} \Big|_{Y_L=0} = \frac{y_{22}}{\Delta y} = \frac{1/R_o}{(1 + GH)/(R_i R_o)} = \frac{R_i}{1 + GH} \quad (10.67a)$$

$$G_c = \frac{V_2}{I_1} = \frac{V_1}{I_1} \frac{V_2}{V_1} \Big|_{I_2=0} = Z_{in} A_v \Big|_{Y_L=0} = \frac{y_{22} + Y_L}{\Delta y + y_{11} Y_L} \frac{-y_{21}}{y_{22} + Y_L} \Big|_{Y_L=0} = \frac{-y_{21}}{\Delta y} = \frac{G}{1 + GH} \quad (10.67b)$$

$$Z_{out} = \frac{V_2}{I_2} \Big|_{V_1=0} = \frac{y_{11} + Y_s}{\Delta y + y_{22} Y_s} \Big|_{Y_s=0} = \frac{y_{11}}{\Delta y} = \frac{1/R_i}{(1 + GH)/(R_i R_o)} = \frac{R_o}{1 + GH} \quad (10.67c)$$

10.6.4 Parallel (Shunt)–Series Feedback Amplifier

Figure 10.30 shows the basic structure of the parallel–series feedback amplifier, which consists of an amplifier with the open-loop gain G and an CCCS (current-controlled current source) with the gain H in the feedback path. For this parallel–series interconnection of two subnetworks it would be good to use the g -parameters. The g -parameter matrix of the lower subnetwork is

$$\begin{bmatrix} I_1^b \\ V_2^b \end{bmatrix} = \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix} \begin{bmatrix} V_1^b \\ I_2^b \end{bmatrix} \rightarrow \begin{bmatrix} g_{11}^b & g_{12}^b \\ g_{21}^b & g_{22}^b \end{bmatrix} = \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix}$$

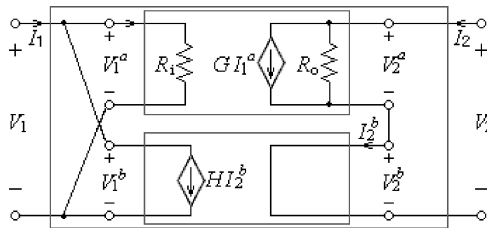


Figure 10.30 The parallel (shunt)–series feedback amplifier

Just because the z -parameter is easy to obtain for the upper subnetwork, first its z -parameter matrix is found and then it is converted into the g -parameter matrix as

$$\begin{bmatrix} z_{11}^a & z_{12}^a \\ z_{21}^a & z_{22}^a \end{bmatrix} = \begin{bmatrix} R_i & 0 \\ -GR_o & R_o \end{bmatrix} \rightarrow \begin{bmatrix} g_{11}^a & g_{12}^a \\ g_{21}^a & g_{22}^a \end{bmatrix} \xrightarrow{\text{Table 10.4}} \begin{bmatrix} 1/z_{11}^a & -z_{12}^a/z_{11}^a \\ z_{21}^a/z_{11}^a & \Delta z^a/z_{11}^a \end{bmatrix} = \begin{bmatrix} 1/R_i & 0 \\ -GR_o/R_i & R_o \end{bmatrix}$$

These two g -parameter matrices can be added to get the g -parameters of the whole network as

$$\begin{bmatrix} g_{11} & g_{12} \\ g_{21} & g_{22} \end{bmatrix} = \begin{bmatrix} 1/R_i & 0 \\ -GR_o/R_i & R_o \end{bmatrix} + \begin{bmatrix} 0 & H \\ 0 & 0 \end{bmatrix} = \begin{bmatrix} 1/R_i & H \\ -GR_o/R_i & R_o \end{bmatrix} \quad (10.68)$$

and the corresponding formulas in Table 10.4 used to find the input–output properties as

$$Z_{\text{in}} = \frac{V_1}{I_1} \Big|_{V_2=0} = \frac{g_{22} + Z_L}{\Delta g + g_{11}Z_L} \Big|_{Z_L=0} = \frac{g_{22}}{\Delta g} = \frac{R_o}{(1 + GH)R_o/R_i} = \frac{R_i}{1 + GH} \quad (10.69a)$$

$$G_c = \frac{I_2}{I_1} \Big|_{V_2=0} = \frac{-g_{21}}{\Delta g + g_{11}Z_L} \Big|_{Z_L=0} = \frac{-g_{21}}{\Delta g} = \frac{G}{1 + GH} \quad (10.69b)$$

$$Z_{\text{out}} = \frac{V_2}{I_2} \Big|_{I_1=0} = \frac{\Delta g + g_{22}Y_s}{g_{11} + Y_s} \Big|_{Y_s=0} = \frac{\Delta g}{g_{11}} = (1 + GH)R_o \quad (10.69c)$$

10.6.5 General Feedback Structure

Figure 10.31 shows the basic structure of a feedback system that consists of a forward amplifier (with an open-loop gain $G(s)$) and a feedback amplifier (with a negative feedback gain $H(s)$). The relationships among the (source) reference signal $r(t)$, the output signal $y(t)$, the feedback signal $b(t)$, and the error signal $e(t)$ are as follows:

$$Y(s) = G(s)E(s) \quad (10.70a)$$

$$B(s) = H(s)Y(s) \quad (10.70b)$$

$$E(s) = R(s) - B(s) \quad (10.70c)$$

where $R(s)$, $Y(s)$, $B(s)$, and $E(s)$ are the Laplace transforms of $r(t)$, $y(t)$, $b(t)$, and $e(t)$, respectively. Substituting Equation (10.70c) together with Equation (10.70b) into Equation (10.70a) yields

$$\begin{aligned} Y(s) &= G(s)E(s) = G(s)[R(s) - B(s)] = G(s)R(s) - G(s)H(s)Y(s) \\ \frac{Y(s)}{R(s)} &= \frac{G(s)}{1 + G(s)H(s)} \end{aligned} \quad (10.71)$$

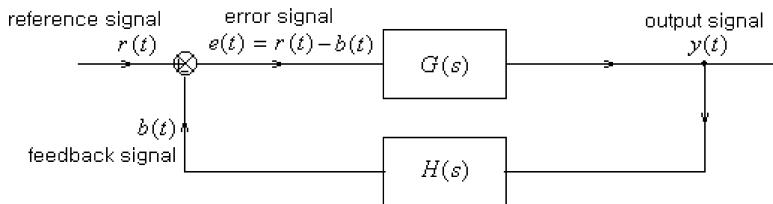
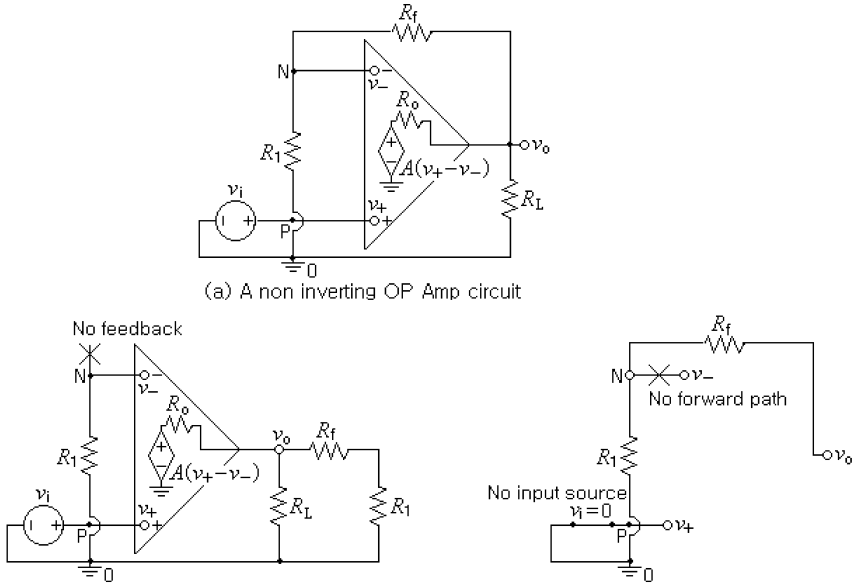


Figure 10.31 The block diagram showing the relationship among several signals in a general feedback system



(b1) The circuit for determining the open-loop gain (b2) The circuit for determining the feedback gain

Figure 10.32 A noninverting OP Amp circuit regarded as a feedback system

which is called the *closed-loop transfer function* or the *closed-loop gain*, while $G(s)$ is the *open-loop transfer function* or the *open-loop gain*, $H(s)$ the *(negative) feedback gain*, and $G(s)H(s)$ the *loop transfer function* or the *loop gain*.

In fact, it is not a coincidence that Equations (10.63b), (10.65b), (10.67b), and (10.69b) for the overall gains of the above four types of feedback amplifiers are all of the same form as this closed-loop gain (10.71). The OP amplifier circuit in Figure 10.32(a) is also a kind of feedback amplifier and therefore Equation (10.71) can be used to get its closed-loop gain as

$$\begin{aligned} \frac{V_o}{V_i} &= \frac{G}{1 + GH} = \frac{\frac{AR_L(R_1 + R_f)}{R_o(R_L + R_1 + R_f) + R_L(R_1 + R_f)}}{1 + \frac{AR_L(R_1 + R_f)}{R_o(R_L + R_1 + R_f) + R_L(R_1 + R_f)} \frac{R_1}{R_1 + R_f}} \\ &= \frac{A(R_1 + R_f)R_L}{R_oR_L + (R_o + R_L)(R_1 + R_f) + AR_1R_L} \end{aligned} \quad (10.72)$$

which agrees with Equation (2.37) in Section 2.9.4. Here, the open-loop gain

$$G = \left. \frac{V_o}{V_+} \right|_{\text{No feedback (} V_- = 0 \text{)}} = A \frac{R_L \parallel (R_1 + R_f)}{R_o + R_L \parallel (R_1 + R_f)} = \frac{AR_L(R_1 + R_f)}{R_o(R_L + R_1 + R_f) + R_L(R_1 + R_f)} \quad (10.73a)$$

is obtained from the circuit with the feedback path disconnected at the negative input terminal as depicted in Figure 10.32(b1), while the (negative) feedback gain

$$H = \left. \frac{V_-}{V_o} \right|_{\text{No input (} V_i = 0 \text{)}} = \frac{R_1}{R_1 + R_f} \quad (10.73b)$$

is obtained from the circuit with the input source and the forward path removed as depicted in Figure 10.32(b2).

The circuit in Figure 10.33(a) is also a kind of feedback amplifier and therefore Equation (10.71) can be used to get its closed-loop gain as

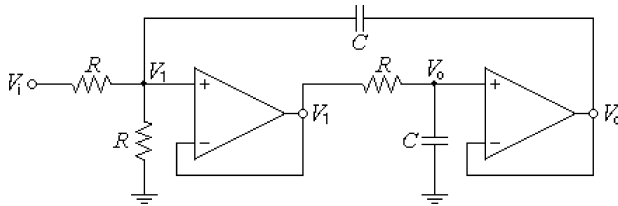
$$\frac{V_o}{V_i} = G_1 \frac{G}{1 + GH} = \frac{1}{RCs + 2} \frac{\frac{1}{RCs + 1}}{1 + \frac{1}{RCs + 1} \left(-\frac{RCs}{RCs + 2} \right)} = \frac{1}{(RC)^2 s + 2RCs + 2} \quad (10.74)$$

Here, the open-loop gain

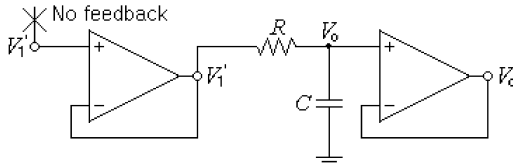
$$G = \left. \frac{V_o'}{V_i'} \right|_{\text{No feedback}} = \frac{1/(sC)}{R + 1/(sC)} = \frac{1}{RCs + 1} \quad (10.75a)$$

is obtained from the circuit with the feedback path removed as depicted in Figure 10.33(b1), the feedforward gain

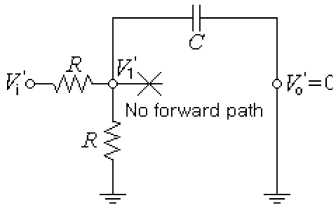
$$G_1 = \left. \frac{V_1'}{V_i'} \right|_{\text{No forward } V_o=0} = \frac{R \parallel (1/sC)}{R + [R \parallel (1/sC)]} = \frac{R/(sC)}{R^2 + 2R/(sC)} = \frac{1}{RCs + 2} \quad (10.75b)$$



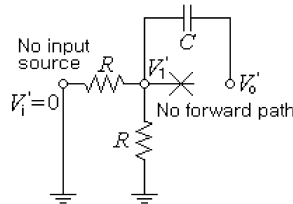
(a) A feedback OP Amp circuit



(b1) The circuit for determining the open-loop gain



(b2) The circuit for determining the feedforward gain



(b3) The circuit for determining the feedback gain

Figure 10.33 An OP Amp circuit regarded as a feedback system

is obtained from the circuit with the forward path removed as depicted in Figure 10.33(b2), and the negative feedback gain

$$H = -\frac{V'_1}{V'_o} \Big|_{\substack{\text{No forward} \\ V'_i=0}} = -\frac{R \parallel R}{(R \parallel R) + 1/(sC)} = -\frac{RCs}{RCs + 2} \quad (10.75c)$$

is obtained from the circuit with the input source and the forward path removed as depicted in Figure 10.33(b3). Note that the negative sign has been put to the negative feedback gain because the feedback input is connected to the positive input terminal of the OP Amp.

The circuit in Figure 10.34(a) is also a kind of feedback amplifier and therefore Equation (10.71) can be used to get its closed-loop gain as

$$\begin{aligned} \frac{V_o}{I_i} &= \frac{G}{1 + GH} = \frac{(1/R_f - g_m)(R_s \parallel R_f \parallel r_{be})(R_f \parallel R_C)}{1 + (1/R_f - g_m)(R_s \parallel R_f \parallel r_{be})(R_f \parallel R_C)(-1/R_f)} \\ &= \frac{(1 - g_m R_f) R_s r_{be} R_C}{R_s R_C + R_s R_f + R_s r_{be} + g_m R_s r_{be} R_C + r_{be} R_f + r_{be} R_C} \end{aligned} \quad (10.76)$$

Here, the open-loop gain

$$G = \frac{V'_o}{I'_i} \Big|_{\text{No feedback}} = \frac{V_{be}}{I'_i} \frac{V'_o}{V_{be}} = \left(\frac{1}{R_f} - g_m \right) (R_s \parallel R_f \parallel r_{be})(R_f \parallel R_C) \quad (10.77a)$$

is obtained from the circuit with the feedback path removed as depicted in Figure 10.34(b1) and the negative feedback gain

$$H = -\frac{I'_f}{V'_o} \Big|_{\substack{\text{No input} \\ (V_{be}=0)}} = -\frac{1}{R_f} \quad (10.77b)$$

is obtained from the circuit with the input source and the forward path removed as depicted in Figure 10.34(b2) (References [R-1] and [S-1]).

Note. Equation (10.76) can also be obtained from the node analysis (see Example 10.9).

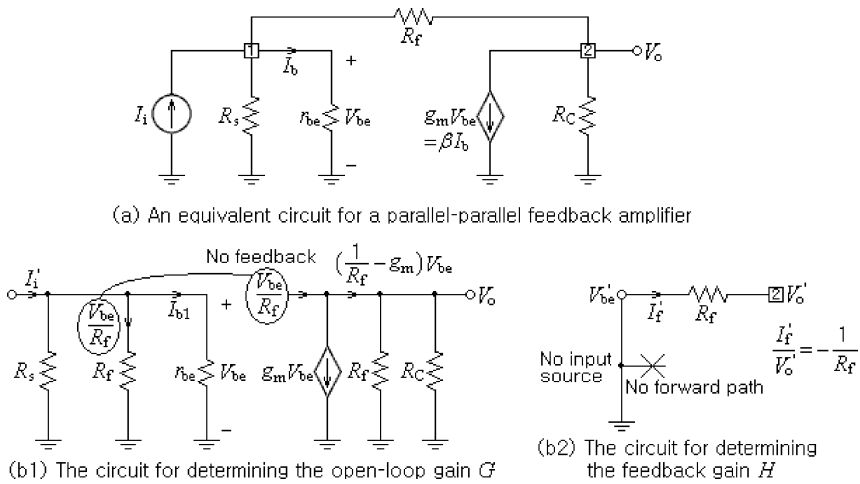


Figure 10.34 A feedback amplifier regarded as a feedback system

10.7 Circuit Models with Given Parameters

So far we have discussed how to find the parameters for a given two-port network. In this section we will see how to construct a two-port model with a given parameter.

10.7.1 Circuit Model with Given z -Parameters

If a given z -parameter matrix is symmetric, the T-model in Figure 10.35(a) suffices; otherwise, we have to resort to the model having two CCVSs (current-controlled voltage sources) as depicted in Figure 10.35(b).

10.7.2 Circuit Model with Given y -Parameters

If a given y -parameter matrix is symmetric, the Π -model in Figure 10.36(a) suffices; otherwise, we have to resort to the model having two VCCSs (voltage-controlled current sources) as depicted in Figure 10.36(b).

10.7.3 Circuit Model with Given h and g -Parameters

For given h and g -parameters, one choice is the model having a VCVS (voltage-controlled voltage source) and a CCCS (current-controlled current source) as depicted in Figures 10.37(a) and (b).

10.7.4 Circuit Model with Given a and b -Parameters

For given a and b -parameters, one choice is the model having a VCVS, a CCVS (current-controlled voltage source), and a CCCS as depicted in Figures 10.38(a) and (b).

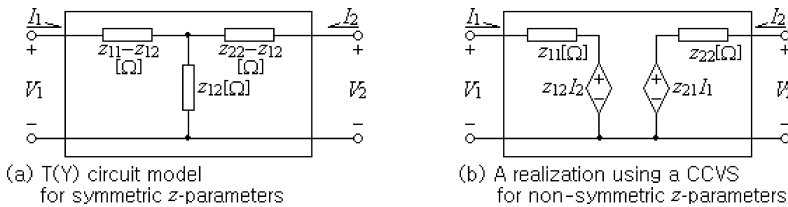


Figure 10.35 Two-port network models with a given z -parameter matrix

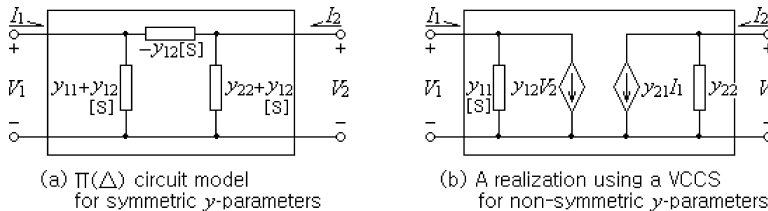


Figure 10.36 Two-port network models with a given y -parameter matrix

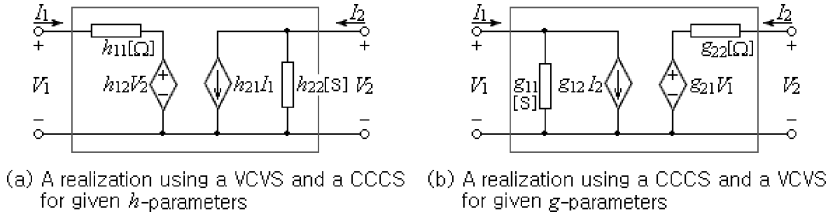


Figure 10.37 Two-port network models with a given h/g -parameter matrix

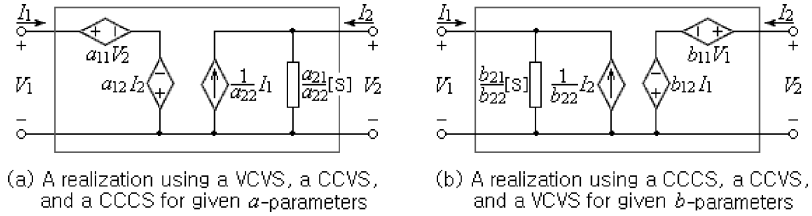


Figure 10.38 Two-port network models with a given a/b -parameter matrix

(Example 10.9) Making Use of `Port_conversion()` and `Port_property()`

Figure 10.39 shows that the feedback amplifier of Figure 10.34(a) can be regarded as a parallel–parallel (shunt) connection of two two-port subnetworks. Note that the y -parameters of the upper and lower subnetworks can be written from Equation (E10.3.5) and Figure 10.37(a) with Table 10.1 as

$$y_1 \stackrel{(E10.3.5)}{=} \begin{bmatrix} 1/R_f & -1/R_f \\ -1/R_f & 1/R_f \end{bmatrix} \quad (E10.9.1)$$

$$h_2 \stackrel{\text{Figure 10.37(a)}}{=} \begin{bmatrix} r_{be} & 0 \\ g_m r_{be} & 1/R_C \end{bmatrix} \xrightarrow[\text{port\_conversion()}]{\text{Table 10.1}} y_2 = \begin{bmatrix} 1/r_{be} & 0 \\ g_m & 1/R_C \end{bmatrix} \quad (E10.9.2)$$

These two y -parameter matrices can be added to get the y -parameters, the input impedance Z_{in} , and the voltage gain A_v of the composite two-port network (excluding the source part consisting of V_s and R_s) as follows:

$$y \stackrel{(10.25)}{=} y_1 + y_2 \stackrel{(E10.9.1)}{\stackrel{(E10.9.2)}{=}} \begin{bmatrix} 1/R_f + 1/r_{be} & -1/R_f \\ -1/R_f + g_m & 1/R_f + 1/R_C \end{bmatrix} \quad (E10.9.3)$$

$$Z_{in} \stackrel{\text{Table 10.4}}{=} \frac{y_{22} + Y_L}{\Delta y + y_{11} Y_L} \bigg|_{Z_L = \infty} = \frac{y_{22}}{\Delta y} \stackrel{(E10.9.3)}{=} \frac{(1/R_f + 1/R_C) r_{be} R_f R_C}{r_{be} + R_f + R_C + g_m r_{be} R_C} \quad (E10.9.4)$$

$$A_v = \frac{V_o}{V_i} \stackrel{\text{Table 10.4}}{\stackrel{\text{port\_property()}}{=}} \frac{-y_{21}}{y_{22} + Y_L} \bigg|_{Z_L = \infty} = \frac{-y_{21}}{y_{22}} \stackrel{(E10.9.3)}{=} -\frac{g_m - 1/R_f}{1/R_f + 1/R_C} \quad (E10.9.5)$$

Now we can obtain the overall voltage gain of the whole two-port network including the source part consisting of V_s and R_s as

$$\frac{V_o}{V_i} = \frac{V_1}{V_i} \frac{V_o}{V_1} \stackrel{(10.77)}{=} \frac{Z_{in}}{R_s + Z_{in}} A_v \stackrel{(E10.9.4)}{\stackrel{(E10.9.5)}{=}} \frac{(1 - g_m R_f) r_{be} R_C}{R_s r_{be} + R_s R_f + R_s R_C + g_m R_s r_{be} R_C + r_{be} R_C + r_{be} R_f} \quad (E10.9.6)$$

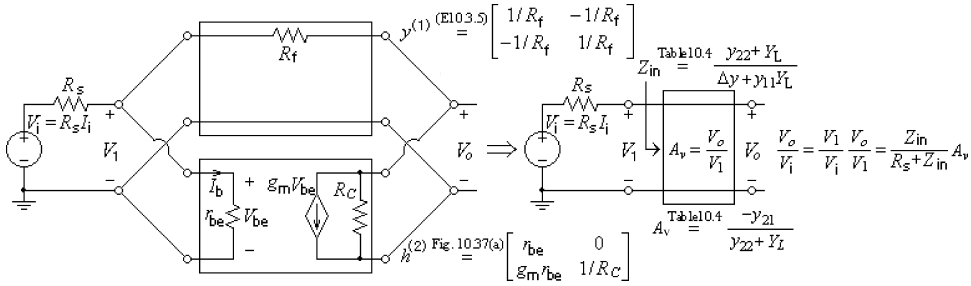


Figure 10.39 The feedback amplifier of Figure 10.34(a) regarded as a parallel–parallel (shunt) connection of two two-port subnetworks

Note. This conforms to Equation (10.76) since $V_o/V_i = V_o/(R_s I_i)$. All these operations are cast into the MATLAB program `cir10e09.m` shown in Problem 10.13, which uses the MATLAB functions `port_conversion()` and `port_property()` introduced above.

Problems

10.1 A Bridge Circuit as a Two-Port Network

Consider the circuit in Figure P10.1(a).

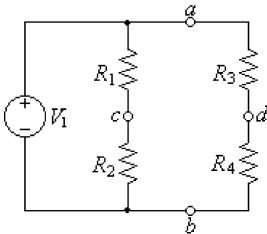
- (a) Noting that it can be regarded as a two-port network, verify that its z and y -parameters are as follows (you might use the results of Example 2.18 and refer to Section 10.4.6):

$$\begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} = \frac{1}{R_1 + R_2 + R_3 + R_4} \begin{bmatrix} (R_1 + R_2)(R_3 + R_4) & R_2 R_3 - R_1 R_4 \\ R_2 R_3 - R_1 R_4 & (R_1 + R_3)(R_2 + R_4) \end{bmatrix} \quad (\text{P10.1.1})$$

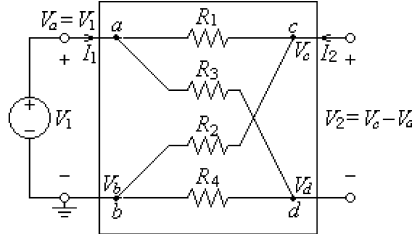
$$\begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} = \frac{1}{G_1 + G_2 + G_3 + G_4} \begin{bmatrix} (G_1 + G_3)(G_2 + G_4) & G_2 G_3 - G_1 G_4 \\ G_2 G_3 - G_1 G_4 & (G_1 + G_2)(G_3 + G_4) \end{bmatrix} \quad (\text{P10.1.2})$$

- (b) Using the input–output relationship (10.2) described by the y -parameters, find the short-circuit current at port 2 in terms of the y -parameters and V_1 :

$$I_{SC} = -I_2|_{V_2=0} = \quad (\text{P10.1.3})$$



(a) A bridge circuit



(b) Regarded as a two-port network

Figure P10.1 A bridge circuit regarded as a two-port network

- (c) Using the formula for the voltage gain (listed in Table 10.3), find the open-circuit voltage at port 2 in terms of R_1 values and V_1 :

$$V_{OC} = V_2|_{I_2=0} = \quad (\text{P10.1.4})$$

Is it equal to the Thevenin equivalent voltage?

- (d) Using the formula for the Thevenin equivalent impedance seen from the output (listed in Table 10.3), find it in terms of R_1 values:

$$Z_{Th} = \quad (\text{P10.1.5})$$

- (e) Let $R_1 = 1 \Omega$, $R_2 = 3 \Omega$, $R_3 = 2 \Omega$, and $R_4 = 4 \Omega$. Assuming that a 170 V voltage source is applied to the input port and a load resistor $R_L = 5 \Omega$ is connected at the output port $c-d$, find the current flowing through $R_L = 5 \Omega$ in the following two ways:
- Use the Thevenin equivalent circuit seen from the output port $c-d$.
 - Use the formula for the overall voltage gain (listed in Table 10.4) to get the output voltage and divide it by R_L .

10.2 Two-Port Networks

- (a) Find the z -parameters of the circuit in Figure P10.2(a).
 (b) Find the h -parameters of the circuit in Figure P10.2(b).
 (c) Find the a -parameters of the circuit in Figure P10.2(c1). You can replace the part of magnetically (inductively) coupled circuit by its T-equivalent circuit as depicted in Figure P10.2(c2). Also find the a -parameters of the network on the assumption that the transformer is an ideal one with $L_1 : M : L_2 = n : 1$ and $L_1, M, L_2 \gg R_2, 1/C_1$ (R_2 and $1/C_1$ are negligible compared with L_1, M , and L_2).

Note. Readers are encouraged to use MATLAB or its equivalent to obtain the solutions.

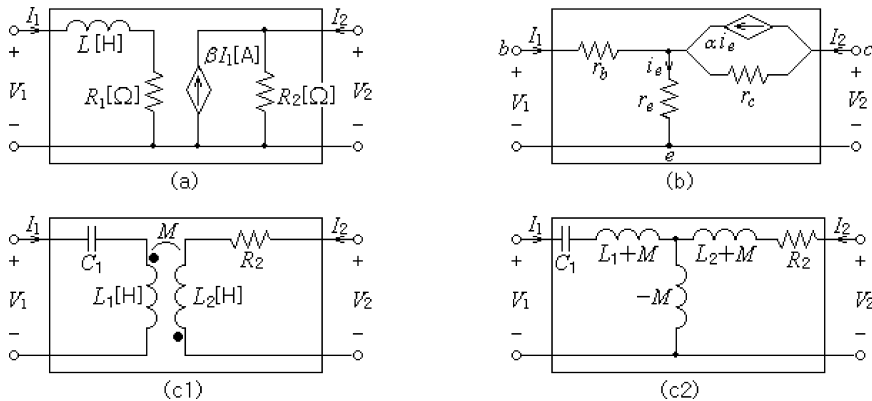


Figure P10.2

10.3 A Two-Port Network

Consider the two-port network in Figure P10.3. Noting that the virtual short principle (Remark 1.2(2)) applied to the OP Amp with negative feedback says that $V_1 = V_f = V_2$ and so $V_{ae} = Z_1 I_{ae} = Z_2 I_{fe} = V_{fe}$, $V_{gf} = Z_3 I_{gf} = Z_4 I_{gc} = V_{gc}$, find the a -parameters of the network.

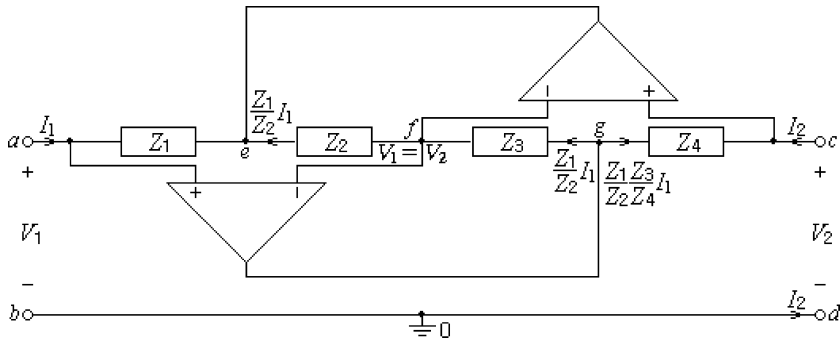


Figure P10.3

10.4 A Two-Port Network as a Connection of Subnetworks

Consider the two-port network in Figure P10.4(a).

- (a) Which connection of the two subnetworks in Figure P10.4(b) can it be regarded as series, parallel, series-parallel, parallel-series, or cascade? In reference to this connection type, which parameter of z , y , h , g , and a is the best as the base parameter for finding the overall parameters?
- (b) Find the base parameter matrices of the two subnetworks and then that of the whole network.
- (c) Find the h -parameters of the whole network, hopefully by using the MATLAB routine `port_conversion()`.

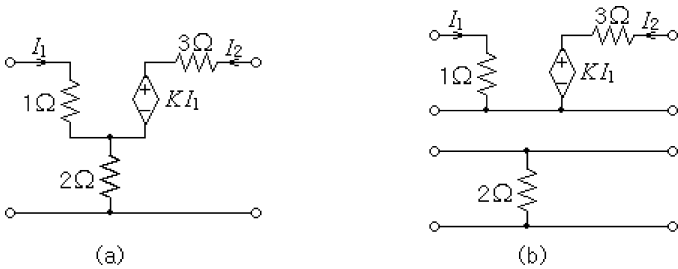


Figure P10.4

10.5 A Two-Port Network as a Connection of Subnetworks

Consider the two-port network in Figure P10.5(a).

- (a) Which connection of two subnetworks in Figures P10.5(b1) and (b2) can it be regarded as series, parallel, series-parallel, parallel-series, or cascade? In reference to this connection type, which parameter is the best as the base parameter for finding the overall parameters? Find the parameters of each subnetwork, hopefully by using the MATLAB routine `port_conversion()`.
- (b) Find its y -parameters.
- (c) Find the open-loop voltage gain $A_v = V_2/V_1|_{I_2=0}$ by using Table 10.3. Does it agree with Equation (8.44) with $\sigma = 0$?

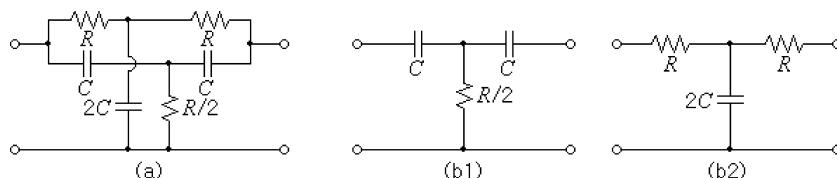


Figure P10.5 A twin T-network

10.6 A Two-Port Network as a Connection of Subnetworks

Consider the two-port network in Figure P10.6(a).

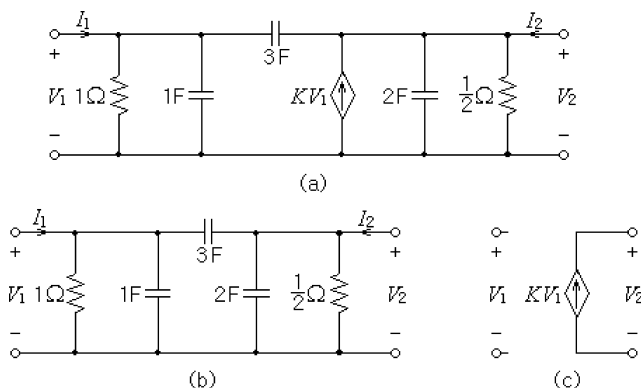


Figure P10.6 A two-port network

- Which connection of the two subnetworks in Figures P10.6(b) and (c) can it be regarded as series, parallel, series-parallel, parallel-series, or cascade? In reference to this connection type, which parameter is the best as the base parameter for finding the overall parameters? Find the parameters of each subnetwork.
- Find its a -parameters, hopefully by using the MATLAB routine `port_conversion()`.

10.7 A Two-Port Network as a Connection of Three Subnetworks

Consider the two-port network in Figure P10.7(a).

- What connection of the three subnetworks in Figures P10.7(b1), (b2), and (b3) can it be regarded as? In reference to this connection type, fill in the blanks in the following statement: It can be regarded as the () connection of the () connection of Figures (b1) and (b2) and (b3).
- Which parameter of z , y , h , g , and a is easiest to find for the T-subnetworks in Figures P10.7(b1) and (b2)? Find the parameter matrices of the two subnetworks.
- Find the overall z -parameters, hopefully by using the MATLAB routine `port_conversion()`.

10.8 A Two-Port Network with Source/Load Impedances

- Find the input and output impedances of the two-port network in Figure P10.2(b), where a voltage source V_s with an impedance Z_s is connected at the input port and a load impedance Z_L is connected at the output port.
- Using the a -parameters obtained in Problem 10.3 and the corresponding formula listed in Table 10.4, find the input impedance of the two-port network in Figure P10.3, where a load impedance Z_L is connected at the output port.

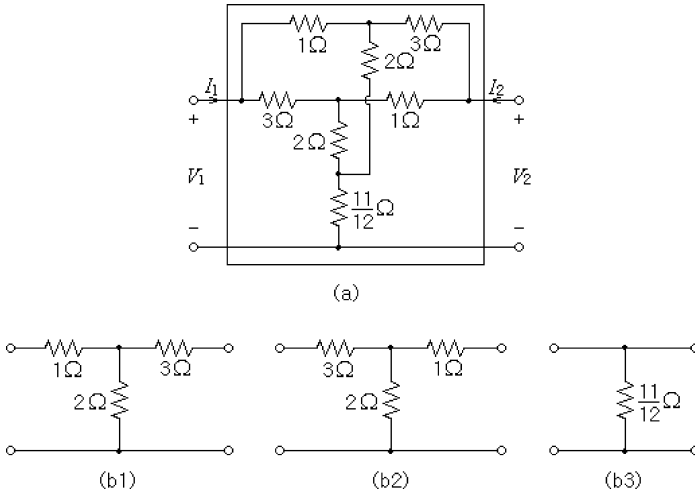


Figure P10.7

10.9 A Two-Port Network for Impedance Matching – Image Impedances

- (a) Consider the two-port network in Figure P10.9(a), where the source and load impedances are purely resistive. Verify that the z -parameters and the a -parameters should satisfy the following conditions in order for the two-port network to provide impedance matchings simultaneously for the source and the load ($R_s = Z_{in}$ and $Z_{out} = R_L$):

$$\frac{z_{22}}{z_{11}} = \frac{a_{22}}{a_{11}} = \frac{R_L}{R_s}, \quad \Delta z = \frac{a_{12}}{a_{21}} = R_s R_L \quad (\text{P10.9.1})$$

- (b) Consider the two-port network in Figure P10.9(b), where the (angular) frequency of the AC voltage source is ω [rad/s]. Find the z -parameters or a -parameters of the two-port network consisting of a capacitor C and an inductor L and the values of C and L such that the impedance matching conditions are satisfied. Do they agree with the results of Example 6.7?

10.10 Interconnection of Two-Port Networks and the Port Condition

- (a) Find the z -parameters of the two-port network in Figure P10.10(b).

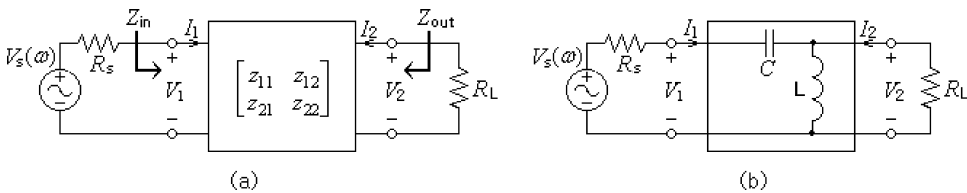


Figure P10.9 A two-port network for impedance matching

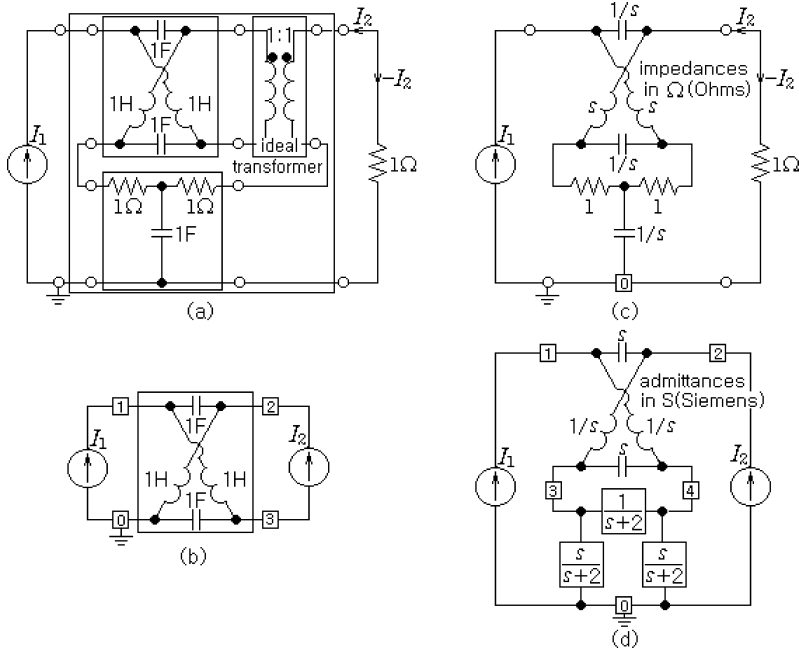


Figure P10.10

- (b) Regarding the upper part of the network in Figure P10.10(a) as a cascade interconnection of two subnetworks, find its a -parameters, hopefully by using the MATLAB routine `port_conversion()`.
- (c) Regarding the whole network as a series-series interconnection of the upper/lower subnetworks, find its z -parameters, hopefully by using the MATLAB routine `port_conversion()`.
- (d) Using the z -parameters obtained in (c) and the corresponding formula listed in Table 10.4 or the MATLAB routine `port_property()`, find the overall current gain $A_i = -I_2/I_1$ of the whole network with the load impedance of $1\ \Omega$ as

$$G(s) = A_i = \frac{-I_2(s)}{I_1(s)} = \frac{s^2 + 1}{s^2 + 4s + 3} \quad (\text{P10.10.1})$$

- (e) Plot the magnitude of the frequency response $|G(j\omega)|$ for $\omega = 2\pi f$ ($f = 1\text{ mHz} - 10\text{ Hz}$) by using MATLAB. Also perform the PSpice simulation to get the frequency response as follows:
- Set the coupling coefficient of the magnetically coupled circuit (with the part name of `XFRM_LINEAR`) to $k = 1$ (corresponding to the ideal transformer condition (5.19a)) and the inductances of the two coils to $L_1 = 10^6$ and $L_2 = 10^6$ (corresponding to the ideal transformer condition of infinite permeance).
 - Apply an AC current source (IAC) of 1 A to the input port.
 - Set the Analysis type in the simulation profile to 'AC Sweep' and the frequency range 1 mHz (the start frequency)–10 Hz (the end frequency) with 200 points/decade.
 - Put the current marker to one terminal of the load resistor connected to the output port.

Do the two frequency responses obtained using MATLAB and PSpice look the same?

```

%cir10p10.m
clear, clf
syms s I1 I2

% (a) z-parameter of the left-upper part
YA=[s+1/s -s -1/s; ?? ????? ?; -1/s 0 s+1/s];
ZA=YA^-1; ZA1=[ ZA(1,:); ZA(2,:)-ZA(3,:)];
Za=[ZA1(:,1) ZA1(:,2)-ZA1(:,3)]; pretty(simple(Za))

% (b) a-parameter of the upper part
A1=port_conversion(?,?,????)*eye(2); pretty(simple(A1))
% z-parameter of the upper part
Z1=port_conversion(?,?,????); pretty(simple(Z1))

% (c) z-parameter of the whole system
Z2=[s+1 1; ? ???] /s;
Z=Z1+Z2; pretty(simple(Z))

% (d) Current Gain G(s)=Ai=-I2(s)/I1(s)=(s^2+1)/(s^2+4s+3)
Ail=port_property(?,?, '???'); pretty(simple(Ail))

% (e) Frequency Response
f=logspace(-3,1,801); % frequency vector from 0.001 to 10Hz
Ailw=subs(Ail,s,j*2*pi*f); % frequency response
subplot(221), semilogx(f,abs(Ailw)), axis([f(1) f(end) 0 1.1])

% (f) Time Response by using the Laplace transform
I2=Ail*3/s; % -I2(s)=Ai*I1(s)
i2=ilaplace(I2) % step response=1+5*exp(-3*t)-3*exp(-t)
t=[0:1000]/100; % time vector for 10sec
i2t=eval(i2);
subplot(222), plot(t,i2t,'b', t,1+5*exp(-3*t)-3*exp(-t),'r')

% (i) The node impedance matrix of the network without transformer
Y=[s+1/s -s 0 -1/s; ?? ????? ????? ?;
    0 -1/s 1/s+s+(s+1)/(s+2) -s-1/(s+2);
    ????? ?????????? ??????????????????];

% The z-parameter matrix of the network without transformer
Yinv=Y^-1; Z=Yinv(1:2,1:2)

% Current gain Ai2(s)=-I2(s)/I1(s)
Ai2=port_property(?,?, '???'); pretty(simple(Ai2))

% Frequency Response of the network without transformer
Ai2w=subs(Ai2,s,j*2*pi*f); % frequency response
subplot(223), semilogx(f,abs(Ai2w)), axis([f(1) f(end) 0 1.1])

% System (transfer) Function G(s)=N(s)/D(s)
Ns=[2 3 6 4 4 1]; Ds=[2 7 10 9 6 1]; % numerator/denominator
Gs=tf(Ns,Ds)

% Time Response to 3*u(t) - 3 times step response
i2t=3*step(Gs,t); % step response
% Alternatively, using residue() or ilaplace_my()
[r,p,k]=residue(3*Ns,[Ds 0]); i2t1=real(r.*exp(p*t));
i2t2=ilaplace_my(3*Ns,[Ds 0]) % inverse Laplace of Y(s)=G(s)*3/s
subplot(224), plot(t,i2t,'b', t,i2t1,'r', t,eval(i2t2),'m')

```

- (f) Verify that the output current $-i_2(t)$ of the whole network is

$$-i_2(t) = 1 - 3e^{-t} + 5e^{-3t} \quad (\text{P10.10.2})$$

when a DC current source of 3 A is applied to the input port at time $t = 0$. Plot the output current for $t = 0 \sim 10$ s.

- (g) Perform the PSpice simulation to get the output current $-i_2(t)$ for $t = 0 \sim 10$ s as follows:
- Apply the DC current source (IDC) of 3 A to the input port.
 - Set the Analysis type in the simulation profile to 'Time Domain (Transient)' and the final time (Run to time) to 10 s, and check the square box before 'skip the initial transient bias point calculation (SKIPBP)'.

Does the time response agree with that obtained by the analytical method in (f)?

- (h) Since the a -parameter matrix of the ideal transformer with a turns ratio of 1:1 is an identity matrix, it seems that the same parameters could be obtained for the two-port network without the ideal transformer, in Figure P10.10(c). In order to prove or disprove this idea, perform the PSpice simulation to get the frequency response and the time response of the network without the ideal transformer, as in (e) and (g). Are they the same as those obtained with the transformer? If not, what do you think the cause of the difference is? In connection with this diagnosis, think about how the current flows in the initial state of the DC circuit with the inductors/capacitors open/short and in the final (steady) state with the inductors/capacitors short/open, respectively. Does it satisfy the port condition that was introduced in Section 10.4.6?
- (i) In order to analyze the network without the ideal transformer, set up the node equation in $V_1(s)$, $V_2(s)$, $V_3(s)$, and $V_4(s)$ for the network of Figure P10.10(c) where the T(Y) circuit in the lower part can be replaced by its $\Pi(\Delta)$ equivalent as depicted in Figure P10.10(d). Solve it to get the expressions for $V_1(s)$ and $V_2(s)$ in terms of $I_1(s)$ and $I_2(s)$. Find the z -parameters and the overall current gain as

$$G(s) = A_i = \frac{-I_2(s)}{I_1(s)} = \frac{2s^5 + 3s^4 + 6s^3 + 4s^2 + 4s + 1}{2s^5 + 7s^4 + 10s^3 + 9s^2 + 6s + 1} \quad (\text{P10.10.3})$$

Plot the frequency response and the time response (to the 3 A current source) of the network and check if they look the same as those obtained from the PSpice simulation in (h).

Note. You might finish the above MATLAB program and use it.

Note. For the time response to the 3 A current source, it would be better to use the MATLAB command `step(sys,t)` or `residue(num,den)` to get the numerical values of $i_2(t)$ since `ilaplace()` does not seem to be good at obtaining the analytical expression of the inverse Laplace transform of a high-degree s -function.

```
>>Ns=[2 3 6 4 4 1];Ds=[2 7 10 9 6 1]; % numerator/denominator
>>Gs=tf(Ns,Ds) % transfer function
>>t=[0:1000]/100; % time vector for 10sec
>>i2t= 3*step(Gs,t); % 3 times the step response
>>plot(t,i2t,'b')
>>[r,p,k]=residue(3*Ns,[Ds 0]); % partial fraction of Y(s)=G(s)*3/s
>>i2t1=real(r.*exp(p*t)); % inverse Laplace transform
>>hold on, plot(t,i2t1,'r')
```

10.11 Two-Port Networks in Various Types of Connection

- (a) Regarding the two-port network of Figure P10.11(a) as appropriate connections of subnetworks N^1 , N^2 , and N^3 , find the overall a -parameters, the input impedance Z_i with R_L taken into consideration, the output impedance Z_o with R_1 taken into consideration, and the overall voltage gain $A_v = V_o/V_s$.
- (b) Regarding the two-port network of Figure P10.11(b) as appropriate connections of subnetworks N^1 , N^2 , N^3 , and N^4 , find the input impedance Z_i and the overall voltage gain $A_v = V_o/V_s$.

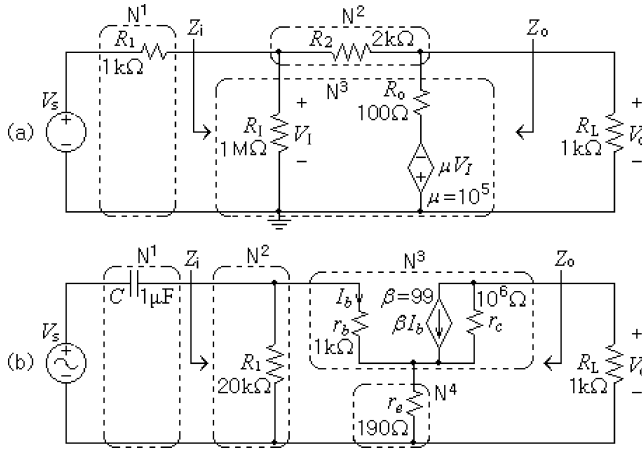


Figure P10.11

10.12 Two-Port Networks in a Cascade Connection and Impedance Matching Design

Consider the circuit of Figure P10.12 in which $R_s = 75 \Omega$, $R_{12} = 2 \text{ k}\Omega$, $R_L = 16 \Omega$, and the z -parameter matrices of the two two-port sub-networks are

$$\begin{bmatrix} z_{11}^a & z_{12}^a \\ z_{21}^a & z_{22}^a \end{bmatrix} = \begin{bmatrix} 350 & 2.667 \\ -10^6 & 6667 \end{bmatrix} \quad \text{and} \quad \begin{bmatrix} z_{11}^b & z_{12}^b \\ z_{21}^b & z_{22}^b \end{bmatrix} = \begin{bmatrix} 1.0262 \times 10^6 & 6791 \\ 1.0258 \times 10^6 & 6794 \end{bmatrix} \quad (\text{P10.12.1})$$

respectively.

- (a) Find the overall a -parameters for the cascade interconnection of N^a , R_{12} , and N^b . Then find the input impedance Z_{in} with R_L taken into consideration, the output impedance Z_{out} with R_s taken into consideration, and the overall voltage gain $A_v = V_2/V_s$.

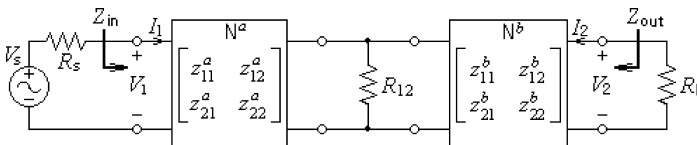


Figure P10.12 Two-port networks in cascade connection

- (b) Does the output impedance Z_{out} match the load impedance R_L ? If not, find the new value of the resistance R_{12} such that Z_{out} matches R_L .

10.13 Two-Port Networks in a Parallel–Parallel (Shunt) Connection and Overall Voltage Gain

- (a) Run the following program `cir10e09.m` to solve Example 10.9 and check if the results conform to those obtained in the example. Identify the statements corresponding to Equations (10.76), (E10.9.2), (E10.9.3), (E10.9.4), (E10.9.5), and (E10.9.6).
- (b) Apply the node analysis to solve the circuit of Figure 10.34(a) and find V_o/I_i by using MATLAB or its equivalent to check if it conforms to Equation (10.76).

```
%cir10e09.m
clear
syms Rs rbe Rf gm Rc ZL Ii
G=(1/Rf-gm)*parallel_comb([Rs Rf rbe])*parallel_comb([Rf Rc]);
H=-1/Rf;
Av0=G/(1+G*H); pretty(simplify(Av0))
y1=[1/Rf -1/Rf; -1/Rf 1/Rf];
h2=[rbe 0; gm*rbe 1/Rc];
y2=port_conversion(h2,'h2y')
y=y1+y2
ZL=inf;
Zin=port_property(y,ZL,'ZiY');
Av=port_property(y,ZL,'AvY')
Avo=Av*Zin/(Rs+Zin); pretty(simplify(Avo))
% Node Analysis
Y=[1/Rs+1/rbe+1/Rf -1/Rf; gm-1/Rf 1/Rf+1/Rc]; Is=[Ii; 0]
V=Y\Is;
V(2)/Ii
```

Appendices

Appendix A: Laplace Transform

The Laplace transform was discovered originally by Leonhard Euler (1707–1783), the great eighteenth century Swiss mathematician and physicist, but is named in honor of a French mathematician and astronomer Pierre-Simon Laplace (1749–1827), who used the transform in his work on probability theory. He was such a genius, not only in mathematics, where the great mathematician Simeon Poisson (1781–1840) labeled him the Isaac Newton of France, but also in politics, where he could serve three regimes in revolutionary France – the republic, the empire of Napoleon, and the Bourbon restoration, having been bestowed a count from Napoleon and a marquis from Louis XVIII.

The Laplace transform is a very useful tool in solving differential equations and also plays an important role in dealing with linear time-invariant systems. Tables A.1 and A.2 list the Laplace transforms of basic functions and the Laplace transform properties, respectively.

A.1 Definition of Laplace Transform

The (unilateral or one-sided) Laplace transform is defined for a function $x(t)$ of a real variable t (often meaning the time) as

$$X(s) = \mathcal{L}\{x(t)\} = \int_{0-}^{\infty} x(t)e^{-st} dt \quad (\text{A.1})$$

where s is a complex variable, the lower limit, t_- , of the integration interval is the instant just before $t = 0$, and $x(t)$ is often assumed to be causal in the sense that it is zero for all $t < 0$.

A.2 Examples of Laplace Transform

A.2.1 Laplace Transform of the Unit Step Function

The unit step function is defined as

$$u_s(t) = \begin{cases} 1 & \text{for } t \geq 0 \\ 0 & \text{for } t < 0 \end{cases} \quad (\text{A.2})$$

which is depicted in Figure A.1(a). Equation (A.1) can be used to obtain the Laplace transform of the unit step function as

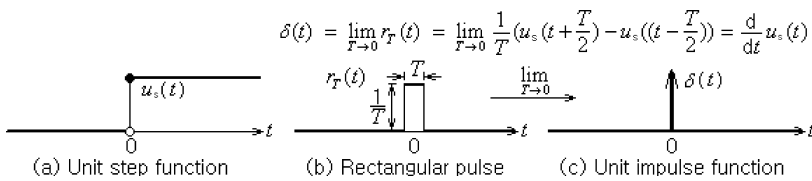
$$\mathcal{L}\{u_s(t)\} = \int_0^{\infty} u_s(t) e^{-st} dt = \int_0^{\infty} e^{-st} dt \stackrel{(\text{F.33})}{=} \left. \frac{1}{-s} e^{-st} \right|_0^{\infty} = \frac{1}{-s} (0 - 1) = \frac{1}{s}$$

Table A.1 Laplace transforms of basic functions

$x(t)$	$X(s)$	$x(t)$	$X(s)$
(1) $\delta(t)$	1	(6) $t^m e^{-at} u_s(t)$	$\frac{m!}{(s+a)^{m+1}}$
(2) $\delta(t-t_1)$	$e^{-t_1 s}$	(7) $\sin \omega t u_s(t)$	$\frac{\omega}{s^2 + \omega^2}$
(3) $u_s(t)$	$\frac{1}{s}$	(8) $\cos \omega t u_s(t)$	$\frac{s}{s^2 + \omega^2}$
(4) $t^m u_s(t)$	$\frac{m!}{s^{m+1}}$	(9) $e^{-at} \sin \omega t u_s(t)$	$\frac{\omega}{(s+a)^2 + \omega^2}$
(5) $e^{-at} u_s(t)$	$\frac{1}{s+a}$	(10) $e^{-at} \cos \omega t u_s(t)$	$\frac{s+a}{(s+a)^2 + \omega^2}$

Table A.2 Properties of Laplace transform

Definition	$X(s) = \mathcal{L}\{x(t)\} = \int_0^\infty x(t) e^{-st} dt, \quad x(t) \leftrightarrow X(s)$
(1) Linearity	$\alpha x(t) + \beta y(t) \leftrightarrow \alpha X(s) + \beta Y(s)$
(2) Time shifting (real translation)	$x(t-t_1) u_s(t-t_1), t_1 > 0 \leftrightarrow e^{-st_1} \left[X(s) + \int_{-t_1}^0 x(\tau) e^{-s\tau} d\tau \right]$ $= e^{-st_1} X(s) \text{ for } x(t) = 0 \forall t < 0$
(3) Frequency shifting (complex translation)	$e^{st_1} x(t) \leftrightarrow X(s-s_1)$
(4) Real convolution	$g(t) * x(t) \leftrightarrow G(s) X(s)$
(5) Time derivative (differentiation property)	$x'(t) = \frac{d}{dt} x(t) \leftrightarrow sX(s) - x(0)$
(6) Time integral (integration property)	$\int_{-\infty}^t x(\tau) d\tau \leftrightarrow \frac{1}{s} X(s) + \frac{1}{s} \int_{-\infty}^0 x(\tau) d\tau$
(7) Complex derivative	$t x(t) \leftrightarrow -\frac{d}{ds} X(s)$
(8) Complex convolution	$x(t)y(t) \leftrightarrow \frac{1}{2\pi j} \int_{\sigma_0-\infty}^{\sigma_0+\infty} X(v)Y(s-v) dv$
(9) Initial value theorem	$x(0) = \lim_{s \rightarrow \infty} sX(s)$
(10) Final value theorem	$x(\infty) = \lim_{s \rightarrow 0} sX(s)$

**Figure A.1**

This Laplace transform pair is denoted by

$$u_s(t) \xleftrightarrow{\mathcal{L}} \frac{1}{s} \quad (\text{A.3})$$

A.2.2 Laplace Transform of the Unit Impulse Function

The unit impulse function can be defined as the limit of a rectangular pulse function $r_T(t)$ with the pulsewidth $T \rightarrow 0$ (converging to zero) or simply the time derivative of the unit step function as

$$\delta(t) = \lim_{T \rightarrow 0} r_T(t) = \lim_{T \rightarrow 0} \frac{1}{T} \left[u_s\left(t + \frac{T}{2}\right) - u_s\left(t - \frac{T}{2}\right) \right] = \frac{d}{dt} u_s(t) \quad (\text{A.4})$$

which is depicted in Figure A.1(c). Equation (A.1) can be used to obtain the Laplace transform of the unit impulse function as

$$\mathcal{L}\{\delta(t)\} = \int_0^\infty \delta(t) e^{-st} dt = 1$$

This Laplace transform pair is denoted by

$$\delta(t) \xleftrightarrow{\mathcal{L}} 1 \quad (\text{A.5})$$

A.2.3 Laplace Transform of the Ramp Function

The Laplace transform of the unit ramp function $t u_s(t)$ is obtained as

$$\mathcal{L}\{t u_s(t)\} \stackrel{(\text{A.1})}{=} \int_0^\infty t u_s(t) e^{-st} dt = \int_0^\infty t e^{-st} dt \stackrel{(\text{F.36})}{=} \left. \frac{t}{-s} e^{-st} \right|_0^\infty + \frac{1}{s} \int_0^\infty e^{-st} dt \stackrel{(\text{F.33})}{=} \frac{1}{s^2}$$

This Laplace transform pair is denoted by

$$t u_s(t) \xleftrightarrow{\mathcal{L}} \frac{1}{s^2} \quad (\text{A.6})$$

A.2.4 Laplace Transform of the Exponential Function

The Laplace transform of the exponential function $e^{-at} u_s(t)$ is obtained as

$$\mathcal{L}\{e^{-at} u_s(t)\} = \int_0^\infty e^{-at} u_s(t) e^{-st} dt = \int_0^\infty e^{-at} e^{-st} dt = \int_0^\infty e^{-(s+a)t} dt \stackrel{(\text{F.33})}{=} \frac{1}{s+a}$$

This Laplace transform pair is denoted by

$$e^{-at} u_s(t) \xleftrightarrow{\mathcal{L}} \frac{1}{s+a} \quad (\text{A.7})$$

A.2.5 Laplace Transform of the Complex Exponential Function

Substituting $\sigma + j\omega$ for a into (A.7) yields the Laplace transform of the exponential function as

$$e^{-(\sigma+j\omega)t} u_s(t) \stackrel{(\text{F.20})}{=} e^{-\sigma t} (\cos \omega t - j \sin \omega t) u_s(t) \xleftrightarrow{\mathcal{L}} \frac{1}{s + \sigma + j\omega} = \frac{s + \sigma}{(s + \sigma)^2 + \omega^2} - j \frac{\omega}{(s + \sigma)^2 + \omega^2}$$

$$e^{-\sigma t} \cos \omega t u_s(t) \xleftrightarrow{\mathcal{L}} \frac{s + \sigma}{(s + \sigma)^2 + \omega^2} \quad (\text{A.8})$$

$$e^{-\sigma t} \sin \omega t u_s(t) \xleftrightarrow{\mathcal{L}} \frac{\omega}{(s + \sigma)^2 + \omega^2} \quad (\text{A.9})$$

A.3 Properties of the Laplace Transform

Let the Laplace transforms of two functions $x(t)$ and $y(t)$ be $X(s)$ and $Y(s)$, respectively.

A.3.1 Linearity

The Laplace transform of a linear combination of $x(t)$ and $y(t)$ can be written as

$$\alpha x(t) + \beta y(t) \xrightarrow{\mathcal{L}} \alpha X(s) + \beta Y(s) \quad (\text{A.10})$$

A.3.2 Time Differentiation

The Laplace transform of the derivative of $x(t)$ w.r.t. t can be written as

$$x'(t) \xrightarrow{\mathcal{L}} sX(s) - x(0) \quad (\text{A.11})$$

Proof

$$\mathcal{L}\{x'(t)\} \stackrel{(\text{A.1})}{=} \int_0^\infty \frac{dx}{dt} e^{-st} dt \stackrel{(\text{F.36})}{=} x(t) e^{-st} \Big|_0^\infty - (-s) \int_0^\infty x(t) e^{-st} dt \stackrel{(\text{A.1})}{=} sX(s) - x(0)$$

Repetitive application of this time differentiation property yields the Laplace transform of the n th-order derivative of $x(t)$ w.r.t. t as

$$x^{(n)}(t) \xrightarrow{\mathcal{L}} s^n X(s) - s^{n-1}x(0) - s^{n-2}x'(0) - \dots - x^{(n-1)}(0) \quad (\text{A.12})$$

A.3.3 Time Integration

The Laplace transform of the integral of $x(t)$ w.r.t. t can be written as

$$\int_{-\infty}^t x(\tau) d\tau \xrightarrow{\mathcal{L}} \frac{1}{s} X(s) + \frac{1}{s} \int_{-\infty}^0 x(\tau) d\tau \quad (\text{A.13})$$

This can be derived by substituting $\int_{-\infty}^t x(\tau) d\tau$ and $x(t)$ for $x(t)$ and $x'(t)$ into Equation (A.11) as

$$x(t) \xrightarrow{\mathcal{L}} X(s) = s \mathcal{L} \left\{ \int_{-\infty}^t x(\tau) d\tau \right\} - \int_{-\infty}^0 x(\tau) d\tau$$

Repetitive application of this time integration property yields the Laplace transform of the n th-order integral of $x(t)$ w.r.t. t as

$$\int_{-\infty}^t \int_{-\infty}^t \dots \int_{-\infty}^t x(\tau) d\tau^n \xrightarrow{\mathcal{L}} s^{-n} X(s) + s^{-n} \int_{-\infty}^0 x(\tau) d\tau + \dots + \int_{-\infty}^0 \int_{-\infty}^0 \dots \int_{-\infty}^0 x(\tau) d\tau^n \quad (\text{A.14})$$

A.3.4 Time Shifting – Real Translation

The Laplace transform of a delayed function $x(t)$ can be obtained as follows:

$$\begin{aligned}\mathcal{L}\{x(t-t_1)\} &\stackrel{(A.1)}{=} \int_{-\infty}^{t_1} x(t-t_1) e^{-st} dt + \int_{t_1}^{\infty} x(t-t_1) e^{-st} dt \\ &\stackrel{t-t_1=\tau}{=} e^{-st_1} \int_{-t_1}^0 x(\tau) e^{-s\tau} d\tau + e^{-st_1} \int_0^{\infty} x(\tau) e^{-s\tau} d\tau \\ x(t-t_1), t_1 > 0 &\stackrel{\mathcal{L}}{\longleftrightarrow} e^{-st_1} \left\{ X(s) + \int_{-t_1}^0 x(\tau) e^{-s\tau} d\tau \right\} \xrightarrow{\text{if } x(t)=0 \forall t < 0} e^{-st_1} X(s)\end{aligned}\quad (A.15)$$

A.3.5 Frequency Shifting – Complex Translation

$$e^{s_1 t} x(t) \stackrel{\mathcal{L}}{\longleftrightarrow} X(s-s_1) \quad (A.16)$$

A.3.6 Real Convolution

The (real) convolution of two (causal) functions $g(t)$ and $x(t)$ is defined as

$$g(t) * x(t) = \int_{-\infty}^{\infty} g(\tau) x(t-\tau) d\tau \stackrel{\text{if } g(t)=0 \text{ and } x(t)=0 \forall t < 0}{\stackrel{\text{causality}}{=}} \int_0^t g(\tau) x(t-\tau) d\tau \quad (A.17)$$

The Laplace transform of the convolution $y(t) = g(t) * x(t)$ turns out to be the product of the Laplace transforms of the two functions as

$$y(t) = g(t) * x(t) \stackrel{\mathcal{L}}{\longleftrightarrow} Y(s) = G(s) X(s) \quad (A.18)$$

Proof

$$\begin{aligned}\mathcal{L}\{g(t) * x(t)\} &= \int_0^{\infty} g(t) * x(t) e^{-st} dt \stackrel{(A.17)}{=} \int_0^{\infty} \left[\int_0^t g(\tau) x(t-\tau) d\tau \right] e^{-st} dt \\ &\stackrel{\text{Figure A.2}}{=} \int_0^{\infty} g(\tau) e^{-s\tau} \int_{\tau}^{\infty} x(t-\tau) e^{-s(t-\tau)} dt d\tau \stackrel{(t-\tau=v)}{=} \int_0^{\infty} g(\tau) e^{-s\tau} \left[\int_0^{\infty} x(v) e^{-sv} dv \right] d\tau \\ &= \int_0^{\infty} g(\tau) e^{-s\tau} d\tau X(s) = G(s) X(s)\end{aligned}$$

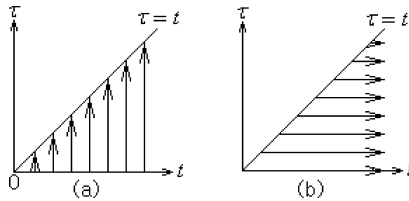


Figure A.2

This property is used to describe the input–output relationship of a linear time-invariant (LTI) system which has the input $x(t)$, the impulse function $g(t)$, and the output $y(t)$, where the Laplace transform of the impulse function, i.e. $G(s) = \mathcal{L}\{g(t)\}$, is referred to as the system function or transfer function of the system.

A.3.7 Partial Differentiation

Given the Laplace transform of a function having a parameter a , i.e. $\mathcal{L}\{x(t, a)\} = X(s, a)$, the Laplace transform of the derivative of the parameterized function $x(t, a)$ w.r.t. a equals the derivative of its Laplace transform $X(s, a)$ w.r.t. a :

$$\frac{\partial}{\partial a} x(t, a) \xrightarrow{\mathcal{L}} \frac{\partial}{\partial a} X(s, a) \quad (\text{A.19})$$

A.3.8 Complex Differentiation

The Laplace transform of $-tx(t)$ equals the derivative of $\mathcal{L}\{x(t)\} = X(s)$ w.r.t. s :

$$tx(t) \xrightarrow{\mathcal{L}} -\frac{d}{ds} X(s) \quad (\text{A.20})$$

This can be derived by differentiating Equation (A.1) w.r.t. s .

(Example A.1) Applying the Partial/Complex Differentiation Property

To find the Laplace transform of $t e^{-at} u_s(t)$, Equation (A.7) is differentiated w.r.t. a and multiplied by -1 to get

$$t e^{-at} u_s(t) \xrightarrow{\mathcal{L}} \frac{1}{(s+a)^2} \quad (\text{A.21})$$

Multiplying t by the left-hand side (LHS) and applying the complex differentiation property (A.20) repetitively, gives

$$t^2 e^{-at} u_s(t) \xrightarrow{\mathcal{L}} \frac{2!}{(s+a)^3} \quad (\text{A.22})$$

.....

$$t^m e^{-at} u_s(t) \xrightarrow{\mathcal{L}} \frac{m!}{(s+a)^{m+1}} \quad (\text{A.23})$$

A.3.9 Initial Value Theorem

The initial value of $x(t)$ is obtained from its Laplace transform $\mathcal{L}\{x(t)\} = X(s)$ as follows:

$$x(0) = \lim_{s \rightarrow \infty} sX(s) \quad (\text{A.24})$$

This can be derived by substituting $s = \infty$ into the time differentiation property (A.11) as

$$\lim_{s \rightarrow \infty} \mathcal{L}\{x'(t)\} = \lim_{s \rightarrow \infty} \int_0^{\infty} x'(t) e^{-st} dt = 0 = \lim_{s \rightarrow \infty} sX(s) - x(0)$$

A.3.10 Final Value Theorem

The final value of $x(t)$ can be obtained from its Laplace transform $\mathcal{L}\{x(t)\} = X(s)$ as follows:

$$x(\infty) = \lim_{s \rightarrow 0} sX(s) \quad (\text{A.25})$$

on the premise that $x(t)$ is convergent or, equivalently, all poles of $X(s)$ are in the left-half plane (LHP) except for a simple pole at $s = 0$. This can be derived by substituting $s = 0$ into the time differentiation property (A.11) as

$$\lim_{s \rightarrow 0} \mathcal{L}\{x'(t)\} = \lim_{s \rightarrow 0} \int_0^{\infty} x'(t) e^{-st} dt = \int_0^{\infty} x'(t) dt = x(\infty) - x(0) = \lim_{s \rightarrow 0} sX(s) - x(0)$$

(Question) Can this final value theorem be applied to give $\sin \omega t|_{t=\infty} = \lim_{s \rightarrow 0} sX(s) \stackrel{\text{Table A.1(7)}}{=} \lim_{s \rightarrow 0} s\omega/(s^2 + \omega^2) = 0$?

A.4 Inverse Laplace Transform

Suppose the s -function $X(s)$ is given in the form of a rational function, i.e. a ratio of an M th-degree polynomial $Q(s)$ to an N th-degree polynomial $P(s)$ in s and is expanded into the partial fractions as

$$X(s) = \frac{Q(s)}{P(s)} = \frac{b_M s^M + \cdots + b_1 s + b_0}{a_N s^N + \cdots + a_1 s + a_0} \quad (M \leq N) \quad (\text{A.26})$$

$$= \left(\sum_{n=1}^{N-L} \frac{r_n}{s - p_n} \right) + \frac{r_{N-L+1}}{s - p} + \cdots + \frac{r_N}{(s - p)^L} + K \quad (\text{A.27})$$

where

$$r_n = (s - p_n) \frac{Q(s)}{P(s)} \Big|_{s=p_n}, \quad n = 1, 2, \dots, N - L \quad (\text{A.28a})$$

$$r_{N-l} = \frac{1}{l!} \frac{d^l}{ds^l} \left[(s - p)^L \frac{Q(s)}{P(s)} \right] \Big|_{s=p}, \quad l = 0, 1, \dots, L - 1 \quad (\text{A.28b})$$

Then the inverse Laplace transform of $X(s)$ can be obtained as

$$x(t) = \left[\sum_{n=1}^{N-L} r_n e^{p_n t} + r_{N-L+1} e^{pt} + r_{N-L+2} t e^{pt} + \cdots + \frac{r_N}{(N-1)!} t^{L-1} e^{pt} \right] u_s(t) + K \delta(t) \quad (\text{A.29})$$

(Example A.2) Inverse Laplace Transform

Let us find the inverse Laplace transform of the following s -functions.

$$(a) X(s) = \frac{3s^2 + 11s + 11}{s^3 + 4s^2 + 5s + 2} = \frac{3s^2 + 11s + 11}{(s+1)^2(s+2)} = \frac{r_1}{s+2} + \frac{r_2}{s+1} + \frac{r_3}{(s+1)^2} \quad (\text{with a double pole at } s = -1) \quad (1)$$

We can use the formula (A.28a) to find the simple-pole coefficient r_1 as

$$r_1 = (s+2)X(s) \Big|_{s=-2} \stackrel{(1)}{=} (s+2) \frac{3s^2 + 11s + 11}{(s+1)^2(s+2)} \Big|_{s=-2} = 1 \quad (2)$$

and can also use the formula (A.28b) to find the multiple-pole coefficient r_2 and r_3 as

$$r_3 = (s+1)^2 X(s) \Big|_{s=-2} \stackrel{(1)}{=} \frac{3s^2 + 11s + 11}{s+2} \Big|_{s=-1} = 3 \quad (3)$$

$$r_2 = \frac{d}{ds} (s+1)^2 X(s) \Big|_{s=-1} = \frac{d}{ds} \frac{3s^2 + 11s + 11}{s+2} \Big|_{s=-1} \stackrel{(F.31)}{=} \frac{(6s+11)(s+2) - (3s^2 + 11s + 11)}{(s+2)^2} \Big|_{s=-1} = 2 \quad (4)$$

Thus the inverse Laplace transform can be written as

$$x(t) \stackrel{(1)}{\underset{(A.29)}{=}} (r_1 e^{-2t} + r_2 e^{-t} + r_3 t e^{-t}) u_s(t) = (e^{-2t} + 2e^{-t} + 3t e^{-t}) u_s(t) \quad (5)$$

$$>>Ns=[3 \ 11 \ 11]; \ Ds=[1 \ 4 \ 5 \ 2]; \ [r,p,k] = \text{residue}(Ns,Ds), \ [A,B] = \text{residue}(r,p,k)$$

$$(b) X(s) = \frac{s}{s^2 + 2s + 2} = \frac{s}{(s+1)^2 + 1^2} \text{ (with complex conjugate poles at } s = -1 \pm j) \quad (6)$$

The formula (A.28a) may be used to find the coefficients of the following partial fraction expansion form:

$$X(s) = \frac{s}{s^2 + 2s + 2} = \frac{r_1}{s+1-j} + \frac{r_2}{s+1+j} \quad (7)$$

as

$$\begin{aligned} r_1 &\stackrel{(A.28a)}{=} (s+1-j) X(s) \Big|_{s=-1+j} \stackrel{(6)}{=} \frac{s}{s+1+j} \Big|_{s=-1+j} = 0.5(1+j) \\ r_2 &\stackrel{(A.28a)}{=} (s+1+j) X(s) \Big|_{s=-1-j} \stackrel{(6)}{=} \frac{s}{s+1-j} \Big|_{s=-1-j} = 0.5(1-j) \end{aligned} \quad (8)$$

Thus the inverse Laplace transform can be written as

$$x(t) \stackrel{(7)}{\underset{(A.29)}{=}} \left[0.5(1+j) e^{(-1+j)t} + 0.5(1-j) e^{(-1-j)t} \right] u_s(t) \stackrel{(F.20)}{=} e^{-t} (\cos t - \sin t) u_s(t) \quad (9)$$

In the case of complex conjugate poles, it may be simpler to equate $X(s)$ with the following form:

$$X(s) = \frac{s}{s^2 + 2s + 2} = \frac{A(s+1)^2}{(s+1)^2 + 1} + \frac{B \times 1}{(s+1)^2 + 1} \stackrel{\text{common denominator}}{=} \frac{As + (A+B)}{(s+1)^2 + 1^2} \quad (10)$$

and get the coefficients as $A = 1$ and $B = -1$. Then each term of the inverse Laplace transform can be found directly from the Laplace transform Table A.1 and the inverse Laplace transform written as

$$X(s) \stackrel{(10)}{=} \frac{s+1}{(s+1)^2 + 1^2} + \frac{(-1) \times 1}{(s+1)^2 + 1^2} \xrightarrow{\text{Table A.1(9),(10)}} x(t) = (e^{-t} \cos t - e^{-t} \sin t) u_s(t) \quad (11)$$

(c) Use of MATLAB for Partial Fraction Expansion and Inverse Laplace Transform

The MATLAB command `residue()` can be used to get the partial fraction expansion and `ilaplace()` to obtain the whole inverse Laplace transform. It should, however, be noted that `ilaplace()` might not work properly for high-degree rational functions.

```
>>Ns=[ 3 11 11]; Ds=[ 1 4 5 2]; [r,p,k]=residue(Ns,Ds); [r p],k % (1)
    r = 1.0000    p = -2.0000    % (2) 1/(s-(-2))
        2.0000    -1.0000    % (4) 2/(s-(-1))
        3.0000    -1.0000    % (3) 3/(s-(-1))^2
    k=[]
>>syms s, x=ilaplace((3*s^2+11*s+11)/(s^3+4*s^2+5*s+2))
    x = exp(-2*t)+3*t*exp(-t)+2*exp(-t)    % (5)
>>Ns=[ 1 0]; Ds=[ 1 2 2]; [r,p,k]=residue(Ns,Ds); [r p],k % (6)
    r = 0.5000+0.5000i p = -1.0000+1.0000i % (8) (0.5+0.5i)/(s+1-i)
        0.5000-0.5000i p = -1.0000-1.0000i % (8) (0.5-0.5i)/(s+1+i)
    k=[]
>>syms s, x=ilaplace(s/(s^2+2*s+2))
    x = exp(-t)*cos(t)-exp(-t)*sin(t)    % (9) or (11)
>>ilaplace(s/(s^4+10*s^3+9*s^2+6*s+1)) %?? Ns=[ 1 0]; Ds=[ 1 10 9 6 1];
```

A.5 Using the Laplace Transform to Solve Differential Equations

It should be realized how useful the Laplace transform is for dealing with linear ordinary differential equations.

(Example A.3) Solving a Differential Equation

Solve the following differential equation:

$$\frac{d^2}{dt^2}y(t) + 3\frac{d}{dt}y(t) + 2y(t) = 10u_s(t), \quad t \geq 0 \quad (1)$$

with the initial conditions $y(0) = 1$ and $y'(0) = -2$.

Solution

Use is made of the time differentiation properties (A.11) and (A.12) of the Laplace transform to write the Laplace transform of both sides as

$$\begin{aligned} s^2Y(s) - sy(0) - y'(0) + 3[sY(s) - y(0)] + 2Y(s) &= \frac{10}{s} \\ (s^2 + 3s + 2)Y(s) &= sy(0) + y'(0) + 3y(0) + \frac{10}{s} = \frac{10}{s} + s + 1 \end{aligned} \quad (2)$$

This algebraic equation can be solved for $Y(s)$ as

$$Y(s) = \frac{s(s+1) + 10}{s(s^2 + 3s + 2)} \quad (3)$$

This s -domain solution is expanded into partial fractions as

$$Y(s) = \frac{s(s+1) + 10}{s(s^2 + 3s + 2)} = \frac{r_1}{s} + \frac{r_2}{s+1} + \frac{r_3}{s+2} = \frac{5}{s} + \frac{-10}{s+1} + \frac{6}{s+2} \quad (4)$$

where the coefficients are found by using the formula (A.28a) as

$$\begin{aligned} r_1 &= s Y(s) \Big|_{s=0} \stackrel{(3)}{=} \frac{s^2 + s + 10}{s^2 + 3s + 2} \Big|_{s=0} = 5 \\ r_2 &= (s + 1) Y(s) \Big|_{s=-1} \stackrel{(3)}{=} \frac{s^2 + s + 10}{s(s + 2)} \Big|_{s=-1} = -10 \\ r_3 &= (s + 2) Y(s) \Big|_{s=-2} \stackrel{(3)}{=} \frac{s^2 + s + 10}{s(s + 1)} \Big|_{s=-2} = 6 \end{aligned} \quad (5)$$

Thus the time-domain solution $y(t)$ can be written as

$$y(t) = (r_1 e^{-0t} + r_2 e^{-t} + r_3 e^{-2t}) u_s(t) = (5 - 10e^{-t} + 6e^{-2t}) u_s(t) \quad (6)$$

```
>>syms s, Ns=[ 1 1 10] ; Ds=[ 1 3 2 0] ;
>>y=ilaplace(poly2sym(Ns,s)/poly2sym(Ds,s)) % Inverse Laplace transform
>>y1=ilaplace((s^2+s+10)/(s^3+3*s^2+2*s)) % Or, directly from Eq. (3)
>>t=0:0.01:10; yt=eval(y); plot(t,yt), hold on % plot y(t) for t=[ 0,10]
>>[ r,p,k]=residue(Ns,Ds), % Partial fraction expansion
>>yt1=real(r.'*exp(p*t)); plot(t,yt1,'r') % Another way to get y(t)
```

Another alternative way to use MATLAB for solving the differential equation is introduced in Section E.3 in Appendix E. The symbolic differential solver `dsolve()` can be used to solve Equation (1) as follows:

```
>>y=dsolve('D2y+3*Dy+2*y=10','y(10)=1,Dy(0)=-2')
y=5+6*exp(-2*t)-10*exp(-t)
```

Appendix B: Matrix Operations with MATLAB

B.1 Addition and Subtraction

$$A + B = \begin{bmatrix} a_{11} & a_{12} & \cdot & a_{1N} \\ a_{21} & a_{22} & \cdot & a_{2N} \\ \cdot & \cdot & \cdot & \cdot \\ a_{M1} & a_{M2} & \cdot & a_{MN} \end{bmatrix} + \begin{bmatrix} b_{11} & b_{12} & \cdot & b_{1N} \\ b_{21} & b_{22} & \cdot & b_{2N} \\ \cdot & \cdot & \cdot & \cdot \\ b_{M1} & b_{M2} & \cdot & b_{MN} \end{bmatrix} = \begin{bmatrix} c_{11} & c_{12} & \cdot & c_{1N} \\ c_{21} & c_{22} & \cdot & c_{2N} \\ \cdot & \cdot & \cdot & \cdot \\ c_{M1} & c_{M2} & \cdot & c_{MN} \end{bmatrix} = C \quad (\text{B.1a})$$

$$\text{with } a_{mn} + b_{mn} = c_{mn} \quad (\text{B.1b})$$

B.2 Multiplication

$$AB = \begin{bmatrix} a_{11} & a_{12} & \cdot & a_{1K} \\ a_{21} & a_{22} & \cdot & a_{2K} \\ \cdot & \cdot & \cdot & \cdot \\ a_{M1} & a_{M2} & \cdot & a_{MK} \end{bmatrix} \begin{bmatrix} b_{11} & b_{12} & \cdot & b_{1N} \\ b_{21} & b_{22} & \cdot & b_{2N} \\ \cdot & \cdot & \cdot & \cdot \\ b_{K1} & b_{K2} & \cdot & b_{KN} \end{bmatrix} = \begin{bmatrix} c_{11} & c_{12} & \cdot & c_{1N} \\ c_{21} & c_{22} & \cdot & c_{2N} \\ \cdot & \cdot & \cdot & \cdot \\ c_{M1} & c_{M2} & \cdot & c_{MN} \end{bmatrix} = C \quad (\text{B.2a})$$

$$\text{with } c_{mn} = \sum_{k=1}^K a_{mk} b_{kn} \quad (\text{B.2b})$$

Note. For this multiplication to be done, the number of columns of A must equal the number of rows of B .

Note. Note that the commutative law does not hold for the matrix multiplication, i.e. $AB \neq BA$.

B.3 Determinant

The *determinant* of a $K \times K$ (square) matrix $A = [a_{mn}]$ is defined by

$$\det(A) = |A| = \sum_{k=0}^K a_{kn} (-1)^{k+n} M_{kn} \quad \text{or} \quad \sum_{k=0}^K a_{mk} (-1)^{m+k} M_{mk} \quad (\text{B.3})$$

for any fixed $1 \leq n \leq K$ or $1 \leq m \leq K$

where the *minor* M_{kn} is the determinant of the $(K-1) \times (K-1)$ (minor) matrix formed by removing the k th row and the n th column from A and $c_{kn} = (-1)^{k+n} M_{kn}$ is called the *cofactor* of a_{kn} .

Especially, the determinants of a 2×2 matrix $A_{2 \times 2}$ and a 3×3 matrix $A_{3 \times 3}$ are

$$\det(A_{2 \times 2}) = \begin{vmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{vmatrix} = \sum_{k=0}^2 a_{kn} (-1)^{k+n} M_{kn} = a_{11}a_{22} - a_{12}a_{21} \quad (\text{B.4a})$$

$$\begin{aligned} \det(A_{3 \times 3}) &= \begin{vmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{vmatrix} = a_{11} \begin{vmatrix} a_{22} & a_{23} \\ a_{32} & a_{33} \end{vmatrix} - a_{12} \begin{vmatrix} a_{21} & a_{23} \\ a_{31} & a_{33} \end{vmatrix} + a_{13} \begin{vmatrix} a_{21} & a_{22} \\ a_{31} & a_{32} \end{vmatrix} \\ &= a_{11}(a_{22}a_{33} - a_{23}a_{32}) - a_{12}(a_{21}a_{33} - a_{23}a_{31}) + a_{13}(a_{21}a_{32} - a_{22}a_{31}) \end{aligned} \quad (\text{B.4b})$$

Note the following properties:

1. If the determinant of a matrix is zero, the matrix is *singular*.
2. The determinant of a matrix equals the product of the eigenvalues of a matrix.
3. If A is upper/lower triangular having only zeros below/above the diagonal in each column, its determinant is the product of the diagonal elements.
4. $\det(A^T) = \det(A)$, $\det(AB) = \det(A) \det(B)$, $\det(A^{-1}) = 1/\det(A)$

B.4 Inverse Matrix

The *inverse* matrix of a $K \times K$ (square) matrix $A = [a_{mn}]$ is denoted by A^{-1} and defined to be a matrix that is premultiplied/postmultiplied by A to form an identity matrix, i.e. satisfies

$$A \times A^{-1} = A^{-1} \times A = I \quad (\text{B.5})$$

An element of the inverse matrix $A^{-1} = [\alpha_{mn}]$ can be computed as

$$\alpha_{mn} = \frac{1}{\det(A)} c_{mn} = \frac{1}{|A|} (-1)^{m+n} M_{mn} \quad (\text{B.6})$$

where M_{mn} is the minor of a_{mn} and $c_{mn} = (-1)^{m+n} M_{mn}$ is the cofactor of a_{mn} .

Note that a square matrix A is invertible/nonsingular if and only if

- (a) no eigenvalue of A is zero or, equivalently,
- (b) the rows (and the columns) of A are linearly independent or, equivalently,
- (c) the determinant of A is nonzero.

(Example B.1) Matrix Inversion

Let us find the inverse of a 3×3 matrix

$$A = \begin{bmatrix} 3 & -2 & -1 \\ -2 & 5 & -3 \\ -1 & -3 & -8 \end{bmatrix} \quad (1)$$

where the cofactors are

$$\begin{aligned} c_{11} &= (-1)^{1+1} M_{11} = \begin{vmatrix} 5 & -3 \\ -3 & 8 \end{vmatrix} = 5 \times 8 - (-3)(-3) = 31 \\ c_{12} &= (-1)^{1+2} M_{12} = - \begin{vmatrix} -2 & -3 \\ -1 & 8 \end{vmatrix} = -(-2 \times 8 - (-3)(-1)) = 19 \\ c_{13} &= (-1)^{1+3} M_{13} = \begin{vmatrix} -2 & 5 \\ -1 & -3 \end{vmatrix} = -2 \times (-3) - 5(-1) = 11 \\ c_{21} &= (-1)^{2+1} M_{21} = - \begin{vmatrix} -2 & -1 \\ -3 & 8 \end{vmatrix} = -\{-2 \times 8 - (-1)(-3)\} = 19 \\ c_{22} &= (-1)^{2+2} M_{22} = \begin{vmatrix} 3 & -1 \\ -1 & 8 \end{vmatrix} = 3 \times 8 - (-1)(-1) = 23 \\ c_{23} &= (-1)^{2+3} M_{23} = - \begin{vmatrix} 3 & -2 \\ -1 & -3 \end{vmatrix} = -\{3 \times (-3) - (-2)(-1)\} = 11 \end{aligned}$$

$$\begin{aligned}
 c_{31} &= (-1)^{3+1} M_{31} = \begin{vmatrix} -2 & -1 \\ 5 & -3 \end{vmatrix} = (-2) \times (-3) - (-1) \times 5 = 11 \\
 c_{32} &= (-1)^{3+2} M_{32} = - \begin{vmatrix} 3 & -1 \\ -2 & -3 \end{vmatrix} = -(3 \times (-3) - (-1)(-2)) = 11 \\
 c_{33} &= (-1)^{3+3} M_{33} = \begin{vmatrix} 3 & -2 \\ -2 & 5 \end{vmatrix} = 3 \times 5 - (-2)(-2) = 11
 \end{aligned}$$

Equations (B.3) and (B.6) can be used to get the determinant and the inverse matrix as

$$|A| = a_{11}c_{11} + a_{12}c_{12} + a_{13}c_{13} = 3 \times 31 - 2 \times 19 - 1 \times 11 = 93 - 38 - 11 = 44 \quad (2)$$

$$A^{-1} = \frac{1}{|A|} \begin{bmatrix} c_{11} & c_{21} & c_{31} \\ c_{12} & c_{22} & c_{32} \\ c_{13} & c_{23} & c_{33} \end{bmatrix} = \frac{1}{44} \begin{bmatrix} 31 & 19 & 11 \\ 19 & 23 & 11 \\ 11 & 11 & 11 \end{bmatrix} \quad (3)$$

B.5 Solution of a Set of Linear Equations Using the Inverse Matrix

Consider the following set of linear equations in three unknown variables x_1 , x_2 , and x_3 :

$$\begin{aligned}
 3x_1 - 2x_2 - x_3 &= -4 \\
 -2x_1 + 5x_2 - 3x_3 &= -1 \\
 -x_1 - 3x_2 + 8x_3 &= 17
 \end{aligned}$$

This can be formulated in the matrix–vector form

$$\begin{bmatrix} 3 & -2 & -1 \\ -2 & 5 & -3 \\ -1 & -3 & 8 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix} = \begin{bmatrix} -4 \\ -1 \\ 17 \end{bmatrix}; \quad Ax = b$$

so that it can be solved as

$$x = A^{-1}b = \frac{1}{44} \begin{bmatrix} 31 & 19 & 11 \\ 19 & 23 & 11 \\ 11 & 11 & 11 \end{bmatrix} \begin{bmatrix} -4 \\ -1 \\ 17 \end{bmatrix} = \frac{1}{44} \begin{bmatrix} 31 \times (-4) + 19 \times (-1) + 11 \times 17 \\ 19 \times (-4) + 23 \times (-1) + 11 \times 17 \\ 11 \times (-4) + 11 \times (-1) + 11 \times 17 \end{bmatrix} = \begin{bmatrix} 1 \\ 2 \\ 3 \end{bmatrix}$$

B.6 Operations on Matrices and Vectors Using MATLAB

The following statements and their running results illustrate the powerful usage of MATLAB in dealing with matrices and vectors.

```

>>a=[ -2 2 3] % a 1x3 matrix (row vector)
a = -2 2 3
>>b=[ -2; 2; 3] % 3x1 matrix (column vector)
B =
    -2
     2
     3
    
```

```
>>b=a.' % transpose
>>A=[ 1 -1 2; 0 1 0; -1 5 1] % a 3x3 matrix
```

$$A = \begin{bmatrix} 1 & -1 & 2 \\ 0 & 1 & 0 \\ -1 & 5 & 1 \end{bmatrix}$$

```
>>A(1,2) % will return -1
>>A(:,1) % will return the 1st column of the matrix A
ans = -1
      0
     -1
>>A(:,2:3) % will return the 2nd and 3rd columns of the matrix A
ans = -1    2
      1    0
      5    1
>>c=a*A % vector-matrix multiplication
C = -5   19   -1
>>A*b
ans = 2
      2
     15
>>A*a % not permissible for matrices with incompatible dimensions
??? Error using ==>mtimes
Inner matrix dimensions must agree
>>a*c' % Inner product: multiply a with the conjugate transpose of c
ans = 45
>>a.*c % (termwise) multiplication element by element
ans = 10   38   -3
>>a./c % (termwise) division element by element
ans = 0.4000   0.1053  -3.0000
>>det(A) % determinant of matrix A
ans = 3
>>inv(A) % inverse of matrix A
ans = 0.3333    3.6667   -0.6667
      0        1.0000    0
      0.3333   -1.3333    0.3333
>>[ V,E] = eig(A) % eigenvector and eigenvalue of matrix A
V = 0.8165            0.8165            0.9759
      0                0                0.1952
      0+0.5774i        0-0.5774i        0.0976
E = 1.0000+1.4142i      0                0
      0                1.0000-1.4142i      0
      0                0                1.0000
>>I= eye(3) % a 3x3 identity matrix
>>O= zeros(size(I)) % a zero matrix of the same size as I
>>A= sym('[ a b c; d e f]') % a matrix consisting of (non-numeric) symbols
```

```
A=[ a, b, c]
    [ d, e, f]
>>B= [ 1 0; 0 1; 1 1]
B=1    0
    0    1
    1    1
>>A*B
ans = [ a+c,    b+c]
      [ d+f,    e+f]
```

Appendix C: Complex Number Operations with MATLAB

C.1 Addition

$$(a_1 + jb_1) + (a_2 + jb_2) = (a_1 + a_2) + j(b_1 + b_2) \quad (\text{C.1})$$

C.2 Multiplication

$$\text{Rectangular form : } (a_1 + jb_1)(a_2 + jb_2) = (a_1a_2 - b_1b_2) + j(a_1b_2 + b_1a_2) \quad (\text{C.2a})$$

$$\text{Polar form : } r_1 \angle \theta_1 \times r_2 \angle \theta_2 = r_1 e^{j\theta_1} r_2 e^{j\theta_2} = r_1 r_2 e^{j(\theta_1 + \theta_2)} = r_1 r_2 \angle (\theta_1 + \theta_2) \quad (\text{C.2b})$$

C.3 Division

$$\text{Rectangular form : } \frac{a_2 + jb_2}{a_1 + jb_1} = \frac{a_2 + jb_2}{a_1 + jb_1} \frac{a_1 - jb_1}{a_1 - jb_1} = \frac{a_1a_2 + b_1b_2}{a_1^2 + b_1^2} + j \frac{a_1b_2 - a_2b_1}{a_1^2 + b_1^2} \quad (\text{C.3a})$$

$$\text{Polar form : } \frac{r_2 \angle \theta_2}{r_1 \angle \theta_1} = \frac{r_2 e^{j\theta_2}}{r_1 e^{j\theta_1}} = \frac{r_2}{r_1} e^{j(\theta_2 - \theta_1)} = \frac{r_2}{r_1} \angle (\theta_2 - \theta_1) \quad (\text{C.3b})$$

C.4 Conversion between Rectangular Form and Polar/Exponential Form

$$a_r + ja_i = r \angle \theta = r e^{j\theta} \quad \text{with } r = |a_r + ja_i| = \sqrt{a_r^2 + a_i^2} \text{ and } \theta = \tan^{-1} \frac{a_i}{a_r} \quad (\text{C.4})$$

Here, r and θ are referred to as the *absolute value* and *argument* or *phase angle* of the complex number $a_r + ja_i$, respectively, and j is the unit imaginary number $\sqrt{-1}$.

C.5 Operations on Complex Numbers Using MATLAB

If `i` and `j` are not used for any other purpose, they represent the basic imaginary unit $\sqrt{-1}$ by default. Try typing the following statements into the MATLAB command window.

```
>>c1=1+2i; c2= 3-4i;
>>c1*c2 % multiplication of complex numbers
>>c1/c2 % division of complex numbers
>>r=abs(c2) % absolute value of the complex number c2
>>sqrt(real(c2)^2+imag(c2)^2) % equivalent to the absolute value
>>th=angle(c2) % phase angle of the complex number c2 in radians
>>atan2(imag(c2),real(c2)) % equivalent to the phase angle
>>imag(log(c2)) % equivalent to the phase angle
>>th*180/pi % radian-to-degree conversion
>>r*exp(j*th) % polar-to-rectangular conversion
ans = 3.0000 - 4.0000i
```

```
>>C=[ 1+i 1-2i; -1+3i -1-4i] % a complex matrix
C1=1.0000 + 1.0000i 1.0000 - 2.0000i
    -1.0000 + 3.0000i -1.0000 - 4.0000i
>>C1=C' % conjugate transpose
C1=1.0000 - 1.0000i -1.0000 - 3.0000i
    1.0000 + 2.0000i -1.0000 + 4.0000i
```


Appendix D: Nonlinear/Differential Equations with MATLAB

D.1 Nonlinear Equation Solver `<[ x,fx] = fsolve(ftn,x0,options)>`

MATLAB has the built-in function `fsolve(f,x0)`, which can give a solution for a system of (nonlinear) equations. Suppose we have the following system of nonlinear equations

$$\begin{aligned}x_1^2 + 4x_2^2 &= 5 \\ 2x_1^2 - 2x_1 - 3x_2 &= 2.5\end{aligned}\tag{D.1}$$

To solve this system of equations, rewrite it as

$$\begin{aligned}f_1(x_1, x_2) &= x_1^2 + 4x_2^2 - 5 = 0 \\ f_2(x_1, x_2) &= 2x_1^2 - 2x_1 - 3x_2 - 2.5 = 0\end{aligned}\tag{D.2}$$

and convert into a MATLAB function defined in an M-file, say `cir_d02.m`, as follows.

```
function y = cir_d02(x)
y(1) = x(1)*x(1)+4*x(2)*x(2) -5;
y(2) = 2*x(1)*x(1)-2*x(1)-3*x(2) -2.5;
```

Then type the following statements into the MATLAB command window:

```
>>x0=[ 0.8 0.2]; %initial guess[ 0.8 0.2]
>>x=fsolve('cir_d02',x0,optimset('fsolve')) % with default parameters
x=2.0000 0.5000
```

If you see some warning message with the MATLAB solution obtained by using `fsolve()` like this, you can change the value(s) of the optional parameters such as

- `MaxFunEvals`: maximum number of function evaluations allowed
- `MaxIter`: maximum number of iterations allowed
- `TolFun`: termination tolerance on the function value
- `TolX`: termination tolerance on `x`

by using the MATLAB command `optimset()` as follows:

```
options=optimset('param1',value1,'param2',value2,...)
```

For example, if you feel it would contribute to a better solution to set `MaxFunEvals` and `TolX` to 1000 and `1e-8`, respectively, you need to type the following MATLAB statements:

```
>>options=optimset('MaxFunEvals',1000,'TolX',1e-8);
>>x=fsolve('cir_d02',x0,options) % with new values of parameters
x=2.0000 0.5000
```

If you feel like doing a crosscheck, you might use another MATLAB routine `newtons()` (Reference [Y-1], which is defined below.

```
>>x=newtons('cir_d02',x0) % Alternatively,
    x=2.0000  0.5000
```

```
function[ x,fx,xx] = newtons(f,x0,TolX,MaxIter,varargin)
%newtons.m to solve a set of nonlinear eqs f1(x)=0, f2(x)=0,...
%input:   f=1st-order vector ftn equivalent to a set of equations
%         x0=the initial guess of the solution
%         TolX=the upper limit of |x(k)-x(k-1)|
%         MaxIter= the maximum # of iteration
%output: x =the point which the algorithm has reached
%        fx=f(x(last))
%        xx=the history of x
%  Copyleft: Won Y. Yang, wyyang53@hanmail.net, CAU for academic use only
h=1e-4; TolFun=eps; EPS=1e-6;
fx= feval(f,x0,varargin{:}); Nf=length(fx); Nx=length(x0);
if Nf~=Nx, error('Incompatible dimensions of f and x0!'); end
if nargin<4, MaxIter=100; end
if nargin<3, TolX=EPS; end
xx(1,:)=x0(:).'; %Initialize the solution as the initial row vector
for k=1:MaxIter
    dx= -jacob(f,xx(k,:),h,varargin{:})\fx(:); %-[ dfdx ] ^-1*fx
    xx(k+1,:)= xx(k,:)+dx.';
    fx= feval(f,xx(k+1,:),varargin{:}); fxn=norm(fx);
    if fxn< TolFun|norm(dx)<TolX, break; end
end
x= xx(k+1,:);
if k==MaxIter, fprintf('The best in %d iterations\n',MaxIter), end
```

```
function g= jacob(f,x,h,varargin) %Jacobian of f(x)
if nargin<3, h=1e-4; end
h2= 2*h; N=length(x); x=x(:).'; I=eye(N);
for n=1:N
    g(:,n)= (feval(f,x+I(n,:)*h,varargin{:})...
            -feval(f,x-I(n,:)*h,varargin{:}))'/h2;
end
```

D.2 Differential Equation Solver <[t,x] = ode45(odeftn,[t0 tf] ,x0)>

Suppose there is a third-order differential equation

$$x^{(3)}(t) + 7x^{(2)}(t) + 14x'(t) + 8x(t) = \sin(t) \quad (\text{D.3})$$

with the initial conditions $x(0) = 1$, $x'(0) = 2$, and $x^{(2)}(0) = 3$

This can be rewritten as a first-order vector differential equation called a 'state equation' as

$$\begin{bmatrix} x_1'(t) \\ x_2'(t) \\ x_3'(t) \end{bmatrix} = \begin{bmatrix} 0 & 1 & 0 \\ 0 & 0 & 1 \\ -8 & -14 & -7 \end{bmatrix} \begin{bmatrix} x_1(t) \\ x_2(t) \\ x_3(t) \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \\ 1 \end{bmatrix} \sin(t) \quad \text{with} \quad \begin{bmatrix} x_1(0) \\ x_2(0) \\ x_3(0) \end{bmatrix} = \begin{bmatrix} x(0) \\ x'(0) \\ x^{(2)}(t) \end{bmatrix} = \begin{bmatrix} 1 \\ 2 \\ 3 \end{bmatrix} \quad (\text{D.4a})$$

$$x(t) = x_1(t) \quad (\text{D.4b})$$

With this equation cast into a MATLAB function and saved as an M-file named, say `df_apd.m`

```
function dx=df_apd(t,x)
dx= zeros(size(x)); dx(1)=x(2); dx(2)=x(3);
dx(3)= -8*x(1) -14*x(2) - 7*x(3) + sin(t); % (D.3)
```

the following statements can be typed into the MATLAB command window to solve it:

```
>>tspan=[ 0 10]; %time span[ 0,10]
>> x0=[ 1 2 3]; %the initial values of x(0), x'(0), x''(0)
>>[ t, x] =ode45(@df_apd, tspan, x0);
>>plot (t, x(:,1)) %plot x1(t) against t
```

Appendix E: Symbolic Computations with MATLAB

E.1 How to Declare Symbolic Variables and Handle Symbolic Expressions

To declare any variable(s) as a symbolic variable, you should use the `sym` or `syms` command as below:

```
>>a=sym('a'); x=sym('x'); y=sym('y'); t=sym('t'); n=sym('n');
>>syms a x y t n %or, equivalently and more efficiently
```

Once the variables have been declared as symbolic, they can be used in expressions and as arguments to many functions without being evaluated as numeric:

```
>>f=x^2/(1+tan(x)^2);
>>ezplot(f,-pi,pi) %easy plot of f for [-pi,+pi]
>>simplify(cos(x)^2+sin(x)^2) %simplify an expression
ans = 1
>>simplify(cos(x)^2-sin(x)^2) %simplify an expression
ans = 2*cos(x)^2-1
>>simple(cos(x)^2-sin(x)^2) %simple expression
ans = cos(2*x)
>>simple(cos(x)+i*sin(x)) %simple expression
ans = exp(i*x)
>>eq1=expand((x+y)^3-(x+y)^2) %expand
eq1 = x^3+3*x^2*y+3*x*y^2+y^3-x^2-2*x*y-y^2
>>collect(eq1,y) %collect similar terms in descending order w.r.t. y
ans = y^3+(3*x-1)*y^2+(3*x^2-2*x)*y+x^3-x^2
>>factor(eq1) %factorize
ans = (x+y-1)*(x+y)^2
>>horner(eq1) %nested multiplication form
ans = (-1+y)*y^2+((-2+3*y)*y+(-1+3*y+x)*x)*x
>>pretty(ans) %pretty form
      2 y^2+c
(-1 + y) y + ((-2 + 3 y) y + (-1 + 3 y + x) x) x
```

If you need to substitute numeric values or other expressions for some symbolic variables in an expression or take the limit of an expression, you can use `subs()` and `limit()` as below.

```
>>subs(eq1,x,0) %substitute numeric value x=0 into eq1 = (x+y-1)*(x+y)^2
ans = -y^2+y^3
>>subs(eq1,{x,y},{0,x-1}) %substitute x=0 and y=x-1
ans = (x-1)^3-(x-1)^2
>>limit((1+x/n)^n,n,inf) %lim(1+ $\frac{x}{n}$ )^n = e^x
ans = exp(x)
```

E.2 Solving Algebraic Equations

The backslash(\) operator can be used to solve a set of linear equations written in the matrix–vector form:

```
>>syms R11 R12 R21 R22 b1 b2
>>R=[ R11 R12; R21 R22]; b=[ b1; b2];
>>x=R\b % or R^-1*b or inv(R)*b
x = [ (R12*b2-b1*R22)/(-R11*R22+R21*R12)
      [ (-R11*b2+R21*b1)/(-R11*R22+R21*R12)]
```

MATLAB has many commands and functions that can be very helpful in dealing with complex analytic(symbolic) expressions and equations as well as in finding numerical solutions. One of them is `solve()`, which can be used for obtaining the symbolic or numeric roots of equations. According to what can be seen by typing `help solve` into the MATLAB command window, its usages are as follows:

```
>>syms a b c x
>>fx=a*x^2+b*x+c;
>>solve(fx) %formula for roots of 2nd-order polynomial eq
    ans = [ 1/2/a*(-b+(b^2-4*a*c)^(1/2))
           1/2/a*(-b-(b^2-4*a*c)^(1/2))]
>>syms x1 x2 b1 b2
>>fx1=x1+x2-b1; fx2=x1+2*x2-b2; %a system of simultaneous algebraic eq.
>>[ x1o,x2o] = solve(fx1,fx2) %
    x1o = 2*b1-b2
    x2o = -b1+b2
>>solve('p*sin(x)=r') %regarding x as an unknown variable and p as a parameter
    ans=asin(r/p) % sin-1(r/p)
>>[ x1,x2] = solve('x1^2+4*x2^2-5=0','2*x1^2-2*x1-3*x2-2.5=0')
    x1 = [ 2.] x2=[ 0.500000]
          [ -1.206459] [ 0.941336]
          [ 0.603229-0.392630*i] [ -1.095668-0.540415e-1*i]
          [ 0.603229+0.392630*i] [ -1.095668+0.540415e-1*i]
>>S=solve('x^3-y^3=2','x=y') %returns the solution in a structure.
    S = x: [ 3x1 sym]
          y: [ 3x1 sym]
>>S.x
    ans = [ 1]
          [ -1/2+1/2*i*3^(1/2)]
          [ -1/2-1/2*i*3^(1/2)]
>>S.y
    ans = [ -1]
          [ 1/2-1/2*i*3^(1/2)]
          [ 1/2+1/2*i*3^(1/2)]
```

E.3 Solving Differential Equations

The MATLAB function `dsolve()` can be used to solve symbolic differential equations.

```
>>syms a b c x
>>xo=dsolve('Dx+a*x=0') % a differential eq. (d.e.) w/o initial condition
    xo = C1*exp(-a*t) % a solution with undetermined constant
>>xo=dsolve('Dx+a*x=0','x(0)=2') % a d.e. with initial condition
    xo = 2*exp(-a*t) % a solution with determined constant
>>xo=dsolve('Dx=1+x^2') % a differential eq. w/o initial condition
    xo = tan(t+C1) % a solution with undetermined constant
>>xo=dsolve('Dx=1+x^2','x(0)=1') % with the initial condition
    xo = tan(t+1/4*pi) % a solution with determined constant
>>yo=dsolve('D2u=-u','t') % a 2nd-order d.e. without initial condition
    yo = C1*sin(t)+C2*cos(t)
>>xo=dsolve('D2u=-u','u(0)=1,Du(0)=0','t') % with the initial conditions
    xo = cos(t)
```

```

>>yo=dsolve(' (Dy)^2+y^2=1','y(0)=0','x') % a 1st-order nonlinear d.e.
yo=[ sin(x)] %two solutions
[ -sin(x)]
>>S=dsolve('Df=3*f+4*g','Dg=-4*f+3*g'); % a set of differential e
>>f=S.f, g=S.g
f= exp(3*t)*(C1*sin(4*t)+C2*cos(4*t))
g= exp(3*t)*(C1*cos(4*t)-C2*sin(4*t))
>>[ f,g]=dsolve('Df=3*f+4*g,Dg=-4*f+3*g','f(0)=0,g(0)=1')
f= exp(3*t)*sin(4*t)
g=exp(3*t)*cos(4*t)

```

E.4 Differentiating/Integrating Expressions

The MATLAB functions `diff()`/`int()` can be used to differentiate/integrate expressions:

```

>>syms a b x n t
>>f=exp(a*x)*cos(b*t)
>>diff(f) %equivalently diff(f,x)
ans= a*exp(a*x)*cos(b*t) %  $\frac{d}{dx} f = \frac{d}{dx} e^{ax} \cos(bt) = a e^{ax} \cos(bt)$ 
>>diff(f,t)
ans= -exp(a*x)*sin(b*t)*b %  $\frac{d}{dt} f = \frac{d}{dt} e^{ax} \cos(bt) = -b e^{ax} \sin(bt)$ 
>>int(1/(1+x^2))
ans= atan(x) %  $\int \frac{1}{1+x^2} dx = \tan^{-1} x$ 
>>int(sin(a*t),0,pi) %equivalently int(sin(a*t),t,0,pi)
ans= -cos(pi*a)/a+1/a %  $\int_0^\pi \sin(at) dt = -\frac{1}{a} \cos(at) \Big|_0^\pi = -\frac{1}{a} \cos(a\pi) + \frac{1}{a}$ 

```

Appendix F: Useful Formulas (Reference [K-2])

$$\sin(A \pm 90^\circ) = \pm \cos A \quad (\text{F.1})$$

$$\cos(A \pm 90^\circ) = \mp \sin A \quad (\text{F.2})$$

$$\sin(A \pm 180^\circ) = -\sin A \quad (\text{F.3})$$

$$\cos(A \pm 180^\circ) = -\cos A \quad (\text{F.4})$$

$$\sin(A \pm B) = \sin A \cos B \pm \cos A \sin B \quad (\text{F.5})$$

$$\cos(A \pm B) = \cos A \cos B \mp \sin A \sin B \quad (\text{F.6})$$

$$\tan(A \pm B) = \frac{\tan A \pm \tan B}{1 \mp \tan A \tan B} \quad (\text{F.7})$$

$$\sin A \sin B = \frac{1}{2} [\cos(A - B) - \cos(A + B)] \quad (\text{F.8})$$

$$\sin A \cos B = \frac{1}{2} [\sin(A + B) + \sin(A - B)] \quad (\text{F.9})$$

$$\cos A \sin B = \frac{1}{2} [\sin(A + B) - \sin(A - B)] \quad (\text{F.10})$$

$$\cos A \cos B = \frac{1}{2} [\cos(A + B) + \cos(A - B)] \quad (\text{F.11})$$

$$\sin A + \sin B = 2 \sin\left(\frac{A+B}{2}\right) \cos\left(\frac{A-B}{2}\right) \quad (\text{F.12})$$

$$\cos A + \cos B = 2 \cos\left(\frac{A+B}{2}\right) \cos\left(\frac{A-B}{2}\right) \quad (\text{F.13})$$

$$\sin^2 A = \frac{1}{2} (1 - \cos 2A) \quad (\text{F.14})$$

$$\cos^2 A = \frac{1}{2} (1 + \cos 2A) \quad (\text{F.15})$$

$$\sin 2A = 2 \sin A \cos A \quad (\text{F.16})$$

$$\cos 2A = \cos^2 A - \sin^2 A = 1 - 2 \sin^2 A = 2 \cos^2 A - 1 \quad (\text{F.17})$$

$$a \cos A - b \sin A = \sqrt{a^2 + b^2} \cos(A + \theta), \quad \theta = \tan^{-1}\left(\frac{b}{a}\right) \quad (\text{F.18})$$

$$a \sin A + b \cos A = \sqrt{a^2 + b^2} \sin(A + \theta), \quad \theta = \tan^{-1}\left(\frac{b}{a}\right) \quad (\text{F.19})$$

$$\text{Euler identity: } e^{\pm j\theta} = \cos \theta \pm j \sin \theta \quad (\text{F.20})$$

$$e^{j\theta} + e^{-j\theta} = 2 \cos \theta \quad (\text{F.21})$$

$$e^{j\theta} - e^{-j\theta} = j2 \sin \theta \quad (\text{F.22})$$

$$\text{Sum of (infinite) geometric series: } \sum_{n=0}^N r^n = \frac{1 - r^{N+1}}{1 - r} \xrightarrow{N \rightarrow \infty} \sum_{n=0}^{\infty} r^n = \frac{1}{1 - r}, \quad |r| < 1 \quad (\text{F.23})$$

$$\text{Binomial expansion : } (a + b)^N = \sum_{n=0}^N N C_n a^{N-n} b^n \text{ with } N C_n = N C_{N-n} = \frac{N P_n}{n!} = \frac{N!}{(N-n)!n!} \quad (\text{F.24})$$

$$e^x = \sum_{n=0}^{\infty} \frac{1}{n!} x^n = 1 + \frac{1}{1!} x + \frac{1}{2!} x^2 + \frac{1}{3!} x^3 + \dots \xrightarrow{x \rightarrow 0} e^x = 1 + x \quad (\text{F.25})$$

$$\frac{d}{dt} t^n = n t^{n-1} \quad (\text{F.26})$$

$$\frac{d}{dt} e^{at} = a e^{at} \quad (\text{F.27})$$

$$\frac{d}{dt} \cos \omega t = -\omega \sin \omega t \quad (\text{F.28})$$

$$\frac{d}{dt} \sin \omega t = \omega \cos \omega t \quad (\text{F.29})$$

$$\frac{d}{dt} (uv) = u \frac{d}{dt} v + v \frac{d}{dt} u \quad (\text{F.30})$$

$$\frac{d}{dt} \left(\frac{v}{u} \right) = \frac{u(dv/dt) - v(du/dt)}{u^2} \quad (\text{F.31})$$

$$\int t^n dt = \frac{1}{n+1} t^{n+1} \text{ for } n \neq -1 \quad (\text{F.32})$$

$$\int e^{at} dt = \frac{1}{a} e^{at} \quad (\text{F.33})$$

$$\int \cos \omega t dt = \frac{1}{\omega} \sin \omega t \quad (\text{F.34})$$

$$\int \sin \omega t dt = -\frac{1}{\omega} \cos \omega t \quad (\text{F.35})$$

$$\int u \frac{dv}{dt} dt = uv - \int v \frac{du}{dt} dt \text{ (partial integral)} \quad (\text{F.36})$$

Appendix G: The Standard Values of Resistors, Capacitors, and Inductors

G.1 Color Code of Resistors

Except for wire-wound/cermet/high-power/precision resistors, most common resistors do not have their resistance value printed on them, but rather have a color code representing their resistance value, as illustrated in Figure G.1. Table G.1 shows the numerical value or tolerance (manufacturer’s reliability rating) represented by each color. For example, the resistance value of a resistor with the four-color band of yellow–violet–red–silver is

$$R = (10a + b) \times 10^m \pm \% = (10 \times 4 + 7) \times 10^2 \pm 10\% = 4700 \pm 470 \Omega \tag{G.1}$$

and that of a resistor with the five-color band of orange–black–white–gold–gold is

$$R = (100a + 10b + c) \times 10^m \pm \% = (100 \times 3 + 10 \times 0 + 1) \times 10^{-1} \pm 5\% = 30.1 \pm 0.15 \Omega \tag{G.2}$$

Note. Visit the website <<http://xtronics.com/kits/rcode.htm>> for more details about the color code.

G.2 Standard Values of Resistors

Discrete resistors are commercially available only in standard values depending on their tolerance, as listed in Table G.2. Consequently, when the designed value of a resistor is 3.1 kΩ, then $30 \times 10^2 \pm 5\% \Omega$ or $309 \times 10^1 \pm 1\% \Omega$ should be used unless somehow a resistor of 3.1 kΩ could be fabricated.

Note. The commercially available resistors have the power (wattage) ratings of 1/8, 1/4, 1/2, 1, and 2 W depending on their physical sizes.

Note. Some resistors may exhibit parasitic effects of series inductance and parallel capacitance, especially if the circuit containing them operates at a very high frequency.

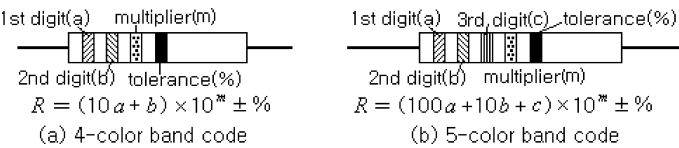


Figure G.1 Color code for resistors

Table G.1 Color code of resistors

Color	Digit	Color	Digit	Color	Tolerance
Black	0	Blue	6	Brown	1 %
Brown	1	Violet	7	Red	2 %
Red	2	Gray	8	Gold	5 %
Orange	3	White	9	Silver	10 %
Yellow	4	Gold	–1(applied only to multiplier <i>m</i>)	None	20 %
Green	5	Silver	–2(applied only to multiplier <i>m</i>)		

Table G.2 Standard values of resistors

1 % tolerance												5 % tolerance				10 % tolerance		20 % tolerance	
100	121	147	178	215	261	316	383	464	562	681	825	10	18	33	56	10	33	10	
102	124	150	182	221	267	324	392	475	576	698	845	11	20	36	62	12	39	15	
105	127	154	187	226	274	332	402	487	590	715	866	12	22	39	68	15	47	22	
107	130	158	191	232	280	340	412	499	604	732	887	13	24	43	75	18	56	33	
110	133	162	196	237	287	348	422	511	619	750	909	15	27	47	82	22	68	47	
113	137	165	200	243	294	357	432	523	634	768	931	16	30	51	91	27	82	68	
115	140	169	205	249	301	365	442	536	649	787	953								
118	143	174	210	255	309	374	453	549	665	806	976								

G.3 Standard Values of Capacitors

Discrete capacitors are commercially available only in standard values depending on their physical material/shape, as listed in Tables G.3 and G.4. Table G.5 shows the letter tolerance code of capacitors. Most of them have their value (like 22 μF) printed on their body together with their breakdown voltage, while the capacitance value of a ceramic condenser is printed as, say, 104, which means the capacity of

$$10 \times 10^4 \text{ pF} = 10^5 \times 10^{-12} \text{ F} = 0.1 \times 10^{-6} \text{ F} = 0.1 \mu\text{F}$$

Table G.3 Standard values of electrolytic capacitors [μF]

Maximum voltage 10 V				Maximum voltage 25 V			Maximum voltage 50 V				
	100	1000	10 000	10	100	1000	0.1	1.0	10	100	1000
22	220	2200		22	220	2200	0.22	2.2	22	220	2200
33	330	3300		33	330	3300	0.33	3.3	33	330	
47	470	4700		47	470	4700	0.47	4.7	47	470	
		6800									

Table G.4 Standard values of ceramic and mylar polyester capacitors

Ceramic disc capacitors [pF] with maximum voltage 200 V				Mylar polyester capacitors [μF] with maximum voltage 100V			
10	100	1000	10 000	0.001	0.01	0.1	0.33
15	150	1500	15 000	0.0015	0.015	0.12	0.39
22	220	2200		0.0022	0.022	0.15	0.47
33	330	3300		0.0033	0.033	0.18	0.56
47	470	4700		0.0047	0.047	0.22	0.68
68	680	6800		0.0068	0.068	0.27	0.82
				0.0082	0.082		1.0

Table G.5 Letter tolerance code of capacitors [%]

B	C	D	E	F	G	H	J	K	M	N	P	Z
+0.1	+0.25	+0.5	+0.5	+1	+2	+3	+5	+10	+20	+0.05	+100	+80
-0.1	-0.25	-0.5	-0.5	-1	-2	-3	-5	-10	-20	-0.05	-0	-20

If the tolerance is missing, it can be assumed to be $\pm 20\%$. Note that electrolytic capacitors have positive/negative terminals with the positive one having a longer leg. If care is not taken to connect them in accordance with the polarity, they will leak or may be destroyed.

Note. The commercially available capacitors have working voltages of about 3~1000 V.

Note. Some capacitors may exhibit parasitic effects of parallel/series resistance and series inductance.

G.4 Standard Values of Inductors

Discrete inductors are commercially available only in standard values (with tolerances of $\pm 5\%$, $\pm 10\%$, and $\pm 20\%$), listed in Table G.6.

Note. Most inductors exhibit parasitic effects of series resistance (due to the large number of turns) and parallel capacitance.

Note. Visit the website <http://www.rfcafe.com/references/electrical/inductor_values.htm> for more details about the standard values of inductors.

For the desired values of the resistances, capacitances, and inductances, the standard values can easily be chosen from Tables G.2, G.3, G.4, and G.6 by using the following MATLAB routine `standard_value(val,RLC,glc,tol)`, where the input arguments are supposed to be given as follows:

- `val` : the desired value of a resistance, a capacitance, or an inductance
- `RLC` : 'R', 'C', or 'L' for resistance, a capacitance, or an inductance
- `glc` : 'g', 'l', or 'c' to look for the standard value slightly greater than, less than, or just close to the desired value
- `tol` : 1, 5, or 10 for the 1%/5%/10% tolerance standard resistance value

Table G.6 Standard values of inductors

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	8.7	9.1 nH, μ H
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	8.7	9.1 $\times 10$ nH, μ H
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	8.7	9.1 $\times 10^2$ nH, μ H
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	8.7	9.1 $\times 10^3$ nH, μ H

```

function std_val = standard_value(val,RLC,glc,tol)
%Input: R= the value of desired Resistance/Inductance/Capacitance
%
%       RLC= 'R' / 'L' / 'C'
%       glc= 'g' / 'l' / 'c' for greater than or equal/less than or equal/close
%       tol= 1/5/10 for error tolerance
if nargin<4, tol=1; end
if nargin<3, glc= 'c' ; end
if nargin<2, RLC= 'R' ; end
Rs1=[ 100 102 105 107 110 113 115 118 121 124 127 130 133 137 140 143 147] ;
Rs1=[ Rs1 150 154 158 162 165 169 174 178 182 187 191 196 200 205 210 215] ;
Rs1=[ Rs1 221 226 232 237 243 249 255 261 267 274 280 287 294 301 309 316] ;
Rs1=[ Rs1 324 332 340 348 357 365 374 383 392 402 412 422 432 442 453 464] ;
Rs1=[ Rs1 475 487 499 511 523 536 549 562 576 590 604 619 634 649 665 681] ;
Rs1=[ 698 715 732 750 768 787 806 825 845 866 887 909 931 953 976] ;
Multipliers= 10.^[-2:9]; % Table G.1
Rs1=Multipliers'*Rs1; Rs1=Rs1(:)'; % 1%-tolerance (Table G.2)
Rs=[ 10 11 12 13 15 16 18 20 22 24 27 30 33 36 39 43 47 51 56 62 68 75 82 91] ;
Rs5=Multipliers'*Rs5; Rs5=Rs5(:)'; % 5%-tolerance (Table G.2)
Rs10=[ 10 12 15 18 22 27 33 39 47 56 68 82] ;
Rs10=Multipliers'*Rs10; Rs10=Rs10(:)'; % 10%-tolerance (Table G.2)
Cs1=[ 10 15 22 33 47 68 100 150 220 330 470 680 1000 1500 2200 3300 4700 6800] ;
Cs1=[ Cs1 10000 15000]*1e-12;
Cs2=[ 0.001 0.0015 0.0022 0.0033 0.0047 0.0068 0.0082 0.01 0.015 0.022 0.033] ;
Cs2=[ Cs2 0.047 0.068 0.082 0.1 0.12 0.15 0.18 0.22 0.27 0.33 0.39 0.47 0.56] ;
Cs2=[ 0.68 0.82 1.2 2.3 3.4 7 10 22 33 47 100 220 330 470 1000 2200 3300 4700] ;
Cs2=[ Cs2 6800 10000]*1e-6; Cs=[ Cs1 Cs2] ;
Ls0=[ 1 1.1 1.2 1.3 1.5 1.6 1.8 2.2 2.4 2.7 3 3.3 3.6 3.9 4.3 4.7 5.1 5.6 6.2] ; Ls0=[ Ls0
6.8 7.5 8.2 8.7 9.1] ;
Ls=[ Ls0*1e-6 Ls0*1e-9] ; Ls=[ 1; 10; 100]*Ls; Ls=[ Ls(:)' Ls0*1e-3] ;
switch lower(RLC)
case 'r' % Resistor
    if tol == 5
        if lower(glc(1)) == 'g', ind=find(Rs5>=val); list=Rs5(ind);
        elseif lower(glc(1)) == 'l', ind=find(Rs5<=val); list=Rs5(ind);
        else list=Rs5;
    end
elseif tol == 10
    if lower(glc(1)) == 'g', ind=find(Rs10>=val); list=Rs10(ind);
    elseif lower(glc(1)) == 'l', ind=find(Rs10<=val); list=Rs10(ind);

else list=Rs10;
    end
    else % if tol=1%
if lower(glc(1)) == 'g', ind=find(Rs1>=val); list=Rs1(ind);
    elseif lower(glc(1)) == 'l', ind=find(Rs1<=val); list=Rs1(ind);
    else list=Rs1;
    end
end
case 'l' % Inductor
    if lower(glc(1)) == 'g', ind=find(Ls>=val); list=Ls(ind);

```

```
elseif lower(glc(1)) == 'l' , ind=find(Ls<=val); list=Ls(ind);
else list=Ls;
end
case 'c' % Capacitor
    if lower(glc(1)) == 'g' , ind=find(Cs>=val); list=Cs(ind);
    elseif lower(glc(1)) == 'l' , ind=find(Cs<=val); list=Cs(ind);

    else list=Cs;
    end
otherwise error('What do you want, neither R nor L nor C?')
end
[em,im] = min(abs(list-val)); std_val=list(im);
```

Appendix H: OrCAD/PSpice (References [K-1] and [R-2])

OrCAD PSpice, formerly MicroSim PSpice (Professional Simulation Program with Integrated Circuit Emphasis), is a powerful circuit simulation program with graphic interface to analyze electric and electronic circuits. The OrCAD demo software can be downloaded together with the OrCAD flow tutorial at the web site <<http://www.orcad.com/download.orcaddemo.aspx>>.

H.1 Starting Capture CIS Session

Once the OrCAD software has been installed, the Capture CIS (Component Information System) Demo session may be started by clicking the corresponding icon to open the Capture CIS window (Figure H.1), in which File/New/Project on the menu bar can be clicked to open the New Project dialog box. Here, you are supposed to do the following in the New Project dialog box:

- In the Name field, type, say, ‘testa’ (with no quotation mark) as a new project name.
- Check the radio button of Analog or Mixed A/D to select it.
- In the Location field, type the pathname of the subdirectory in which you want the new project to be saved. You can use the Browse button to select the location for the project files and click OK.
- Then click OK to close the dialog box and click OK in the Create PSpice Project box.

Then you will see the activated Capture (CIS) window with the menu bar together with the tool bar, which has the Project Manager window containing the design file name ‘testa.dsn’ under the subdirectory named ‘Design Resources’ on the left side (Figure H.2). You can click the design file name and, successively, SCHEMATIC1 and PAGE1 to open the Schematic Editor window, in which you can draw the schematic for the circuit that you would like to analyze.

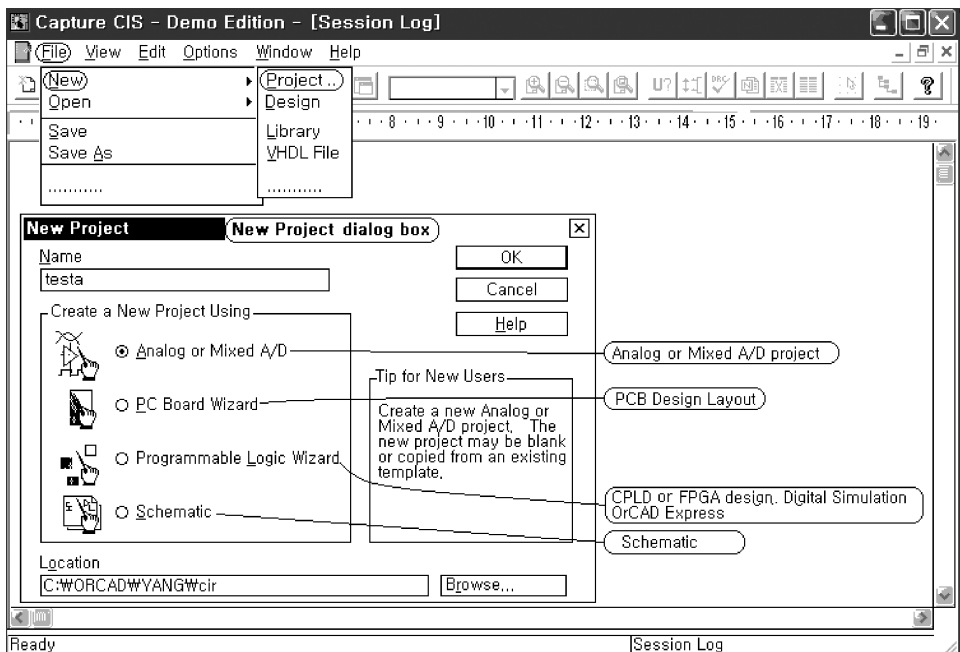


Figure H.1 New Project dialog box opened by clicking File/New/Project in the Capture CIS window

H.2 Drawing Schematic

H.2.1 Fetching Parts (Place/Part or ‘p’)

Click Place/Part on the menu bar or the Place Part button on the tool palette, or just press the ‘p’ key on the keyboard to open the Place Part dialog box (Figure H.2), in which you can either scroll through the part list and select (click) the needed part or just type the part name into the Part field in the upper part of the dialog box. It may be helpful for you to know the part names representing some often-used device types listed in Table H.1. In case you cannot find the device (component) you want to place from the current part list, you need to search for it after clicking the Part Search button and then add the library containing the device.

H.2.2 Placing Parts

Once the wanted part is found, which can be inspected from the graphic symbol displayed in the preview box at the bottom-right corner of the Place Part dialog box, click OK to close the dialog box. Then the selected part attaches to the mouse pointer so that it can be moved and placed anywhere on the schematic page by clicking. The device will remain attached to the mouse pointer, so may be used as many times as needed. To detach the device from the mouse pointer, press the ESC key or right-click and click (select) End Mode in the pop-up menu. One of the recently fetched devices can be attached to the mouse pointer by clicking (selecting) it among the MRU list in the center part of the tool bar just below the top menu bar. Any part on the schematic can be selected by clicking it so that it gets pink-hot and then rotated

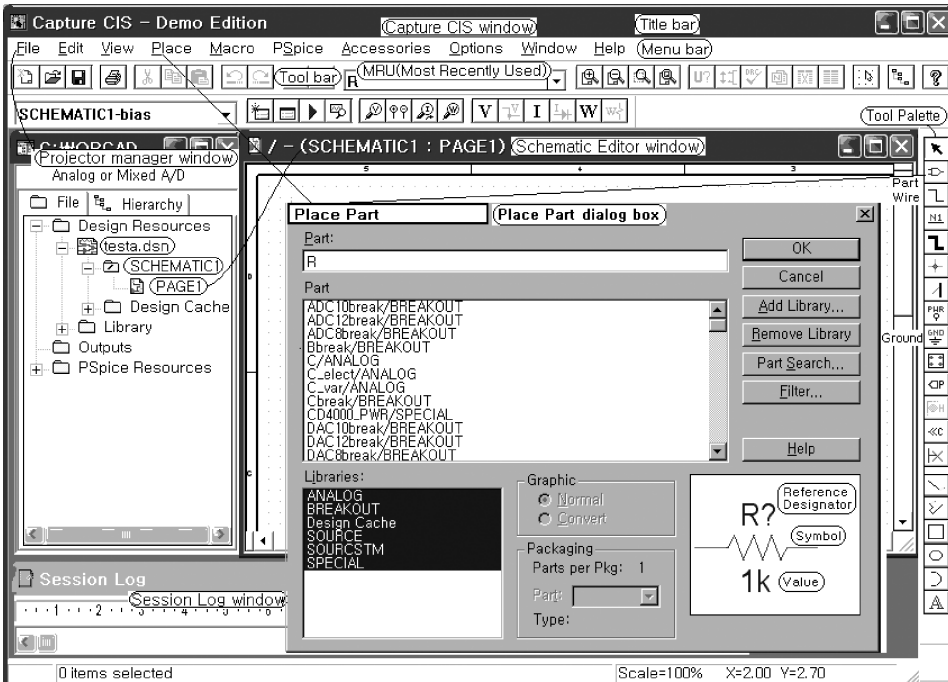


Figure H.2 Capture window, Project Manager window, Schematic Editor window, and Place Part box opened by clicking the Place Part icon in the tool Palette

Table H.1 Device types and PSpice part names

Device type	Part name	Device type	Part name
Capacitor	C	Inductor	L
Diode	D	BJT (bipolar junction transistor)	Q
Voltage-controlled voltage source (VCVS)	E	Resistor	R
Current-controlled voltage source (CCVS)	F	Switch to be closed at specified time	Sw_tClose
Voltage-controlled current source (VCCS)	G	Switch to be opened at specified time	Sw_tOpen
Current-controlled current source (CCCS)	H	OP Amp	uA741
Independent current source	I	Independent voltage source	V
Independent AC current source	IAC	Independent AC voltage source	VAC
Independent DC current source	IDC	Independent DC voltage source	VDC
Independent periodic current source	IPULSE	Independent periodic voltage source	VPULSE
Independent piecewise linear current source	IPWL	Independent piecewise linear voltage source	VPWL
Sinusoidal current source	ISIN	Sinusoidal voltage source	VSIN
		Transformer	XFRM_LINEAR

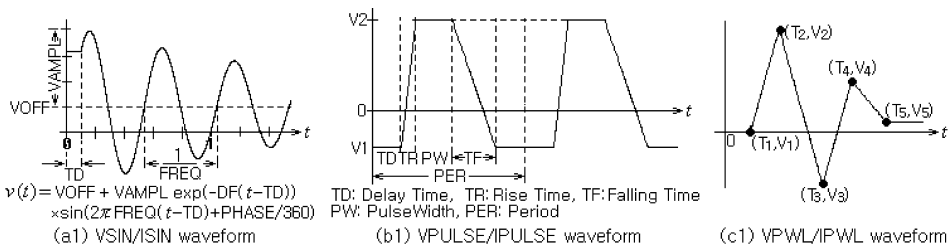
counterclockwise by pressing the ‘r’ key or mirrored horizontally/vertically by pressing the ‘h’/‘v’ key. Any area of the schematic can also be selected by clicking one corner of the area and dragging (pressing/holding the left mouse button) it to the opposite corner. You can then move (click&drag), cut (‘^x’), paste (‘^v’), delete (Del), rotate (‘r’), or mirror horizontally (‘h’)/vertically (‘v’) the selected (pink-hot) area, where ^ indicates ‘with the CTRL key held down’. You can zoom in/out the whole schematic by pressing the ‘i’/‘o’ key. After placing all the parts at the appropriate locations, you should click the GND (ground) button on the tool palette, click (select) 0/SOURCE in the Place Ground dialog box, click OK to close the dialog box, and move the mouse pointer (with the Ground symbol attached to it) to click at an appropriate location to place the ground symbol at that location.

[Remark H.1] Difference between VSIN and VAC

The two parts VSIN/ISIN(sine wave V/I source) and VAC/IAC(AC V/I source) are represented by the graphic symbols of the same shape, but with different parameters. Specifically, compared with VAC, VSIN has an additional parameter, which is the frequency in Hz, while the frequency range of VAC is specified in the simulation profile. In fact, VAC is used for AC sweep (steady state) analysis to find the frequency response, while VSIN is for time-domain (transient) analysis.

H.2.3 Defining(Assigning)/Changing Part Values

You can double-click the value or reference designator of a part to define/change it in the Display Properties dialog box, or double-click the part to open its Property Editor spreadsheet, define or change the values of the parameters, and even create a new parameter (New Column) and set its value. Figure H.3 shows the waveforms of sine/pulse/piecewise-linear/digital sources and their Property Editor spreadsheets, in which the parameters are defined and changed. Sometimes, the value of IC for inductors/capacitors should be set, where IC denotes the initial current/voltage for the inductor/capacitor, respectively. Table H.2 shows magnitude identifiers used by PSpice.



Property Editor												
New Column...			Apply		Display...		Delete Property		Filter		< Current properties >	
											Help	
1	SCHEMATIC1: PAGE1: V2	AC	Color	DC	DF	FREQ	ID	Part Reference	PHASE	Source Part	TD	Value
			Default		0			V2	0	VSIN.Normal	0	VAMPL
												VOFF

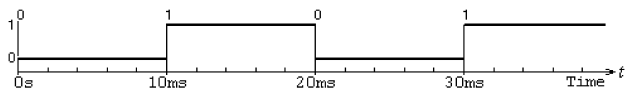
(c) The value of DC/AC is needed for DC/AC analysis, respectively.
(a2) Property Editor spreadsheet for VSIN

Property Editor												
New Column...			Apply		Display...		Delete Property		Filter		< Current properties >	
											Help	
		PER	PW	Part Reference	Source Part	TD	TF	TR	V1	V2	Value	
1	SCHEMATIC1: PAGE1: V3			V3	VPULSE.Normal						VPULSE	

(b2) Property Editor spreadsheet for VPULSE

Property Editor												
New Column...			Apply		Display...		Delete Property		Filter		< Current properties >	
											Help	
		Part Reference	Source Part	T1	T2	T3	T4	V1	V2	V3	V4	Value
1	SCHEMATIC1: PAGE1: V4	V4	VPWL.Normal									VPWL

(c2) Property Editor spreadsheet for VPWL



(d1) Typical waveform generated by digital voltage source STIM1

		COMMAND1	COMMAND2	COMMAND3	COMMAND4	Part Reference	Source Part	TIMESTEP	Value
1	SCHEMATIC1: PAGE1: DSTM1	0ms 0	10ms 1	20ms 0	30ms 1	DSTM1	STIM1.Normal		STIM1

(d2) Property Editor spreadsheet for digital voltage source STIM1

Figure H.3 Property Editor spreadsheets

Table H.2 Magnitude identifiers used by PSpice (Capture)

Unit identifier	F or f (hertz)	P or p (pico)	N or n (nano)	U or u (micro)	M or m (milli)	K or k (kilo)	MEG or meg (mega)	G or g (giga)	T or t (tera)
Magnitude	10 ⁻¹⁵	10 ⁻¹²	10 ⁻⁹	10 ⁻⁶	10 ⁻³	10 ³	10 ⁶	10 ⁹	10 ¹²

Note. PSpice does not require the units in specifying the part values, but if unit is shown, there must be no space between the value and the unit. In fact, PSpice does not differentiate the uppercase and lowercase letters.

H.2.4 Wiring Place/Wire or 'w'

In order to connect the parts here and there in the Schematic page, click the menu Place/Wire on the menu bar or the Place Wire button on the tool palette, or just press the 'w' key on the keyboard to change the arrow-type pointer into a cross-type pointer, which can be used like a pencil. Then click the one point of the connection you want to make, click successively the intermediate points, and either double-click the other point or right-click the other point and click (select) End Wire in the pop-up box. A slant line can be drawn by holding down the SHIFT key and clicking the connection points. To stop wiring, press the ESC key or right-click and click (select) End Wire in the pop-up box.

[Remark H.2] Connections Rejected by PSpice

Figure H.4 shows some examples of connection that are rejected by PSpice, together with the corresponding countermeasures:

1. PSpice insists that every node should have a DC path to ground for a proper analysis, but the top node of the circuit in Figure H.4(a1) does not satisfy the requirement, which is the case of missing DC path to ground. It can be circumvented by connecting a very large-valued resistor of, say, $1\text{ G}\Omega$ between the node, and the ground node, as depicted in Figure H.4(a2).
2. The secondary side of the transformer in the circuit of Figure H.4(b1) has no DC path to ground, dissatisfying the above requirement. It can also be circumvented by connecting a very large-valued dummy resistor between the two nodes of primary and secondary coils, as depicted in Figure H.4(b2).
3. PSpice does not accept any loop consisting of only voltage source(s) and/or inductor(s) that may result in a DC short. This condition is violated by the circuit in Figure H.4(c1). This case of voltage source and/or inductor loop can be avoided by inserting a very small-valued dummy resistor of, say, $1\text{ }\mu\Omega$ into the loop, as depicted in Figure H.4(c2).

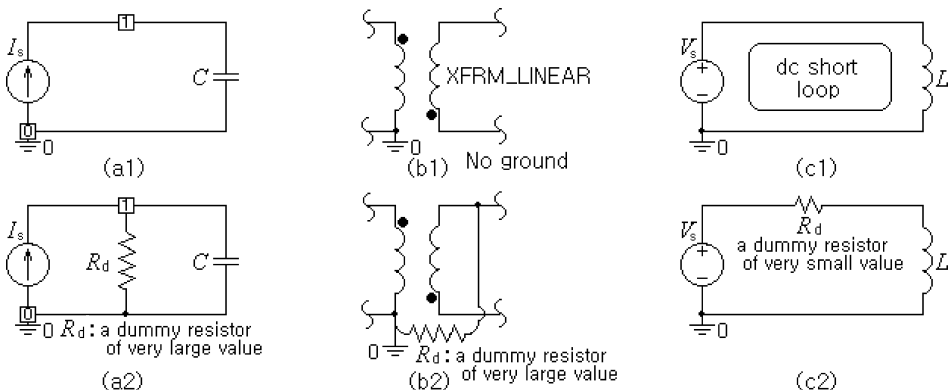


Figure H.4 Examples of connections rejected by PSpice and the corresponding measures

H.3 Setting Simulation Conditions

H.3.1 Creating a Simulation Profile

To set the simulation conditions, click **P**Spice/**N**ew **S**imulation **P**rofile on the menu bar or the corresponding button on the tool bar to open the New Simulation dialog box, in which you can type a profile name, say 'tran' (with no quotation mark), into the Name field and click the Create button (Figure H.5(a)). Then the Simulation Settings dialog box is opened, in which you can do the following:

- Click on the General tab to set the output directory and waveform data filename (Figure H.5(b))
- Click the Analysis tab to set the Analysis type to one of {Time Domain (Transient), DC Sweep, AC Sweep, and Bias Point}, select the Options, and set the simulation time/frequency range, etc. (Figure H.5(c1) or (c2)). Especially in the case of Transient analysis, you can click the Output File Options button to make the results of Fourier analysis and/or operating bias point analysis printed into the output file.
- Click the Data Collection tab to select Data Collection Options (Figure H.5(d)).
- Click the Probe Window tab to set the conditions on the PSpice A/D (Probe) window (Figure H.5(e)).

More often, you are supposed to do only the second thing with the Analysis type and the related parameters.

H.3.2 Placing Voltage/Current Markers (Probes) – PSpice/Markers

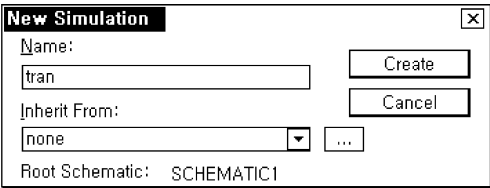
To obtain the voltage/current/power waveform, you can click **P**Spice/**M**arkers on the menu bar and select one of {Voltage Level, Voltage Differential, Current, Power Dissipation, Advanced (dB, Phase, Real, Imaginary)} from the submenu or click one of {V, Vd, I, W}-Marker buttons on the tool bar to attach the probe pins to the mouse and then put one at each target point/device on the schematic. The probe pin will remain attached to the mouse pointer, so it may be used as many times as needed. To detach the probe pin from the mouse pointer, press the ESC key or right-click and click (select) End Mode in the pop-up menu. Alternatively, the Add Trace function in the Probe window can be used in order to take measurements of any voltage/current/power without using schematic markers. Specifically, you can click **T**race/**A**dd_Trace on the menu bar or the Add Trace button on the tool bar of the Probe window to open the Add Traces dialog box, select the variables from the Simulation Output Variables list, and click OK. On the other hand, you can have all the values of voltage/currents (at the operating point) displayed/hidden on the schematic by clicking the Enable Bias Voltage(V)/Current(I)/Power(W) Display button on the tool bar of the Schematic Editor window.

H.3.3 Editing Simulation Profile

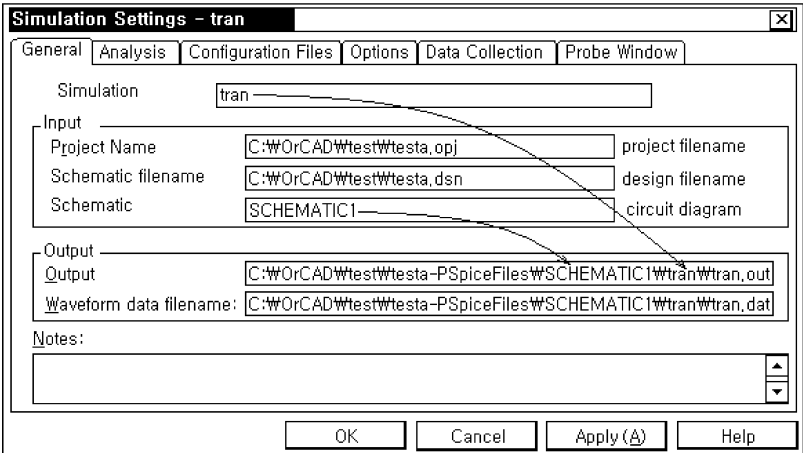
To modify the simulation conditions, either click **P**Spice/**E**dit Simulation **P**rofile on the menu bar or the corresponding button on the tool bar in the Capture window, or click **S**imulation/**E**dit **P**rofile on the menu bar or the corresponding button on the (left) tool bar in the Probe window to reopen the Simulation Settings dialog box, in which the simulation conditions can be changed as well as the related parameters. In case there is any existing Simulation Profile, select it from the Simulation Profile list on the left side of the tool bar in the Capture window.

H.4 Running PSpice Simulation and Observing the Results

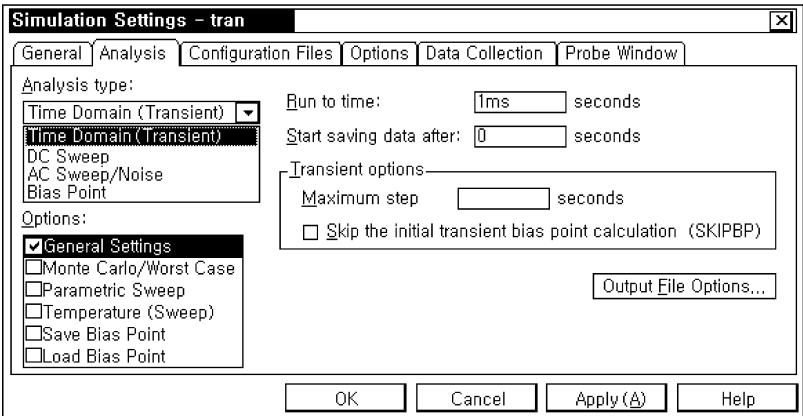
To perform the simulation of the circuit you have drawn in the schematic page, either click **P**Spice/**R**un on the menu bar or the corresponding button (of a right arrow shape) on the tool bar in the Capture



(a) Creating a simulation profile

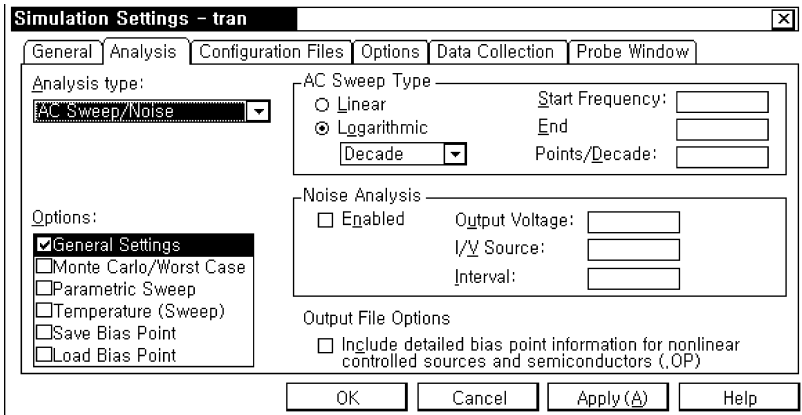


(b) Setting the output directory and filename

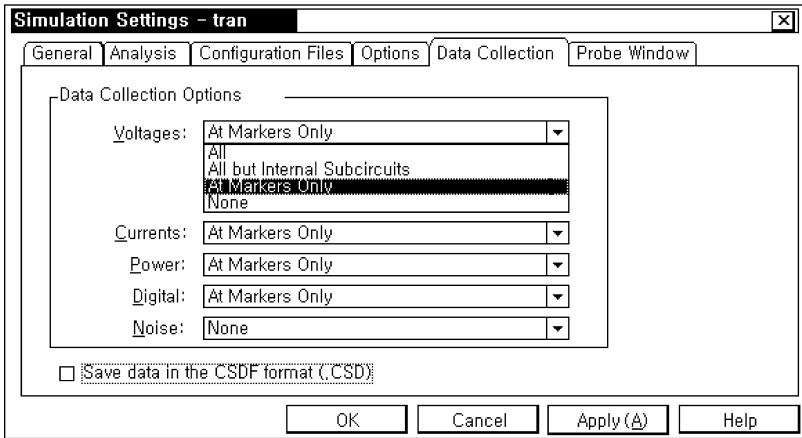


(c1) Setting the Analysis type (as Transient) and parameters

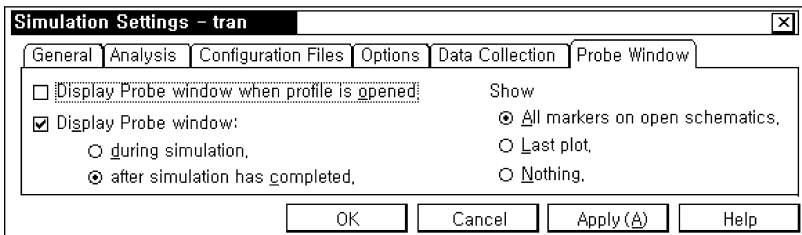
Figure H.5 Setting the simulation condition in the Simulation Settings dialog box



(c2) Setting the Analysis type (as AC Sweep) and parameters

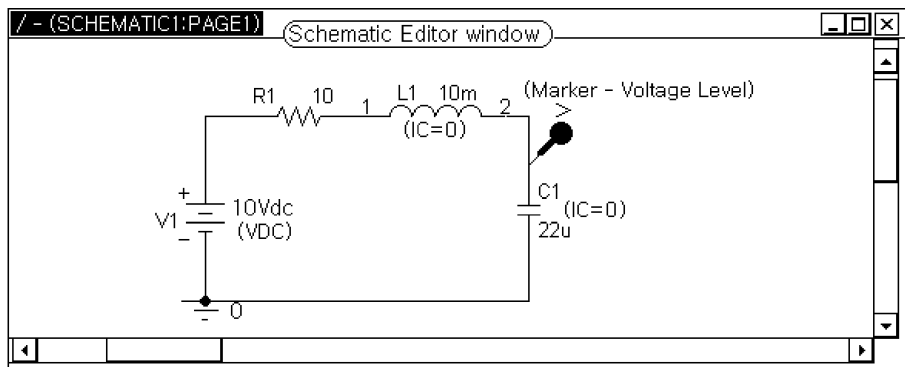


(d) Setting the Data Collection options

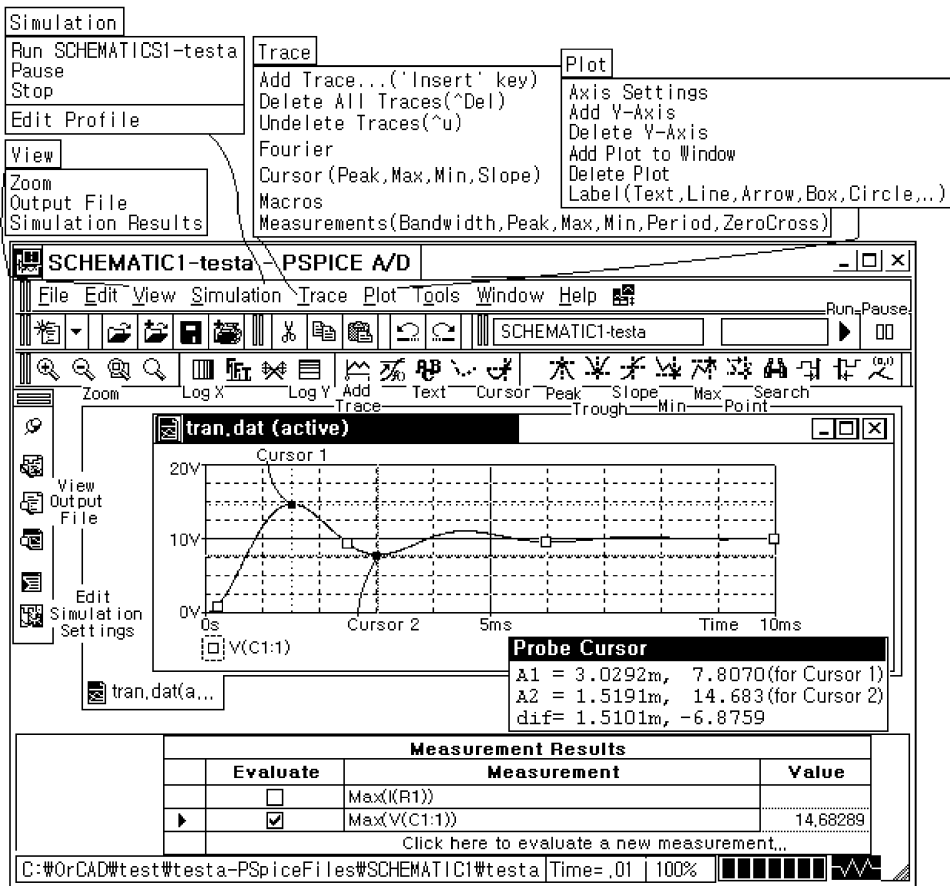


(e) Setting the PSpice A/D (Probe) window options

Figure H.5 (Continued)



(a) A schematic with the DC analysis result displayed in the schematic window



(b) A typical signal waveform seen in the PSpice A/D (Probe) window

Figure H.6 A typical schematic and its simulation result

window, or click Simulation/Run on the menu bar or the corresponding button on the right side of the tool bar in the Probe window. Pressing the 'F11' key is another way to perform the simulation. Then you will see the waveforms of the voltages/currents/powers that you have chosen as the measurement outputs by placing the corresponding Markers. Besides, you can find their peaks, troughs, minimum, maximum, etc., as if you had a digital storage oscilloscope.

H.4.1 Analyzing Waveforms – Trace/Cursor

Suppose the schematic of an *RLC* circuit has been drawn as in Figure H.6(a) and the simulation performed to get the result as in Figure H.6(b) in the PSpice A/D (Probe) window. Then you can click Trace/Cursor on the menu bar or the Toggle Cursor button on the tool bar in the Probe window to have two cross-type cursors appear on a waveform, each of which can be moved either by left/right-clicking at any desired position or by pressing the left or right Arrow/Shift+Arrow key, where the coordinates of the cursor points on the waveform together with their pairwise differences can be seen in the Probe Cursor box. In case there are several waveforms in the Probe window, you can select the waveform to be traced using each cursor by left/right-clicking on the graphic symbol next to the name of the variable in the legend (just below the waveform graph) that you want to trace. Once you have moved a cursor, you can find the peak/trough/minimum/maximum... of the waveform associated with that cursor by clicking Peak/Trough/Min/Maxl... on the menu bar or the corresponding button on the toolbar. You can also click Trace/Evaluate_Measurement on the menu bar or the Evaluate Measurement button on the toolbar to open the Evaluate Measurement dialog box and select a desired measurement (such as the maximum, the minimum, the center frequency, the bandwidth, etc.) and a simulation output variable so that the target trace expression will appear in the Trace Expression field at the bottom of the Evaluate Measurement dialog box as illustrated in Figure H.7. (The target trace expression may be directly typed into the Trace Expression field.) Then you can click OK to have the desired measurement results appear at the bottom of the Probe window, as illustrated in Figure H.6(b).

H.4.2 Adding/Removing Waveforms – Trace/Add_Trace or 'Ins'/Del' key

To see the waveform of another variable (Figure H.8(b)), click the Voltage/Current/Power Marker button on the toolbar of the Schematic Editor window and put the probe pin at the target node or element. Alternatively, you can just press the 'Ins' key or click Trace/Add_Trace on the menu bar or the Add Trace

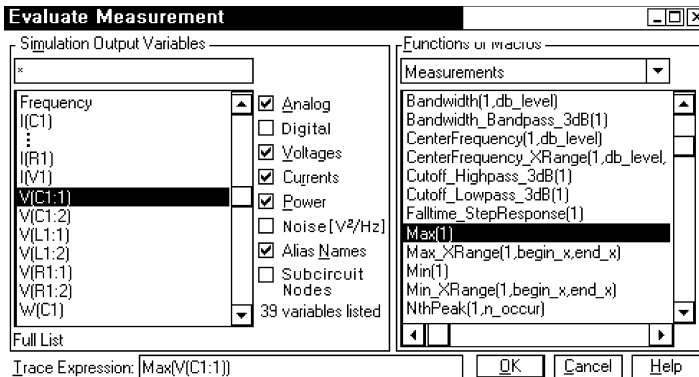
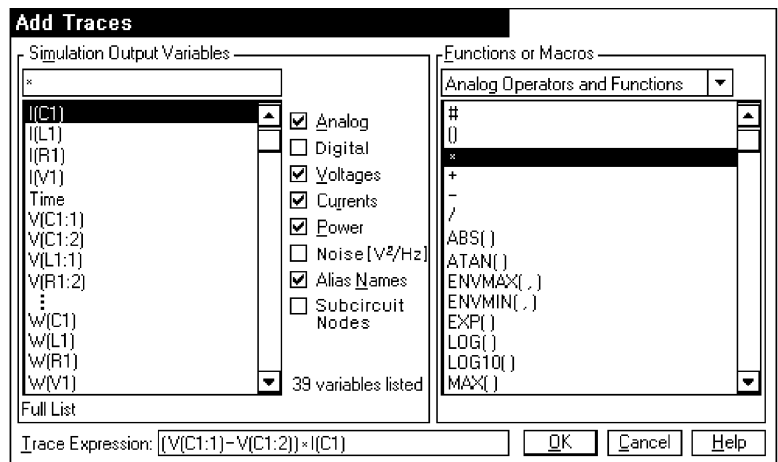
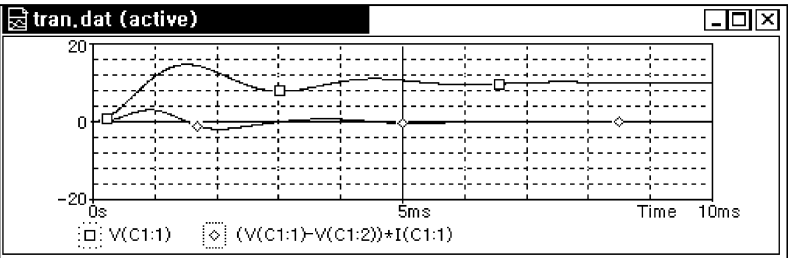


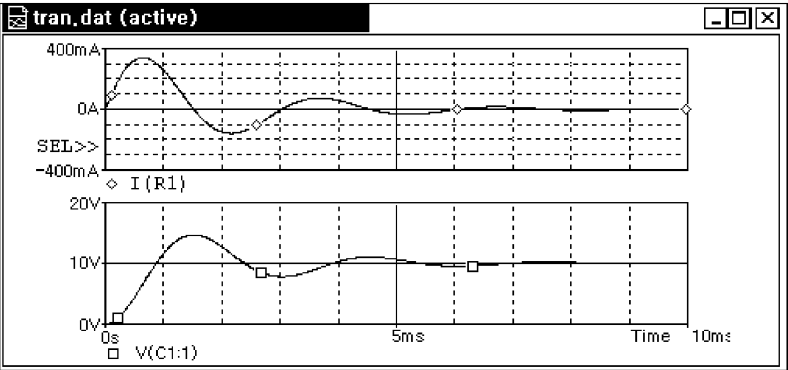
Figure H.7 The Evaluation Measurement dialog box



(a) An example of trace expression entered in the Add Traces dialog box



(b) The waveforms seen in the PSpice A/D (Probe) window



(c) The graph added by using Plot/Add Plot and Trace/Add Trace

Figure H.8 Adding waveform(s) by Add Trace and Add Plot

button on the toolbar (of the Probe window) to open the Add Traces dialog box (Figure H.8(a)) and enter the target trace expression into the Trace Expression field (at the bottom) directly or by selecting related variable(s) from the Simulation Output Variable list (on the left side) and/or the operator/function from the Function or Macro list (on the right side). Any existing waveform can be deleted by clicking the variable name below the waveform graph and pressing the 'Del' key.

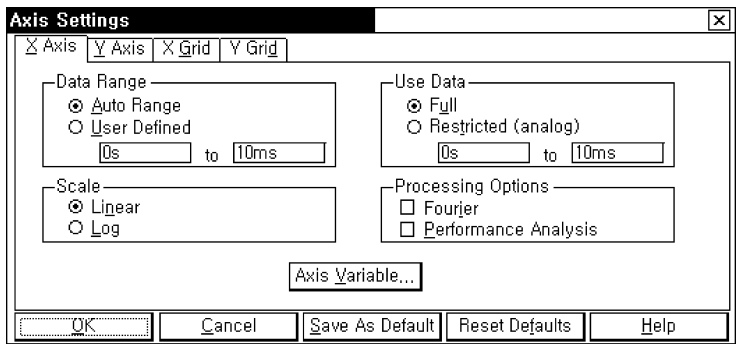
In case the plotted variables differ in the order of magnitude and/or the unit so that you feel like having another graph on which to plot one or more of them, you can click Plot/Add_Plot_to_Window on the menu bar (of the Probe window) to add another graph, as illustrated in Figure H.8(c), press the ‘Ins’ key to open the Add Traces dialog box, and then enter the trace expression to plot in the new graph. Note that you might have to increase the vertical size of the Probe window to accommodate additional graph(s) to be created. After getting more than one graph, you can select one of them by clicking it so that you can modify it, where the selected graph is denoted by ‘SEL>>’. If you want to delete a graph, click the graph for selection and then click Plot/Delete_Plot on the menu bar (of the Probe window) for deletion.

If there is any data file on the same named-variable for the same range in another simulation, you can plot it in the same graph by clicking File/Append_Waveform(.DAT) on the menu bar or the Append File button and then selecting the data file.

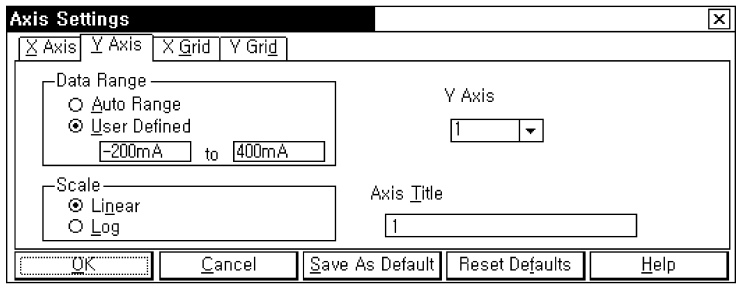
H.4.3 Processing Waveforms/Graph

You can scale up/down the graph by either pressing the ‘^|’‘^o’ key or clicking View/Zoom/(In/Out) on the menu bar or the Zoom_(In/Out) button on the toolbar, and then clicking at any point around which the graph will be zoomed in/out by a factor of 2. You can scale up/down a partial area of the graph by either pressing the ‘^a’ key or clicking View/Zoom/Area on the menu bar or the Zoom_Area button on the toolbar, and then dragging the mouse pointer from a corner point to the opposite one to get the area selected. You can click View/Zoom/Fit on the menu bar or the Zoom_Fit button on the toolbar to have the whole waveform(s) fit the graph.

You can also click the Plot/Axis_Settings menu or double-click the *x* or *y*-axis to open the Axis Settings dialog box (Figure H.9) and select a tab of {X-Axis,Y-Axis,X-Grid,Y-Grid} to make a change of



(a) Axis setting for *x*-axis



(b) Axis setting for *y*-axis

Figure H.9 The Axis Setting dialog box

linear/log scales, ranges, and grid spacings (intervals) of the x -axis and y -axis. Especially, you can draw an x - y plot instead of a t - y plot by clicking the Axis Variable button (Figure H.9(a)) and then changing the x axis variable from Time to another variable in the Axis Settings dialog box for the x -axis.

You can label the y -axis by entering the title string into the Axis Title field (Figure H.9(b)) in the Axis Settings dialog box for the y -axis. You can also write a text string (name) and draw a line/arrow/box/circle on the graph by clicking Plot/Label/(Text|Line|Arrow|Box|Circle). You can see the Fourier spectra of signals plotted versus frequency in the Probe window just by clicking Trace/Fourier on the menu bar or the Fourier button on the toolbar.

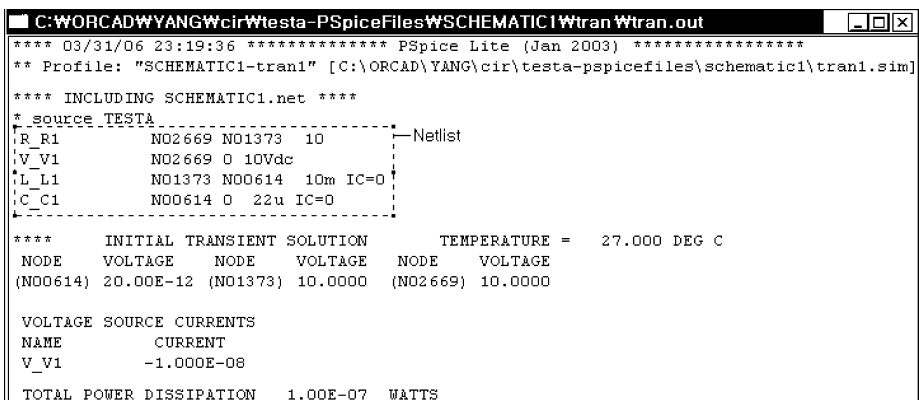
H.4.4 Initial Transient Bias Point

There are two different kinds of bias point: one is the initial conditions for transient analysis and the other is a DC analysis result for small-signal analysis. The former, referred to as the *initial transient bias point*, is found with every inductor/capacitor open/short and every AC source removed. In order to find the initial transient bias point, set the initial current/voltage of every inductor/capacitor to its initial value, even if it is given as zero. The latter, referred to as the *operating point*, is found with every inductor/capacitor short/open and every AC source removed. In order to find the operating point, you should not set the initial current/voltage of every inductor/capacitor.

To have the bias point analysis results displayed/removed on the schematic, click PSpice/Bias_Points/Enable_Bias_(Voltage|Current|Power)_Display on the menu bar or the corresponding button on the toolbar of the Capture window. To have the bias point analysis results printed in the output file, you should check the square box before 'Include detailed bias point information for nonlinear controlled sources and semiconductors (/OP)' in the Simulation Settings dialog box for bias point analysis or in the Transient Output File Options dialog box opened by clicking the Output File Options button in the Simulation Settings dialog box for transient analysis.

H.4.5 Viewing Output File and Netlist File

To see the simulation output file (Figure H.10), click PSpice/View_Output_File on the menu bar of the Capture window or the View Simulation Output File button on the left side of the Probe window. The output file contains the contents of the netlist file named 'schematic1.net', which shows the detailed description of the circuit. The netlist file can also be seen by clicking PSpice/View_Netlist_File on the menu bar of the Capture window.



```

C:\WORCAD\WYANG\cir\Wtesta-PSpiceFiles\WSchematic1\Wtran\Wtran.out
**** 03/31/06 23:19:36 ***** PSpice Lite (Jan 2003) *****
** Profile: "SCHEMATIC1-tran1" [C:\ORCAD\YANG\cir\testa-pspicefiles\schematic1\tran1.sim]

**** INCLUDING SCHEMATIC1.net ****
* _source_TESTA
+-----+-----+-----+-----+
R_R1      NO2669 NO1373  10          Netlist
V_V1      NO2669  0 10Vdc
L_L1      NO1373 NO0614  10m IC=0
C_C1      NO0614  0  22u IC=0
+-----+-----+-----+-----+

****      INITIAL TRANSIENT SOLUTION      TEMPERATURE =  27.000 DEG C
NODE      VOLTAGE      NODE      VOLTAGE      NODE      VOLTAGE
(N00614)  20.00E-12 (N01373)  10.0000 (N02669)  10.0000

VOLTAGE SOURCE CURRENTS
NAME      CURRENT
V_V1      -1.000E-08

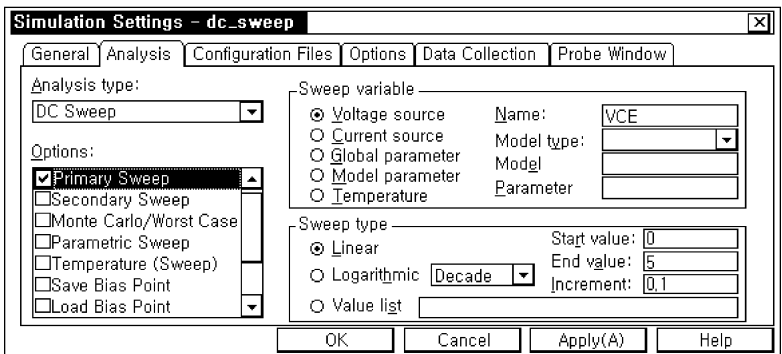
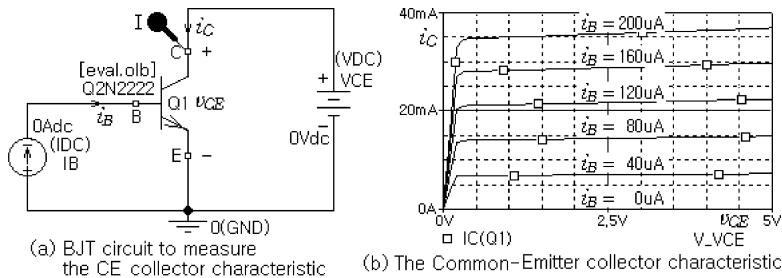
TOTAL POWER DISSIPATION  1.00E-07  WATTS
  
```

Figure H.10 An example of the PSpice output file

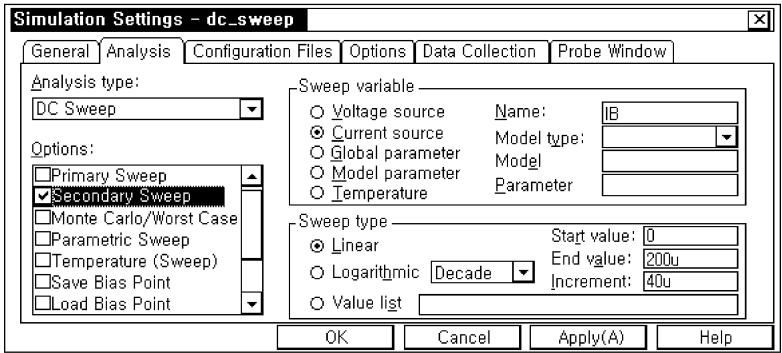
H.5 Circuit Analysis Using OrCAD/Capture

H.5.1 DC Sweep Analysis

To practice the DC sweep analysis, the CE (common-emitter) collector characteristic curves of a BJT (bipolar junction transistor) are found by simulating the circuit in Figure H.11(a). For this job, the



(c) The Simulation Settings dialog box for the primary sweep variable



(d) The Simulation Settings dialog box for the secondary sweep variable

Figure H.11 Obtaining the CE collector characteristic curves ('BJT_CEO.opj')

schematic should be drawn and the parameters set in the Simulation Settings dialog box as depicted in Figure H.11(c) and (d). Then the CE collector characteristic curves will be obtained as depicted in Figure H.11(b) (see References [R-1] and [S-1] for details about BJT).

As another application example of DC sweep analysis, the schematic of a CE transistor circuit can be drawn as in Figure H.12(a) and run to get the curves of the B–E junction voltage v_{BE} , the base current i_B , the collector current i_C , the collector–emitter junction voltage v_{CE} , and the DC current gain h_{FE} versus V_{BB} , as depicted in Figures H.12(b1) to (b5). Instead of Voltage/Current Markers, the Add Trace button

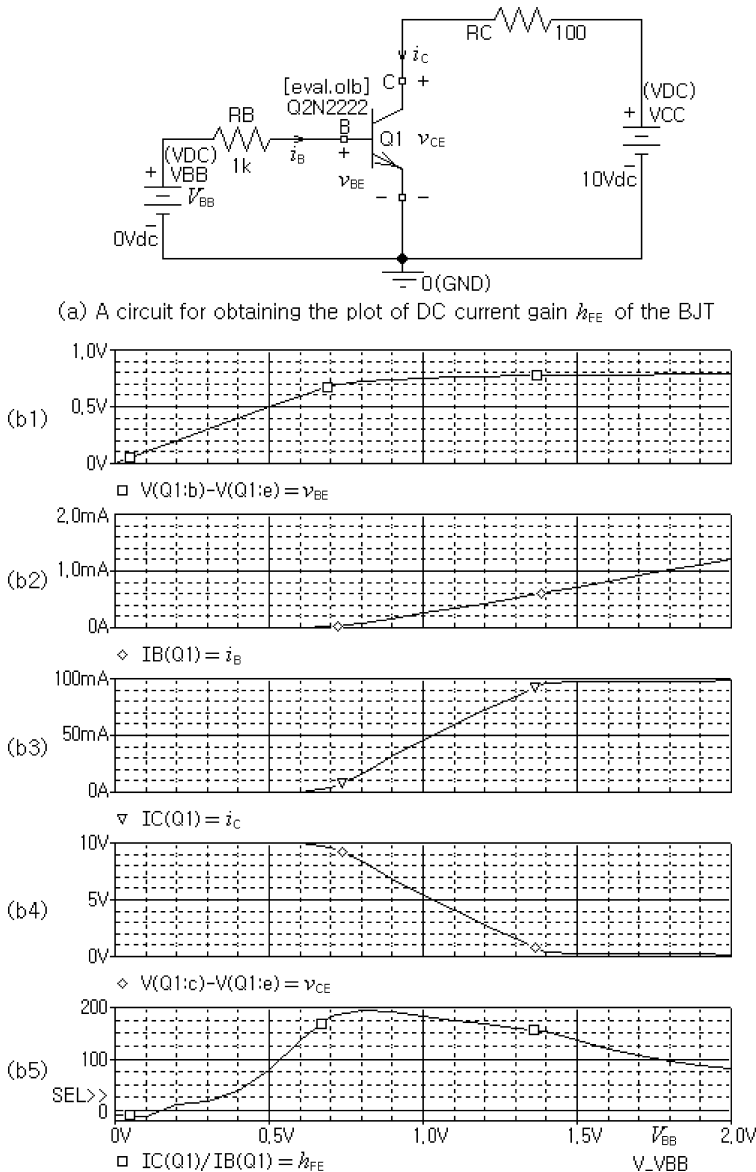
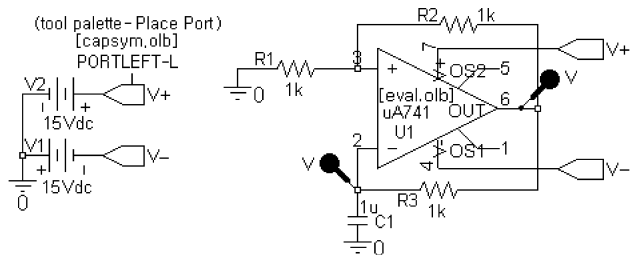


Figure H.12 Plot of v_{BE} , i_B , i_C , v_{CE} and h_{FE} versus, V_{BB} ('BJT_hFE.opj')

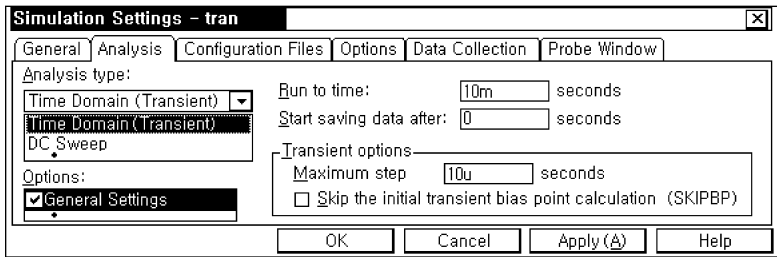
was clicked on the tool bar of the Probe window to open the Add Traces dialog box and each target expression like 'V(Q1:b)-V(Q1:e)' or 'IC(Q1)/IB(Q1)' (with no quotation mark) was typed into the Trace Expression field to obtain these plots in the Probe window.

H.5.2 Transient Analysis

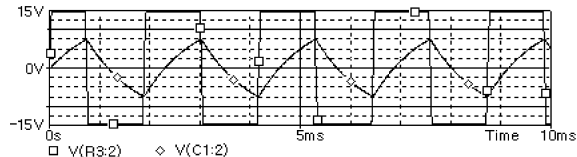
Figures H.13(a), (b), and (c) show the schematic of an OP Amp circuit, the Simulation Settings dialog box for the time-domain (transient) analysis, and the output waveforms obtained from the simulation, respectively. The period of the output waveform may be read either by using the cursor or by clicking the Evaluate Measurement button on the toolbar in the PSpice A/D (Probe) window to open the Evaluate Measurement window and filling out the Trace Expression field as depicted in Figure H.13(d) to get the measurement result (Figure H.13(e)).



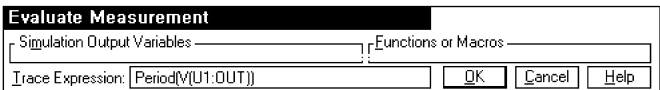
(a) The PSpice schematic for a square-wave generator using an OP Amp



(b) The Simulation Settings dialog box for time-domain (transient) analysis



(c) The Probe output waveforms in the PSpice A/D (Probe) window



(d) The Evaluate Measurement dialog box

Measurement Results		
Evaluate	Measurement	Value
☒	Period(V(U1:OUT))	2.25051m
Click here to evaluate a new measurement...		

(e) The Measurement Results box

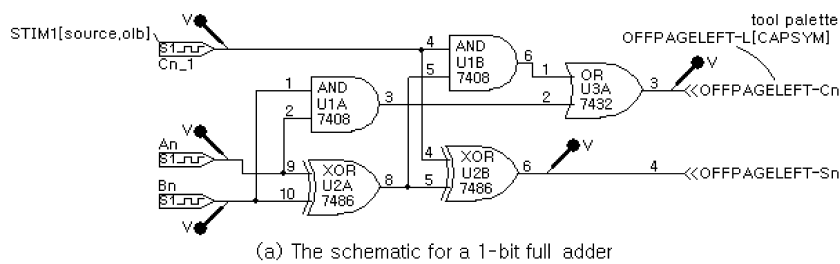
Figure H.13 PSpice simulation of an OP amplifiers circuit ('OP_Amp_pos_feedback, opj')

Figure H.14(a) shows a logic circuit for a one-bit full adder, which gets three inputs A_n, B_n , and C_{n-1} (carry from bit $n-1$) and computes two outputs S_n (sum) and C_n (carry) as

$$S_n = (\overline{A_n} \overline{B_n} + A_n B_n) C_{n-1} + (\overline{A_n} B_n + A_n \overline{B_n}) \overline{C_{n-1}} = \overline{XOR(A_n, B_n)} C_{n-1} + XOR(A_n, B_n) \overline{C_{n-1}}$$
$$= XOR[XOR(A_n, B_n), C_{n-1}] \tag{H.1}$$

$$C_n = A_n B_n (C_{n-1} + \overline{C_{n-1}}) + (\overline{A_n} B_n + A_n \overline{B_n}) C_{n-1} = A_n B_n + XOR(A_n, B_n) C_{n-1} \tag{H.2}$$

Figures H.14(b1) to (b3) and (c) show the Property Editor spreadsheets for each of the three inputs and the timing chart obtained from the transient analysis and seen in the PSpice A/D (Probe) window, respectively.



(a) The schematic for a 1-bit full adder

Property Editor									
New Column...	Apply	Display...	Delete Property	Filter	< Current properties >				Help
			COMMAND1	COMMAND2	COMMAND3	COMMAND4	COMMAND5	COMMAND6	COMMAND7
1	SCHEMATIC1:PA6E1-Cn-1	0s 0	0.4us 1						STIM1

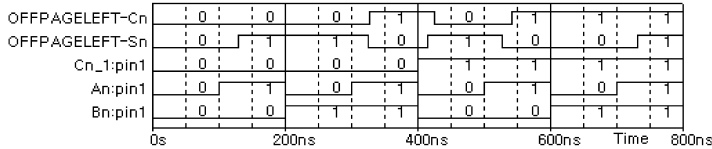
(b1) The Property Editor spreadsheet for digital input source STIM1 – Cn_1

Property Editor									
New Column...	Apply	Display...	Delete Property	Filter	< Current properties >				Help
			COMMAND1	COMMAND2	COMMAND3	COMMAND4	COMMAND5	COMMAND6	COMMAND7
1	SCHEMATIC1:PA6E1-Cn-1	0s 0	0.1us 1	0.2us 0	0.3us 1	0.4us 0	0.5us 1	0.6us 0	0.7us 1

(b2) The Property Editor spreadsheet for digital input source STIM1 – An

Property Editor									
New Column...	Apply	Display...	Delete Property	Filter	< Current properties >				Help
			COMMAND1	COMMAND2	COMMAND3	COMMAND4	COMMAND5	COMMAND6	COMMAND7
1	SCHEMATIC1:PA6E1-Cn-1	0s 0	0.2us 1	0.4us 0	0.6us 1				STIM1

(b3) The Property Editor spreadsheet for digital input source STIM1 – Bn



(c) The timing chart obtained from PSpice simulation (Transient)

Figure H.14 PSpice simulation for a one-bit full adder ('full_adder.opj')

H.5.3 AC Sweep Analysis

Figures H.15(a), (b), and (c) show the schematic of an OP Amp circuit, the Simulation Settings dialog box for the frequency-domain (AC sweep) analysis, and the frequency response curve obtained from the simulation. The peak or center frequency f_p and the lower/upper 3 dB frequencies f_l and f_u of the BPF (bandpass filter) may be read by using the cursor or the Evaluate Measurement function as follows:

1. To find the peak or center frequency f_p at which the magnitude of the frequency response achieves its maximum, click the Toggle Cursor button to activate the cursor and then click the Cursor Max button on the toolbar of the Probe window, which will show you the cursor at the maximum together with the coordinates $(f_p, |G(j2\pi f_p)|) = (254.097, 892.716 \text{ m})$ in the Probe Cursor box (Figure H.15(c)).
2. To find the two 3 dB frequencies f_l and f_u at which the magnitude of the frequency response is $(1/\sqrt{2})$ times its maximum, i.e. $0.893/\sqrt{2} = 0.631$, draw a horizontal line of $y = 0.631$ by clicking the Add Trace button to open the Add Traces dialog box and typing '0.631' (with no quotation marks) into the Trace Expression field. Then position the cursor at the two intersection points of the frequency response curve and the horizontal line of $y = 0.631$ by left/right-clicking at the target points or pressing the (left or right) arrow/Shift+arrow key and read the coordinates of the two points from the Probe Cursor box, which will be

$$f_l \simeq 185.6 \text{ Hz} \quad \text{and} \quad f_u \simeq 344.9 \text{ Hz}$$

respectively (see Figure H.15(c)). It is also implied that the bandwidth is $B = f_u - f_l \simeq 159.3 \text{ Hz}$.

3. Once the frequency response in the Probe window has been found, it is easy to find the center frequency, the 3 dB frequencies, and the bandwidth by using the Evaluate Measurement function. That is, you can click the Evaluate Measurement button to open the Evaluate Measurement dialog box and do the following (see Figure H.15(d)):
– Fill out the Trace Expression field with

Cutoff_Highpass_3dB(V(U1:OUT))

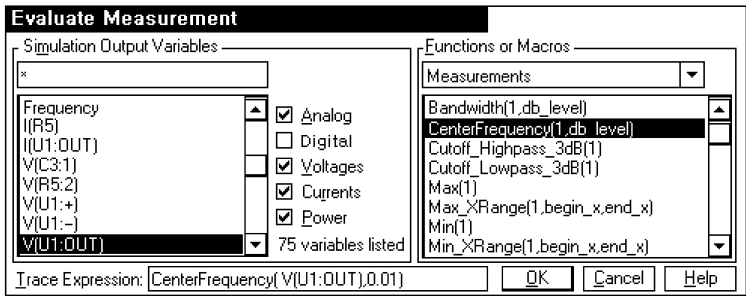
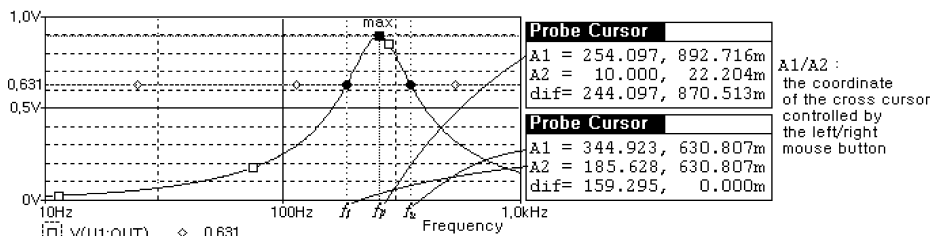
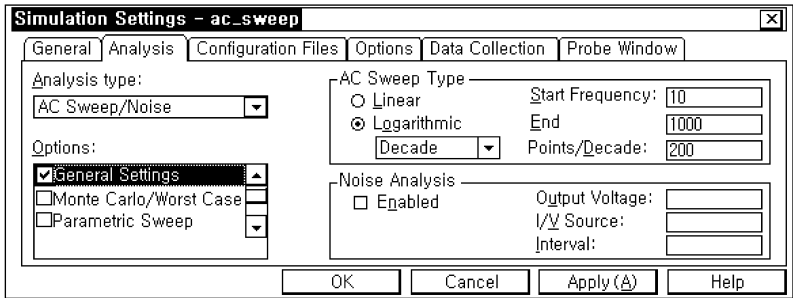
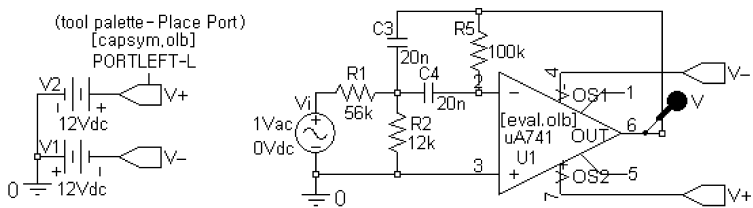
and click OK. Then you will see the value of the lower 3 dB frequency in the Measurement Results box appearing (below the graph) at the bottom of the Probe window. Consecutively, click the bottom part of the Measurement Results box saying that

Click here to evaluate a new measurement. . .

which will reopen the Evaluate Measurement dialog box. Then, in the same way, you can type

Center_Frequency(V(U1:OUT),0.01)
Bandwidth_Bandpass_3dB(V(U1:OUT))
Cutoff_Lowpass_3dB(V(U1:OUT))

one at a time, which will make the center frequency, the bandwidth, and the upper 3 dB frequency appear in the Measurement dialog box as illustrated in Figure H.15(e).



Measurement Results		
Evaluate	Measurement	Value
☒	Cutoff_Lowpass_3dB(V(U1:OUT))	344.53009
☒	Bandwidth_Bandpass_3dB(V(U1:OUT))	158.68715
☒	CenterFrequency(V(U1:OUT),0.01)	253.19116
☒	Cutoff_Highpass_3dB(V(U1:OUT))	185.84294

Click here to evaluate a new measurement...

(e) The Measurement Results box

Figure H.15 PSpice simulation to get a frequency response ('OP_Amp_filter_ac_sweep.opj')

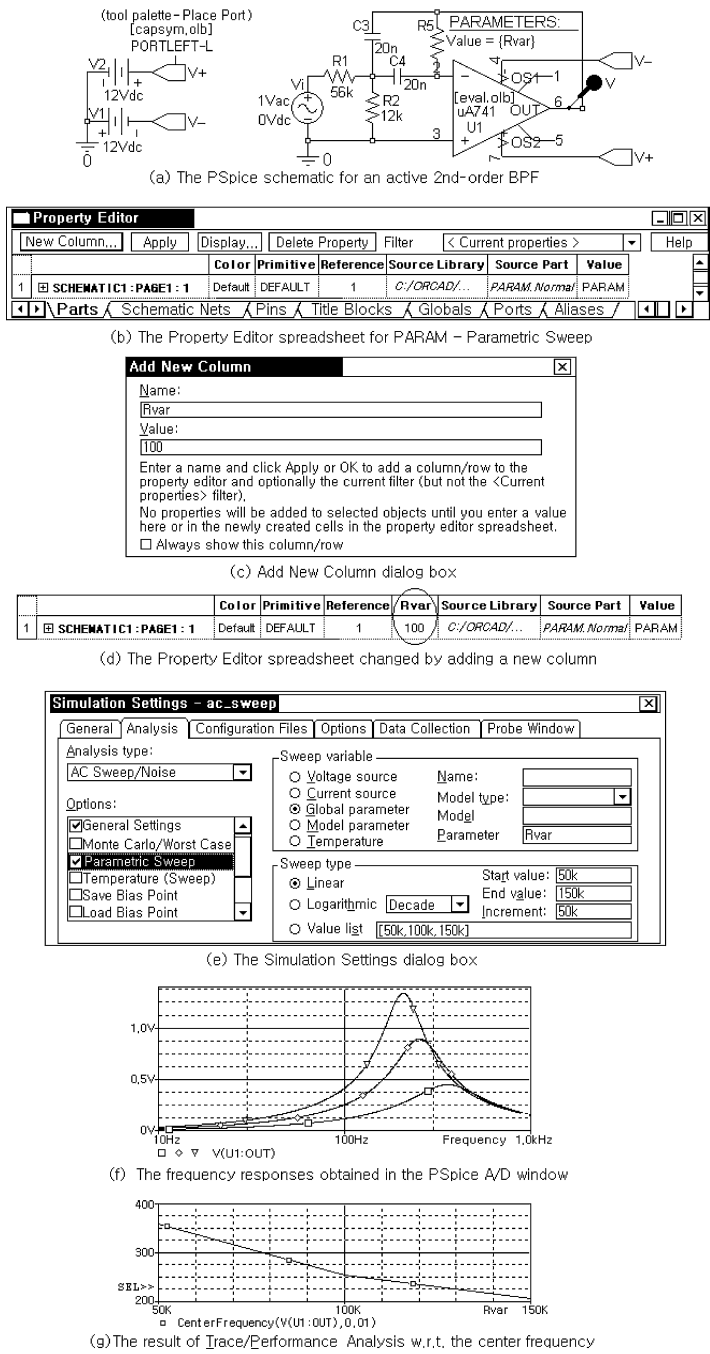


Figure H.16 Multiple AC Sweeps using Parametric Sweep ('OP_Amp_filter_par.opj')

H.5.4 Parametric Sweep Analysis

While the DC Sweep (Section H.5.1) and AC Sweep (Section H.5.3) analyses aim to observe the variation of the output versus the value of a DC source or the frequency of an AC source, the Parametric Sweep analysis aims to observe the variation of the output versus the value of R , L , C , a model parameter, temperature, etc. Let us observe how the frequency response of the BPF in Figure H.16(a) varies with the value of $R_5 = 50, 100, \text{ and } 150 \text{ k}\Omega$ by taking the following steps:

1. Click the value of R_5 (to be varied) and set it to $\{Rvar\}$ (including the braces) in the schematic (Figure H.16(a)).
2. Click the Place Part button on the tool palette of the Schematic window to open the Place Part dialog box, get PARAM (contained in the library 'special.olb'), and click OK to place it somewhere in the schematic (Figure H.16(a)).
3. Double-click the PARAMETERS placed in the schematic to open the Property Editor spreadsheet (Figure H.16(b)) and click the New Column button to open the Add New Column box (Figure H.16(c)), in which you can type $Rvar$ and 100 into the Name and Value fields, respectively, where the numerical value entered as 100 here does not matter. You can see the added column named ' $Rvar$ ' in the Property Editor spreadsheet as in Figure H.16(d).
4. Click the New Simulation Profile button to create a new simulation profile named, say, ' AC_sweep ', click the Edit Simulation Profile button to open the Simulation Settings dialog box, and fill it out as depicted in Figure H.16(e), where the menu of Options is selected as Parametric Sweep, the Sweep variable is chosen to be the Global parameter named ' $Rvar$ ', and the Sweep type is set to Linear with the Start value 50k, the End value 150k, and the Increment 50k. If the values of the parameters do not form an arithmetic progression (with constant difference), you may type the list of the parameter values into the Value list field as illustrated in Figure H.16(e). Then click OK to close the Simulation Settings dialog box.
5. Click the Voltage/Level Marker button to put the voltage probe pin at the OP Amp output node, click the Run button to perform the simulation, and see the multiple frequency curves as in Figure H.16(f).
6. To see how the center frequency f_p varies with the value of $R_5 = \{Rvar\}$, click Trace/Performance Analysis on the menu bar of the PSpice A/D (Probe) window to create an additional graph, click the Add Trace button to open the Add Traces dialog box, and fill out the Trace Expression field with

CenterFrequency(V(U1:OUT),0.01)

Then click OK to see the plot of f_p versus R_5 as depicted in Figure H.16(g).

H.5.5 Transient/Fourier Analysis

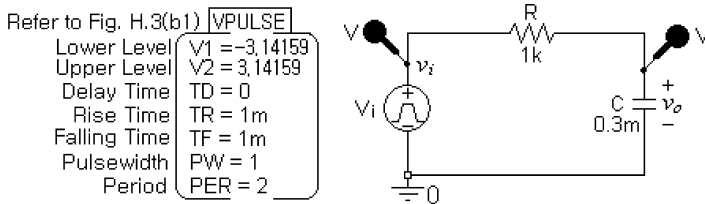
Figure H.17(a) shows the PSpice schematic of an RC circuit excited by a rectangular (square) wave voltage source of height $\pm V_m = \pm \pi$, period $T = 2 \text{ s}$, and duration (pulsewidth) $D = 1 \text{ s}$, where the voltage across the capacitor is taken as the output. To have the Fourier analysis results printed in the output file, the simulation profile is set and the Output File Options button is clicked to open the Transient Output File Options dialog box, in which you are supposed to enter the fundamental frequency $f_0 = 1/T = 1/2 = 0.5 \text{ Hz}$, the number of harmonics that you want to observe, and the list of output variable(s) for Fourier analysis into the corresponding fields, say, as illustrated in Figure H.17(b). Now, the Voltage/Level probe pins are put at the input/output nodes, the Run

button is clicked to perform the simulation, and the input/output waveforms are seen in the Probe window as in Figure H.17(c). Whenever you want to see the spectra of the signals (Figure H.17(d)) whose waveforms are shown in the Probe window, just click the (toggle) FFT button on the toolbar. You can also view the output file as shown in Figure H.17(e) by clicking (selecting) PSpice/View_ Output_File on the top menu bar and pulldown menu in the Capture window or View/Output_File on the menu bar or the View_Simulation_Output_File button on the left toolbar of the Probe window.

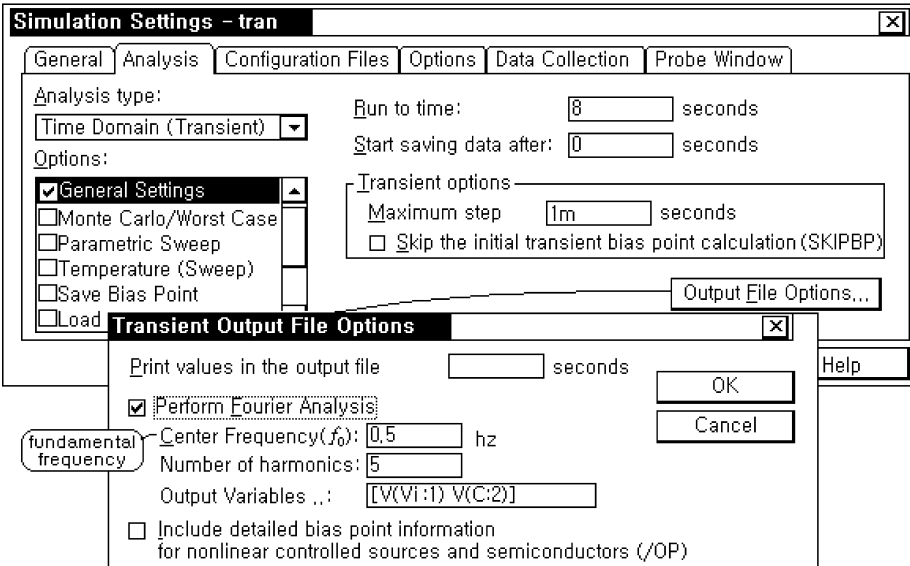
Note that based on the Fourier analysis results printed in the output file, the Fourier series representation of the input and output signals can be written as

$$v_i(t) = \sum_{k=0}^{\infty} d_k \sin(2\pi k f_0 t + \phi_k) \quad (f_0 = 0.5 \text{ Hz})$$

$$\cong -0.031 + 4 \sin(\pi t - 0.89^\circ) + 0.062 \sin(2\pi t - 91.78^\circ) + 1.334 \sin(3\pi t - 2.67^\circ) + \dots \quad (\text{H.3})$$

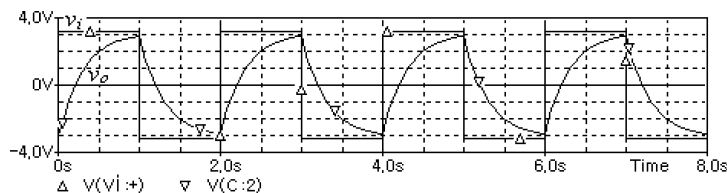


(a) An RC circuit driven by a square wave voltage source (VPULSE)

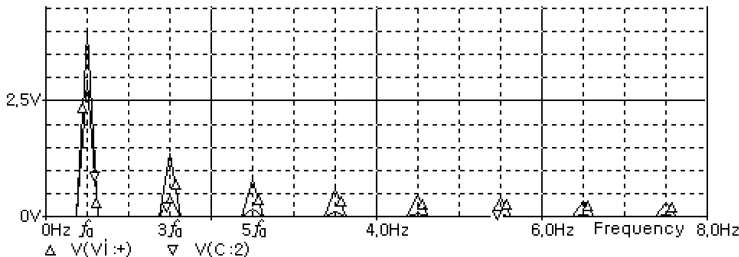


(b) The Simulation Settings window for transient response with Fourier analysis

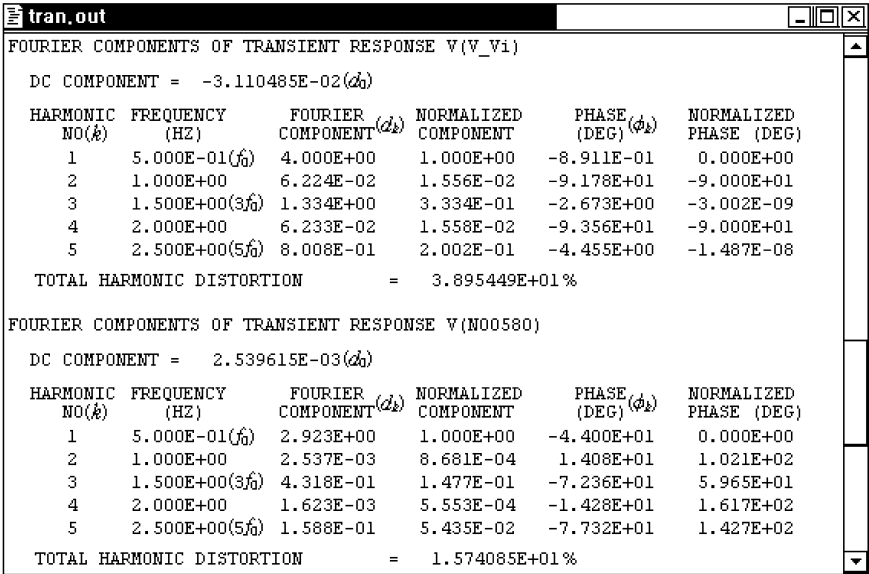
Figure H.17 Example of Fourier analysis using PSpice ('RC_pulse_Fourier.opj')



(c) The voltage generated by the square wave voltage source V1 and the voltage across the capacitor C1



(d) The Fourier spectra of V1 and V(C) obtained by clicking the FFT icon in the tool bar



(e) The Fourier analysis results in the output file

Figure H.17 (Continued)

$$v_o(t) \cong 2.92 \sin(\pi t - 44^\circ) - 0.43 \sin(3\pi t - 72^\circ) + 0.16 \sin(5\pi t - 77^\circ) + \dots \quad (\text{H.4})$$

and the total harmonic distortion (THD) is defined as

$$\text{THD} = \frac{\sqrt{\sum_{k=2}^{\infty} d_k^2}}{d_1} \quad (\text{H.5})$$

H.5.6 Bias Point/DC Sensitivity and Transfer Function Analysis

Figures H.18(a1) and (a2) show two interface circuits, each of which takes on a duty of transferring the DC voltage of 9 V from the 12 V source (with the source impedance of 1.5 k Ω) to a 9 k Ω resistor. Figure H.18(b) shows the Simulation Settings dialog box to be set for DC Sensitivity analysis of the output voltage w.r.t. the load resistance and Transfer Function analysis to find the DC gain and the input/output resistances, where 'V(RL2)' should be typed into the Output and To_Output variable fields instead of 'V(RL1)' for the analysis of circuit (a2). Figures H.18(c1)/(c2) and (d1)/(d2) show the results of the DC Sensitivity and Transfer Function analyses for the two circuits (a1)/(a2), which can be seen from the output file.

In order to understand and verify these analysis results, find the sensitivities, DC gains, and the input/output impedances by using an analytical method as follows.

(a) For circuit (a):

$$v_o(R_{L1}) = \frac{R_{L1}}{R_{s1} + R_S + R_{L1}} V_{s1} = \frac{R_{L1}}{3 + R_{L1}} 12$$

$$\text{Sensitivity of } v_o(R_{L1}) \text{ w.r.t. } R_{L1} \Big|_{R_{L1}=9\text{k}\Omega} = \frac{dv(R_{L1})}{dR_{L1}} \Big|_{R_{L1}=9\text{k}\Omega} = \frac{3V_{s1}}{(3 + R_{L1})^2} \Big|_{R_{L1}=9\text{k}\Omega} = \frac{1}{4\text{k} = 4000} = 2.5\text{E} - 4$$

$$\text{DC gain} = \frac{dv(R_{L1})}{V_{s1}} \Big|_{R_{L1}=9\text{k}\Omega} = \frac{R_{L1}}{3 + R_{L1}} \Big|_{R_{L1}=9\text{k}\Omega} = \frac{3}{4} = 7.5\text{E} - 1$$

$$\text{Input resistance} = R_{s1} + R_S + R_{L1} = 1.5 + 1.5 + 9 = 12\text{k}\Omega = 1.2\text{E} + 4$$

$$\text{Output resistance} = (R_{s1} + R_S) \parallel R_{L1} = (1.5 + 1.5) \parallel 9 = \frac{3 \times 9}{3 + 9} = 2.25\text{k}\Omega = 2.25\text{E} + 3$$

(b) For circuit (b):

$$v_o(R_{L2}) = \frac{R_P \parallel R_{L2}}{R_{s2} + (R_P \parallel R_{L2})} V_s = \frac{9 \times R_{L2} / (9 + R_{L2})}{1.5 + 9 \times R_{L2} / (9 + R_{L2})} 12 = \frac{9R_{L2}}{13.5 + 10.5R_{L2}} 12$$

$$\text{Sensitivity of } v_o(R_{L2}) \text{ w.r.t. } R_{L2} \Big|_{R_{L2}=9\text{k}\Omega} = \frac{dv(R_{L2})}{dR_{L2}} \Big|_{R_{L2}=9\text{k}\Omega} = \frac{9 \times 13.5V_{s1}}{(13.5 + 10.5R_{L2})^2} \Big|_{R_{L2}=9\text{k}\Omega} = 1.25\text{E} - 4$$

$$\text{DC gain} = \frac{dv(R_{L2})}{V_{s1}} \Big|_{R_{L2}=9\text{k}\Omega} = \frac{9R_{L2}}{13.5 + 10.5R_{L2}} \Big|_{R_{L2}=9\text{k}\Omega} = \frac{3}{4} = 7.5\text{E} - 1$$

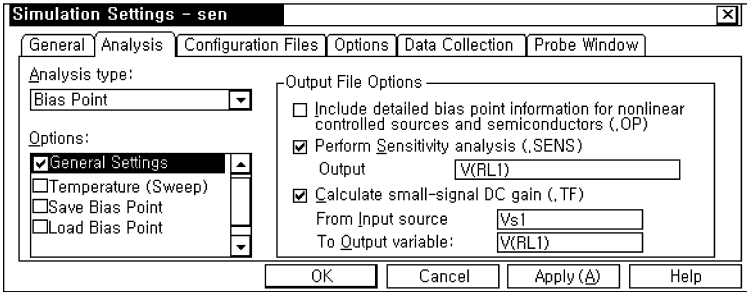
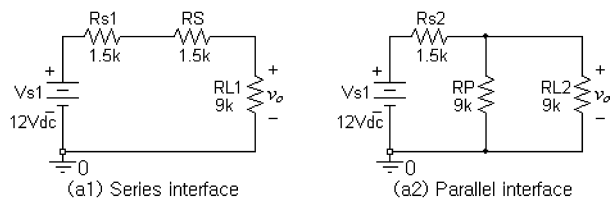
$$\text{Input resistance} = R_{s2} + (R_P \parallel R_{L2}) = 1.5 + \frac{9 \times 9}{9 + 9} = 6\text{k}\Omega = 6\text{E} + 3$$

$$\text{Output resistance} = R_{s2} \parallel R_P \parallel R_{L2} = \frac{1}{1/1.5 + 1/9 + 1/9} = 1.125\text{k}\Omega = 1.125\text{E} + 3$$

H.5.7 Monte Carlo Analysis

Figure H.19(a) shows the Capture window, which has the schematic of a CE (common-emitter) BJT (bipolar junction transistor) amplifier circuit. To see how the output voltage varies with the $\pm 20\%$ change of the load resistance R_L (RL) and the $\pm 50\%$ change of the CE forward current gain β_F , let us take the following steps of Monte Carlo/Worst Case analysis presented by PSpice:

1. Create a PSpice project, say, named 'BJT_CE_Amp1_mon' and draw a schematic in the Schematic Editor window as depicted in Figure H.19(a).



(b) The Simulation Settings dialog box for DC Sensitivity and Transfer Function analyses

DC SENSITIVITIES OF OUTPUT V(R_RL1)			
ELEMENT NAME	ELEMENT VALUE	ELEMENT SENSITIVITY (VOLTS/UNIT)	NORMALIZED SENSITIVITY (VOLTS/PERCENT)
R_Rs1	1.500E+03	7.500E-04	1.125E-02
R_RS	1.500E+03	7.500E-04	1.125E-02
R_RL1	9.000E+03	2.500E-04	-2.250E-02

(c1) The result of DC Sensitivity analysis for the circuit in (a1)

DC SENSITIVITIES OF OUTPUT V(R_RL2)			
ELEMENT NAME	ELEMENT VALUE	ELEMENT SENSITIVITY (VOLTS/UNIT)	NORMALIZED SENSITIVITY (VOLTS/PERCENT)
R_Rs2	1.500E+03	1.500E-03	2.250E-02
R_RP	9.000E+03	-1.250E-04	-1.125E-02
R_RL2	9.000E+03	-1.250E-04	-1.125E-02

(c2) The result of DC Sensitivity analysis for the circuit in (a2)

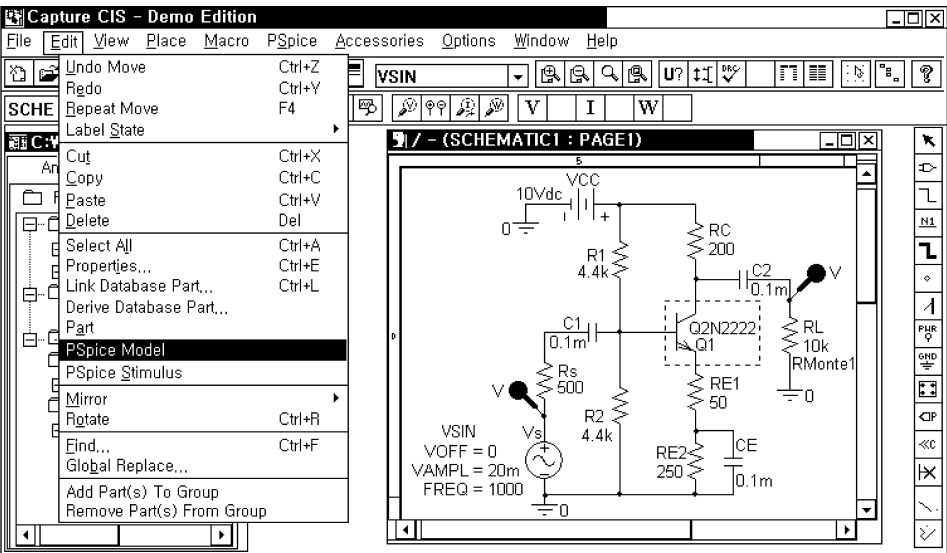
```
****      SMALL-SIGNAL CHARACTERISTICS
V(R_RL2)/V_Vs1 = 7.500E-01
INPUT RESISTANCE AT V_Vs1 = 1.200E+04
OUTPUT RESISTANCE AT V(R_RL1) = 2.250E+03
```

(d1) The result of Transfer Function analysis for the circuit in (a1)

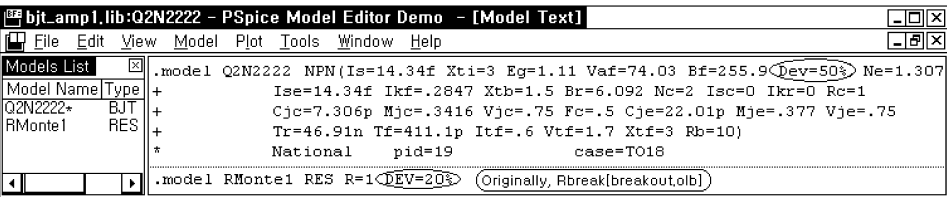
```
****      SMALL-SIGNAL CHARACTERISTICS
V(R_RL2)/V_Vs1 = 7.500E-01
INPUT RESISTANCE AT V_Vs1 = 6.000E+03
OUTPUT RESISTANCE AT V(R_RL2) = 1.125E+03
```

(d2) The result of Transfer Function analysis for the circuit in (a2)

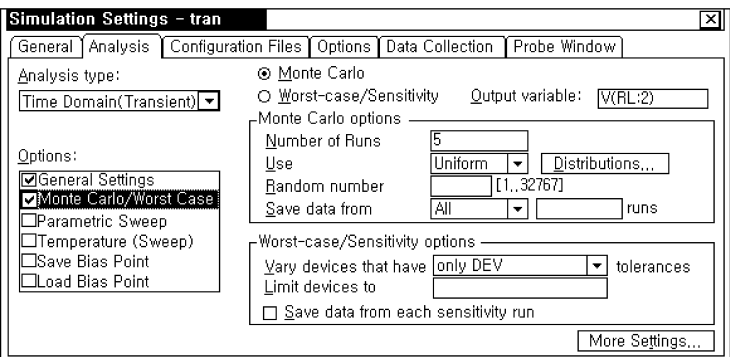
Figure H.18 DC Sensitivity and Transfer Function analysis ('test_sensitivity.opj')



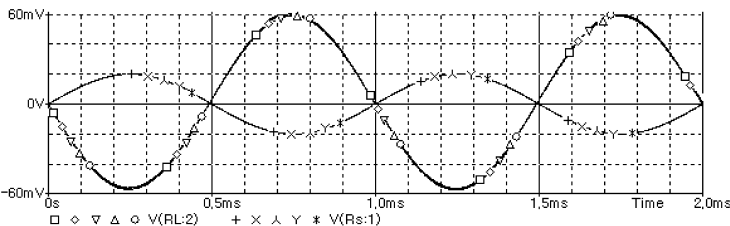
(a) The schematic of a BJT(Bipolar Junction Transistor) amplifier in the Capture Window



(b) The Model Editor window for setting the deviation (Dev) of forward current gain β_F of a BJT



(c) The Simulation Settings dialog box for setting the analysis type to Transient/Monte Carlo



(d) The Monte Carlo simulation result for 50% deviation of β_F of the BJT

Figure H.19 Example of Monte Carlo simulation ('BJT_CE_amp1_mon.opj')

```
** Profile: "SCHEMATIC1-tran1" [ C:\OrCAD\Yang\bjt_ampl-pspicefiles\schematic1\tran1.sim]
**** SORTED DEVIATIONS OF V(M03459) TEMPERATURE = 27.000 DEG C

MONTE CARLO SUMMARY
*****
Mean Deviation = -196.4000E-06, Sigma = 766.4500E-06

RUN MAX DEVIATION FROM NOMINAL
Pass 4 1.3512E-03 (1.76 sigma) lower ( 97.735% of Nominal) at T = 729.3000E-06
Pass 5 669.2600E-06 (.87 sigma) higher (101.14% of Nominal) at T = 1.7183E-03
Pass 2 393.0600E-06 (.51 sigma) lower ( 99.218% of Nominal) at T = 838.3000E-06
Pass 3 281.3900E-06 (.37 sigma) higher (100.62% of Nominal) at T = 630.2900E-06
```

(e) The Monte Carlo simulation result printed in the output file

Simulation Settings - tran

General Analysis Configuration Files Options Data Collection Probe Window

Analysis type: Time Domain(Transient)

Options:

- ☒ General Settings
- ☒ Monte Carlo/Worst Case
- ☐ Parametric Sweep
- ☐ Temperature (Sweep)
- ☐ Save Bias Point
- ☐ Load Bias Point

Monte Carlo options

Number of Runs: 5

Use: Uniform Distributions...

Random number: [1..32767]

Save data from: All runs

Worst-case/Sensitivity options

Vary devices that have: only DEV tolerances

Limit devices to:

☐ Save data from each sensitivity run

More Settings...

Monte Carlo/Worst-Case Output File Options

Collating Function

The collating function is performed on an output variable (for example, V(1)). The result is listed in the output (.OUT) file.

Find: the greatest difference from the nominal run (YMAX)

Threshold value:

Evaluate only when the sweep variable is in the: to:

Worst-Case direction

☐ Hi ☒ Low

☐ List model parameter values in the output file for each run

OK Cancel

(f) The Simulation Settings dialog box for Monte Carlo/Worst-Case analysis

```
** Profile: "SCHEMATIC1-tran1" [ C:\OrCAD\Yang\bjt_ampl-pspicefiles\schematic1\tran1.sim]
**** SORTED DEVIATIONS OF V(M03459) TEMPERATURE = 27.000 DEG C

WORST CASE SUMMARY
*****
Device MODEL PARAMETER NEW VALUE
Q_Q1 Q2N2222 BF 383.85 (Increased)
R_RL RMontel R 1.2 (Increased)
Mean Deviation = 1.1288E-03, Sigma = 0

RUN MAX DEVIATION FROM NOMINAL
WORST CASE ALL DEVICES 1.1288E-03 higher (102 % of Nominal) at T = 691.3000E-06

*****
Device MODEL PARAMETER NEW VALUE
Q_Q1 Q2N2222 BF 127.95 (Decreased)
R_RL RMontel R .8 (Decreased)
Mean Deviation = -2.6316E-03, Sigma = 0

RUN MAX DEVIATION FROM NOMINAL
WORST CASE ALL DEVICES 2.6316E-03 lower (104.4 % of Nominal) at T = 734.3000E-06
```

(g) The Worst Case simulation result printed in the output file

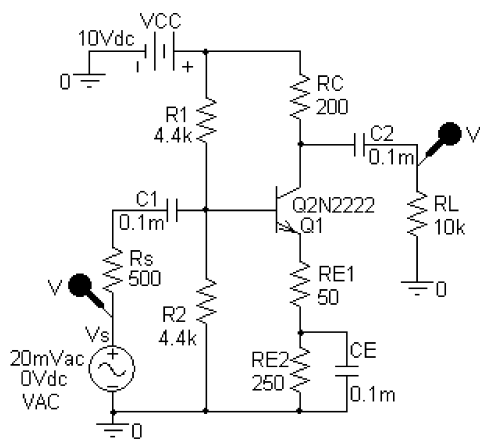
Figure H.19 (Continued)

2. Click the BJT(Q1) for selection so that it will be pink-hot and boxed by a dotted-line rectangle. Then click Edit/PSpice Model on the menu bar to open the PSpice Model Editor window (Figure H.19(b)), in which you can type 'Dev = 50%' (including the first blank space, but without the single quotation marks) just after 'Bf = 255.9' in the model statement. Then press the '^s' key to save the PSpice model for the device Q2N2222 placed in the schematic and click on x to close the editor window.
3. Click the load resistor RL for selection and press the 'Del' key to delete it. Then click the Place Part button to open the Place Part dialog box, search/select Rbreak[breakout.olb], and click OK to put it in place of RL.
4. Click the load resistor RL (having the part name 'Rbreak') for selection, click Edit/PSpice Model on the menu bar to open the PSpice Model Editor window (Figure H.19(b)), in which you can change the model name from Rbreak for RMonte1 and type 'Dev = 20%' just after 'R = 1' (the default value of the resistance) in the model statement. Then press the '^s' key to save the PSpice model for the device RMonte1 placed in the schematic and click on x to close the editor window.
5. Click the New Simulation Profile button on the toolbar to create a new simulation profile named, say, 'tran' and then click the Edit Simulation Settings button to open the Simulation Settings dialog box, in which you can do the following:
 - Set the Analysis type and Run_to_time to 'Time Domain (Transient)' and '2ms', respectively.
 - Select (click) Monte Carlo/Worst Case from the Options menu, select Monte Carlo from the upper-right menu, type 'V(RL:2)' into the Output variable field, and type '5' into the Number of Runs field, as depicted in Figure H.19(c).
 - Then click OK to close the Simulation Settings dialog box.
6. Click the Voltage/Level Marker button to put the voltage probe pins at the OP Amp output node and the + terminal of the sinusoidal input voltage source VSIN.
7. Click the Run button to perform the simulation to see the input/output waveforms in the Probe window as depicted in Figure H.19(d). You can also click the View Simulation Output File button to view the output file as illustrated in Figure H.19(e).
8. Click the Edit Simulation Settings button to reopen the Simulation Settings dialog box (Figure H.19(f)), in which you can do the following:
 - Select Monte Carlo/Worst Case in the Options menu, select Worst-case/Sensitivity from the upper-right menu, and select 'only DEV' in the tolerance field of the Worst-case/Sensitivity options box.
 - Click the More Settings button to open the Monte Carlo/Worst-Case Output File Options dialog box, in which you can select Hi or Low in the Worst-Case direction box.
 - Click OK to close the Monte Carlo/Worst-Case Output File Options dialog box and then click OK to close the Simulation Settings dialog box.
9. Click Run to perform the simulation and see the input/output waveforms in the Probe window. You can also click the View Simulation Output File button to view the output file as illustrated in Figure H.19(g), the contents of which depends on which of Hi and Low has been selected as the Worst-Case direction.

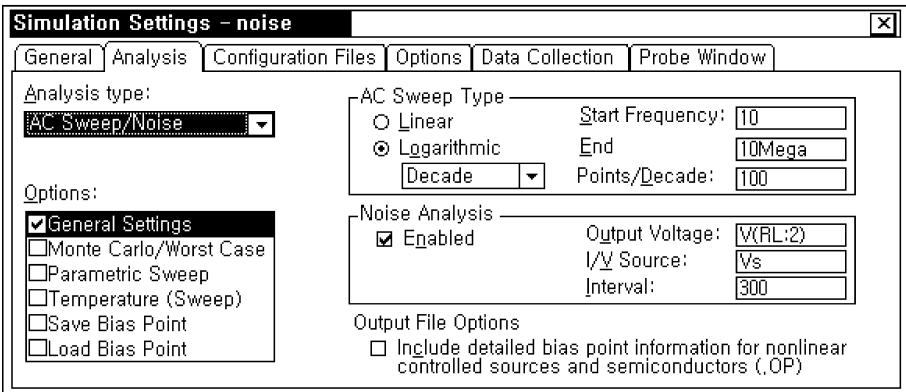
H.5.8 AC Sweep/Noise Analysis

Figures H.20(a) and (b) show the PSpice schematic of a CE (common emitter) BJT (bipolar junction transistor) amplifier circuit and the Simulation Settings dialog box for AC Sweep/Noise analysis, respectively. With the simulation profile made by setting the Simulation Settings dialog box as in

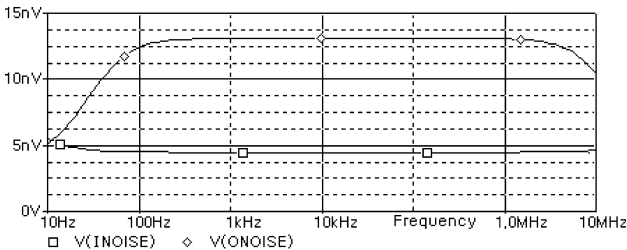
Figure H.20(b) and the Voltage/Level probe pins attached at the upper nodes of RL and Vs, the PSpice simulation can be run to get the input and output noises for the specified frequency range of 10 Hz–10 MHz as depicted in Figure H.20(c), where the output noise is the rms (root mean square) sum of all the device contributions propagated to a specified output and the input noise is the equivalent noise that



(a) The schematic of a BJT(Bipolar Junction Transistor) amplifier in the Capture window



(b) The Simulation Settings window for setting the analysis type to AC Sweep/Noise



(c) The Noise analysis result

Figure H.20 Example of noise analysis ('BJT_amp1_noise.opj')

```

**** 08/07/05 17:41:12 ***** PSpice Lite (Jan 2003) *****
** Profile: "SCHEMATIC1-noise" [C:\OrCAD\Yang\bjt_ampl_noise-pspicefiles\schematic1\noise.sim]
****      NOISE ANALYSIS              TEMPERATURE = 27.000 DEG C
*****
      FREQUENCY = 1.000E+01 HZ

**** SQUARED NOISE VOLTAGES (SQ V/HZ)
TRANSISTOR Q_Q1      RESISTOR
RB      2.172E-19      R_R1      8.993E-19
RC      1.770E-26      R_RC      3.180E-18
RE      0.000E+00      R_RE1     1.088E-18      TOTAL OUTPUT NOISE VOLTAGE = 2.622E-17 SQ V/HZ
IBSN     1.093E-17      R_R2      8.993E-19      = 5.120E-09 V/RT HZ
IC      1.464E-19      R_RS      7.186E-18
IBFN     0.000E+00      R_RL      1.040E-19
TOTAL    1.129E-17      R_RE2     1.569E-18
                                TRANSFER FUNCTION VALUE:
                                V(N09196)/V_Vs      = 9.312E-01
                                EQUIVALENT INPUT NOISE AT V_Vs = 5.499E-09 V/RT HZ

****
      FREQUENCY = 1.000E+04 HZ

**** SQUARED NOISE VOLTAGES (SQ V/HZ)
TRANSISTOR Q_Q1      RESISTOR
RB      2.131E-18      R_R1      8.037E-18
RC      1.410E-25      R_RC      3.168E-18
RE      0.000E+00      R_RE1     1.067E-17      TOTAL OUTPUT NOISE VOLTAGE = 1.717E-16 SQ V/HZ
IBSN     6.774E-17      R_R2      8.037E-18      = 1.310E-08 V/RT HZ
IC      1.135E-18      R_RS      7.073E-17
IBFN     0.000E+00      R_RL      6.336E-20
TOTAL    7.101E-17      R_RE2     2.162E-23
                                TRANSFER FUNCTION VALUE:
                                V(N09196)/V_Vs      = 2.921E+00
                                EQUIVALENT INPUT NOISE AT V_Vs = 4.486E-09 V/RT HZ

****
      FREQUENCY = 1.000E+07 HZ

**** SQUARED NOISE VOLTAGES (SQ V/HZ)
TRANSISTOR Q_Q1      RESISTOR
RB      1.260E-18      R_R1      4.755E-18
RC      2.224E-21      R_RC      2.127E-18
RE      0.000E+00      R_RE1     6.348E-18      TOTAL OUTPUT NOISE VOLTAGE = 1.089E-16 SQ V/HZ
IBSN     4.003E-17      R_R2      4.755E-18      = 1.043E-08 V/RT HZ
IC      7.712E-18      R_RS      4.184E-17
IBFN     0.000E+00      R_RL      4.253E-20
TOTAL    4.901E-17      R_RE2     1.286E-29
                                TRANSFER FUNCTION VALUE:
                                V(N09196)/V_Vs      = 2.247E+00
                                EQUIVALENT INPUT NOISE AT V_Vs = 4.644E-09 V/RT HZ

```

(d) The output file obtained from PSpice noise analysis

Figure H.20 (Continued)

would be needed at the input source to generate the calculated output noise in an ideal (noiseless) circuit. You can also view the output file (Figure H.20(d)) containing every device noise, i.e. the noise contribution propagated to the specified output from every resistor and semiconductor device in the circuit. Note that the three frequencies $f = 10$, 10^4 , and 10^7 Hz at which the noise analysis has been done are extracted with the interval of 300 points from the total 601 frequency points (100 points per decade) from 10 Hz to 10 MHz as specified in the Simulation Settings dialog box (Figure H.20(b)).

Appendix I: MATLAB Introduction (Reference [W-2])

Once MATLAB has been installed, you can click the icon like the one below to run



MATLAB. Then you will see the MATLAB command window on your monitor as depicted in Figure I.1, where a cursor appears (most likely blinking) to the right of the prompt like '>>' waiting for you to type in commands/statements. If you are running MATLAB of version 6.x or above, the main window has not only the command window but also the workspace and command history boxes on the left-up/down side of the command window, in which you can see the contents of MATLAB memory and the commands you have typed into the Command window up to the present time, respectively (see Figure I.1). You might clear the boxes by clicking the corresponding submenu under the 'Edit' menu and even remove/restore them by clicking the corresponding submenu under the 'Desktop/Desktop_{\rm L}ayout' menu.

How do we work with the MATLAB command window?

- By clicking 'File' on the top menu and then 'New'/'Open' in the File pull-down menu, you can create/edit any file with the MATLAB editor.
- By clicking 'File' on the top menu and then 'Set_Path' in the File pull-down menu, you can make the MATLAB search path list include or exclude the paths containing the files you want to or not to be run.
- If you are a beginner in MATLAB, then it may be worthwhile to click 'Help' on the top menu, 'Demos' in the Help pull-down menu, (double-)click any topic that you want to learn about, and watch the visual explanation about it.
- By typing any MATLAB commands/statements in the MATLAB command window, you can use various powerful mathematic/graphic functions of MATLAB.
- If you have an M-file that contains a series of commands/statements composed for performing a target procedure, you can type in the file name (without the extension '.m') to run it.

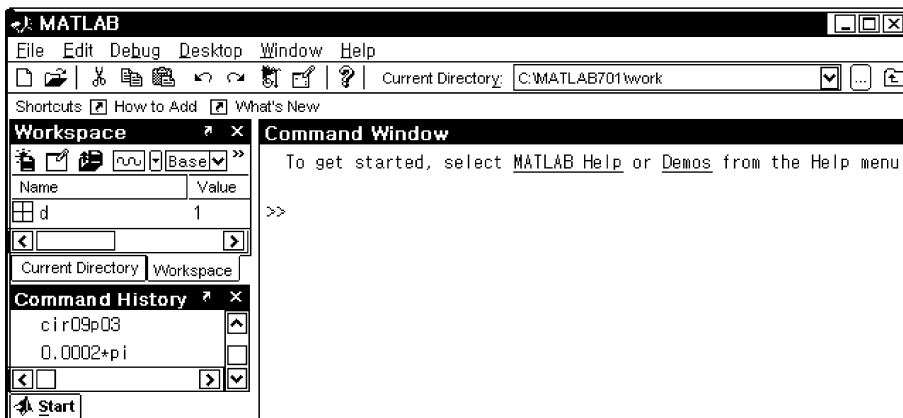


Figure I.1 The MATLAB command window with the workspace and command history boxes

Table I.1 Functions and Variables in MATLAB

Function	Remark	Function	Remark
cos(x)		exp(x)	Exponential function
sin(x)		log(x)	Natural logarithm
tan(x)		log10(x)	Common logarithm
acos(x)	$\cos^{-1}(x)$	abs(x)	Absolute value
asin(x)	$\sin^{-1}(x)$	angle(x)	Phase of a complex number [rad]
atan(x)	$-\pi/2 \leq \tan^{-1}(x) \leq \pi/2$	sqrt(x)	Square root
atan2(y,x)	$-\pi \leq \tan^{-1}(y,x) \leq \pi$	real(x)	Real part
cosh(x)	$(e^x + e^{-x})/2$	imag(x)	Imaginary part
sinh(x)	$(e^x - e^{-x})/2$	conj(x)	Complex conjugate
tanh(x)	$(e^x - e^{-x})/(e^x + e^{-x})$	round(x)	The nearest integer (round-off)
acosh(x)	$\cosh^{-1}(x)$	fix(x)	The nearest integer towards 0
asinh(x)	$\sinh^{-1}(x)$	floor(x)	The greatest integer $\leq x$
atanh(x)	$\tanh^{-1}(x)$	ceil(x)	The smallest integer $\geq x$
max	Maximum and its index	sign(x)	1 (positive) / 0 / -1 (negative)
min	Minimum and its index	mod(y,x)	Remainder of y/x
sum	Sum	rem(y,x)	Remainder of y/x
prod	Product	eval(f)	Evaluate an expression
norm	Norm	feval(f,a)	Function evaluation
sort	Sort in the ascending order	polyval	Value of a polynomial function
clock	Present time	poly	Polynomial with given roots
find	Index of element(s)	roots	Roots of polynomial
tic	Start a stopwatch timer	toc	Read the stopwatch timer (elapsed time from tic)
date	Present date		
<i>Reserved variables with special meaning</i>			
i, j	$\sqrt{-1}$	pi	π
eps	Machine epsilon	Inf, inf	Largest number (∞)
realmax, realmin	Largest/smallest positive number	NaN	Not_a_Number (undetermined)
end	End of for-loop or if, while	break	Exit while/for loop
nargin	Number of input arguments	nargout	Number of output arguments
varargin	Variable input argument list	varargout	Variable output argument list

Table I.2 Graphic line specifications used in the `plot( )` command

Line type	Point type (marker symbol)			Color	
- solid line	. (dot)	+ (plus)	*(asterisk)	r: red	m: magenta
: dotted line	^: Δ	>:>	o (circle)	g: green	y: yellow
-- dashed line	p: ☆	v: ∇	x: x-mark	b: blue	c: cyan (sky blue)
-. dash-dot	d: $\diamond$	<:<	s: $\square$	k: black	

Note the following:

- (a) the index of an array in MATLAB starts from 1, not 0, and
- (b) a dot(.) must be put before an operator to make a termwise (element-by-element) operation.

Some of mathematical functions and special reserved constants/variables defined in MATLAB are listed in Table I.1. Table I.2 shows the graphic line specifications used in the `plot( )` command.

Appendix J: Solutions to Problems**1 Basic Concepts on Electric Circuits**

1.1 (a) $R = 5.35 \times 10^{-2} \Omega$

(b) $L = 1.98 \times 10^{-2} \text{H}$

(c) $C = 8.854 \times 10^{-10} \text{F}$

1.2 (a) $I = 1 \text{ A}, R = 100 \Omega$

(b) $I = 0.1 \text{ A}, V = 100 \text{ V}$

1.3 (a) $I_1 = I_2 = I = 4 \text{ A}, P_1 = 16 \text{ W}, P_2 = 32 \text{ W}$

(b) $P_1 = 144 \text{ W}, P_2 = 72 \text{ W}$

1.4 (a) $i = 3 \text{ A}; i_{R1} = -3 \text{ A}, v_{R1} = 9 \text{ V}$

(b) $p_{R1} = -27 \text{ W}$

(c) $p_{R1} = 27 \text{ W}, p_{s1} = -36 \text{ W}, p_{s2} = 9 \text{ W}$

1.5 $V_o = 1.5 \text{ V}$

1.7 (a) $n - 1 = 2$ and $b - (n - 1) = 1$

(b)
$$\begin{bmatrix} \frac{1}{R_1} + \frac{1}{R_2} & -\frac{1}{R_2} \\ -\frac{1}{R_2} & \frac{1}{R_2} + \frac{1}{R_3} \end{bmatrix} \begin{bmatrix} v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} \frac{V_{s1}}{R_1} + I_{s1} \\ 0 \end{bmatrix}$$

(c)
$$\begin{bmatrix} R_1 & R_2 + R_3 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} V_{s1} \\ I_{s1} \end{bmatrix}$$

1.8 (a) $n - 1 = 3 - 1 = 2, b - (n - 1) = 2$

(b)
$$\begin{bmatrix} 11 & -2 \\ -4 & 7 \end{bmatrix} \begin{bmatrix} v_3 \\ v_4 \end{bmatrix} = \begin{bmatrix} 138 \\ -276 \end{bmatrix}$$

(c)
$$\begin{bmatrix} 3 & -2 \\ -2 & 9 \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} 12 \\ 4 \times 23 - 6 - 7 \times 20 = -54 \end{bmatrix}$$

1.9 (b) $i_{R_2} = \frac{1}{R_1 + R_2} (V_{s2} + R_1 I_{s2})$

(c) $i_{R_2} = \frac{V_{s2} + R_1 I_{s2}}{R_1 + R_2}$

1.10 (a) $i_{R_L} = \frac{v_1}{R_L} = \frac{R_1 V_{s2} + R_2 V_{s1}}{R_1 R_2 + R_1 R_L + R_2 R_L}$

(b) $i_{R_L} = \frac{V_s}{R + R_L} = \frac{R_1 V_{s2} + R_2 V_{s1}}{R_1 R_2 + R_1 R_L + R_2 R_L}$

1.11 $i_{4\Omega} = -\frac{42 + 4}{(2/3) + 3 + 4} = -6 \text{ A}$

2 Resistor Circuits

2.1 (a) 1, 2, 3, 4, 3/4, 2/3, 5/3, 2/5, 1/2, 5/2, 3/2, 3/5, 1/3, 4/3, 1/4 Ω

(b) Neither $R_1 + R_2 + R_3$ nor $(R_1 + R_2) \parallel R_3$

(c) Neither $(R_1 \parallel R_2) + R_3$ nor $R_1 \parallel R_2 \parallel R_3$

2.2 (a) $R_2 = 4.95 \text{ k}\Omega$, $R_1 = 45 \text{ k}\Omega$

(b) $R_b = 111 \Omega$, $R_d = 11 \Omega$

2.3 $R_s = 50 \Omega$

2.4 (a) $R_2 = 2 \text{ k}\Omega$, $R_1 = 8 \text{ k}\Omega$

(b) $R_1 = 1 \text{ k}\Omega$, $R_2 = 10 \text{ k}\Omega$

2.5 (a) $R_{10} = 170/71$

(b) $i_{20} = 42 \text{ A}$, $i_{30} = 29 \text{ A}$, $v_2 = 126 \text{ V}$, $v_3 = 116 \text{ V}$, $i_{12} = 44 \text{ A}$, $i_{13} = 27 \text{ A}$, $i_{23} = 2 \text{ A}$, for (a) and (b)

2.6 (b) $v_1 = 170$, $v_2 = 126$, $v_3 = 116$; $i_{23} = 2 \text{ A}$

(c) $i_1 = 71$, $i_2 = 27$, $i_3 = 29$; $i_{23} = 2 \text{ A}$

2.7 $v_2 = v_e = \frac{(1 + g_m R_{be}) R_e}{R_s + R_{be} + (1 + g_m R_{be}) R_e} V_s$, $i_1 = \frac{v_{be}}{R_{be}} = \frac{v_b - v_e}{R_{be}} = \frac{V_s}{R_s + R_{be} + (1 + g_m R_{be}) R_e}$

2.8 $V_{Th} = \frac{\sum V_k / R_k}{\sum (1/R_k)}$, $R_{Th} = \frac{1}{\sum (1/R_k)}$

2.9 (a) $V_{Th} = (170/12) \text{ V}$, $R_{Th} = (25/12) \Omega$; $i_{23} = 2 \text{ A}$

(b) $V_{Th} = (71/5) \text{ V}$, $R_{Th} = (21/10) \Omega$; $i_{23} = 2 \text{ A}$

2.10 $V_{Th} = (\Delta R / R) V_s$, $R_{Th} = [R^2 - (\Delta R)^2] / R \simeq R$

2.11 $R_{Th} = 30 \Omega$

2.13 $V_{Th} = \frac{G_s (G_f - g_m)}{G_o (G_s + G_{be} + G_f) + G_f (G_s + G_{be} + g_m)} V_s$, $R_{Th} = \frac{G_s + G_{be} + G_f}{G_o (G_s + G_{be} + G_f) + G_f (G_s + G_{be} + g_m)}$

2.14 $V_{Th} = \frac{R_2}{R_1 + R_2} V_s$, $R_{Th} = \frac{R_1 R_2}{R_1 + R_2}$

2.15 (a) $V_{Th} = (5/2) \text{ V}$, $R_{Th} = (1/36) \Omega$

(b) $V_{Th} = 2 \text{ V}$, $R_{Th} = (1/18) \Omega$

2.16 $V_{Th} = I_s [V]$, $R_{Th} = 2 \Omega$

2.17 (a) $R_{10} = R$

(b) $i_{41} = V_{FS}/2R$, $i_{52} = V_{FS}/4R$, $i_{63} = V_{FS}/(8R)$

(c) For $d_2 d_1 d_0 = 100$, $v_o = V_{FS}/2$. For $d_2 d_1 d_0 = 010$, $v_o = V_{FS}/4$. For $d_2 d_1 d_0 = 001$, $v_o = V_{FS}/8$.

2.18 $v_o = \frac{R_{P2}}{R_{P1} + R_{P2}} v_{iP} - \frac{R_{N2}}{R_{N1}} \left(v_{iN} - \frac{R_{P2}}{R_{P1} + R_{P2}} v_{iP} \right)$

- 2.21 $v_{o3} = (5/9)(v_F - 3.2)$
- 2.23 (a) $v_o/v_i = 1$ (b) $v_o/v_i = -1$
- 2.25 $i_{RL} = mV_R/R_1$
- 2.27 (b) $R_{in} = -1 \text{ k}\Omega$
- 2.28 (a) $V_{Th} = 6 \text{ V}$, $R_{Th} = 1.5 \Omega$
(b) The load line: $v = V_{Th} - R_{Th} i = 6 - 1.5 i$, The operating point: $(I_Q, V_Q) = (2 \text{ A}, 3 \text{ V})$
(c) $R_{2s} = 1.5 \Omega$, $r_{2d} = 0.75 \Omega$
(d) $i(t) = 2 + 0.2 \sin \omega t$, $v(t) = 3.6 + 0.18 \sin \omega t$

3 First-Order Circuits

- 3.1 (a) $t = (1/10) \ln 5 \text{ s}$
(b) $t = (1/20) \ln 5 \text{ s}$
- 3.3 (a) $v(t) \stackrel{(3.39)}{=} (V_2 - V_1)e^{-t/T_{ON}} + V_1$
(b) $v(t) \stackrel{(3.39)}{=} (V_1 - V_2)e^{-t/T_{OFF}} + V_2$
- 3.4 (a) $v_C(t) = V_i(1 - e^{-t/(R_1+R_2)C})$ for $t \geq 0$
(b) $v_C(t) = V_i e^{-t/(R_2+R_3)C}$ for $t \geq 0$
- 3.5 (a) $v_i(t) = \begin{cases} (V_{OL} - V_{OH})e^{-t/(RC)} + V_{OH} & \text{for rising} \\ (V_{OH} - V_{OL})e^{-t/(RC)} + V_{OL} & \text{for falling} \end{cases}$
(b) $T_R = RC \ln \frac{2.4 - 0.4}{2.4 - 2.0} \approx 1.61RC$, $T_F = RC \ln \frac{2.4 - 2.0}{0.8 - 0.4} \approx 1.61RC$
- 3.6 (a) Charging time constant: $T_c = 5 \times 10^{-5} \text{ s}$, Discharging time constant: $T_D = 5 \times 10^{-2} \text{ s}$
(b) $V_L = 3.23 \text{ V}$, $T_R = 0.0018 \text{ s}$, $T_F = 0.0149 \text{ s}$
(c) $V_L = 3.26 \text{ V}$, $T_R = 0.002 \text{ s}$, $T_F = 0.0147 \text{ s}$
(d) $v_o(t) = 3.82 \cos(377t) + 3.22 \sin(377t) + 0.00639 e^{-20020t}$
(e) $v_o(t) = 4.35 e^{-20t} \text{ V}$
- 3.7 Charging time constant: $T_c \simeq 349 \times 10^{-5} \text{ s}$, Discharging time constant: $T_D = 33 \times 10^{-3} \text{ s}$
- 3.8 (a) $v_C(t) = \begin{cases} (V_L - V_2)e^{-t/(RC)} + V_2 & \text{for rising} \\ (V_H - V_1)e^{-t/(RC)} + V_1 & \text{for falling} \end{cases}$
(b) $T_H = T \ln \frac{V_2 - V_L}{V_2 - V_H}$, $T_L = T \ln \frac{V_H - V_1}{V_L - V_1}$
- 3.9 (a) $C_2 V_s$
(b) $T_H = R_p(C_p + C_n) \ln \frac{V_s - V_L}{V_s - V_H}$,
 $T_L = R_n(C_p + C_n) \ln \frac{V_H}{V_L}$
- 3.10 (a) $v_C(t) = 2.5 - 0.5e^{-36t}$
(b) $v_C(t) = 2 + (1/2)e^{-18t}$

- 3.11 (a) $i_L(t) = 10 e^{-R_{eq}t/L} = 10 e^{-t/2}$
 (b) $i_L(t) = 10(1 - e^{-t/2})$
- 3.12 (a) $v_C(t) = 10 e^{-t/(R_{eq}C)} = 10 e^{-t/2}$
 (b) $v_C(t) = 10(1 - e^{-t/2})$
- 3.14 (a) $v_C(t) = \begin{cases} 10 e^{-t/2} u_s(t) & \text{for } 0 \leq t < 2 \\ 10[(e^{-1} - 1)e^{-(t-2)/2} + 1] u_s(t-2) & \text{for } t \geq 2 \end{cases}$
 (b) $v_C(t) = \begin{cases} 10(1 - e^{-t/2}) u_s(t) & \text{for } 0 \leq t < 2 \\ 10(1 - e^{-1}) e^{-(t-2)/2} u_s(t-2) & \text{for } t \geq 2 \end{cases}$
- 3.16 (b) $P = T_1 + T_2 = \frac{4R_3}{R_2} R_1 C \xrightarrow{\text{if } R_3 > R_2} 4R_1 C$
- 3.17 (a) $G_a(s) = \frac{1/(sC)}{R + 1/(sC)}$, (b) $G_b(s) = \frac{R + 1/(sC)}{R}$
- 3.19 (a) $V_2(s) = \frac{1/R}{1/R + 1/R_1 + 1/R_2} V_i(s)$
- 3.20 (b) 12.550V is close to $V_{om} = 12.56$ V
 (d) 6.3659 V $\simeq 2 \times (V_{om} = 6.3662/2)$
 (e) $V_o(s) = [2/(sR_1C)] V_i(s)$
 (f) 6.3419 V is close to $V_{om} = 3.1831$ V
 (g) $V_o(s) = sRCV_i(s)$
 (h) 6.2830 V $\simeq 2 \times (V_{om} = 6.2832)$ V
- 3.23 (a) $T_D \simeq 1.1$
 (b) $T_D \simeq 1.1$
- 3.25 (a) $C = 10^{-4} = 100$ μ F
 (b) $v_C(t) = 1$ for $t \geq 0$
 (c) $v_C(t) = e^{+t}$ for $t \geq 0$
- 3.26 (a) $R_1 = 1306 \Omega$ and $R_3 = 96\,951 \Omega$
 (b) $R_1 = 1300 \Omega$ and $R_3 = 97\,600 \Omega$; $R_2 = 10$ k Ω : Period = 4.472 s;
 $R_2 = 100$ k Ω : Period = 0.495 s

4 Second-Order Circuits

- 4.1 (a) $v_C(t) = 2 - e^{-32\,000t}(\cos 24\,000t + \sin 24\,000t)$
 (b) $v_C(t) = 3 \sin 40\,000t - 5 e^{-32\,000t} \sin 24\,000t$
- 4.2 (b) $v_2(t) = e^{-10\,000t}(\cos 20\,000t - \sin 20\,000t)$
- 4.3 (a) $i_L(0) = 0$ and $v_C(0) = 4$ V
 (b) $v_C(t) = 2(2 + e^{-t} - e^{-3t/2})u_s(t)$
- 4.4 $v_C(t) = 2 - e^{-3000t}(2 \cos 4000t - \sin 4000t)$

$$4.5 \quad (a) \quad v_C(t) = [2 - e^{-2t}(2 \cos t + 4 \sin t)] u_s(t), \quad i_L(t) = [4 - e^{-2t}(4 \cos t - 2 \sin t)] u_s(t)$$

$$(b) \quad v_C(t) = 2e^{-2t}(\cos t + 2 \sin t)(t), \quad i_L(t) = e^{-2t}(4 \cos t - 2 \sin t) u_s(t)$$

$$4.6 \quad \frac{V_o(s)}{V_i(s)} = \frac{KG_1G_2/C^2}{s^2 + [G_1 + (2-K)G_2]s/C + G_1G_2/C^2}$$

$$4.7 \quad \frac{V_o(s)}{V_i(s)} = \frac{KG^2/C^2}{s^2 + (3-K)Gs/C + G^2/C^2}$$

$$4.10 \quad v_C(t) = 8e^{-t}u_s(t), \quad i_L(t) = 5(e^{-t} - e^{-5t})u_s(t) \\ i_{R_2}(t) = (-e^{-t} + 5e^{-5t})u_s(t)$$

$$4.11 \quad (a) \quad \frac{V_o(s)}{V_i(s)} = \frac{-G_1G_4}{C_2C_5s^2 + C_5(G_1 + G_3 + G_4)s + G_3G_4}$$

$$(b) \quad \frac{V_o(s)}{V_i(s)} = \frac{-C_1C_4s^2}{C_3C_4s^2 + G_5(C_1 + C_3 + C_4)s + G_2G_5}$$

$$4.13 \quad (c) \quad V_{\text{im,max}} = 5 \times 11.6 = 58 \text{ V}$$

$$4.14 \quad G(s) = \frac{V_o(s)}{V_i(s)} = \frac{1/(R^2C^2)}{s^2 + [2/(RC)]s + 2/(R^2C^2)}$$

$$4.15 \quad (d) \quad V_{\text{om}} = 3\sqrt{2} \simeq 4.2 \text{ V}, \quad T = 2\pi/\omega_r \simeq 6.28 \text{ ms}$$

$$4.16 \quad (a) \quad C_3 = C_4 = 22 \text{ uF}, \quad R_1 = R_2 = 51 \Omega$$

$$(b) \quad v_o(t) = -10e^{-8.913t} \sin 40\pi t u_s(t) [\text{V}]$$

$$(c) \quad 0.05 \equiv 2/\omega_d = 2/40\pi = 1/20$$

5 Magnetically Coupled Circuits

$$5.3 \quad (a) \quad \text{Negative in both of them}$$

$$(b) \quad v_3(t) = 3(e^{-2t/5} - e^{-2t/3}) u_s(t)$$

$$(c) \quad v_3(t) = 3(e^{-2t/3} - e^{-2t/5}) u_s(t)$$

$$5.4 \quad (a) \quad \begin{bmatrix} I_1(s) \\ I_2(s) \end{bmatrix} = \frac{V_i(s)}{\Delta} \begin{bmatrix} R_2 + sL_2 \\ -sM \end{bmatrix}, \quad \Delta = (L_1L_2 - M)^2 + (L_1R_2 + L_2R_1)s + R_1R_2$$

$$(b) \quad V_1(s) = \frac{[sL_1(R_2 + sL_2) - (sM)^2]V_i(s)}{\Delta}, \quad V_2(s) = \frac{sR_2MV_i(s)}{\Delta}$$

$$(c) \quad V_1(s) = \frac{sL_1R_2V_i(s)}{(L_1R_2 + L_2R_1)s + R_1R_2}, \quad V_2(s) = \frac{sR_2MV_i(s)}{(L_1R_2 + L_2R_1)s + R_1R_2}; \quad \frac{V_2(s)}{V_1(s)} = \frac{M}{L_1} = \sqrt{\frac{L_2}{L_1}} = \frac{N_2}{N_1}$$

$$5.6 \quad V_L(s) = \frac{3 \times (4/15)}{s^2 + 16s/15 + 4/15}$$

$$5.7 \quad (a) \quad i_1(0) = 2.4 \text{ A}, \quad i_2(0) = 0 \text{ A}, \quad v_C(0) = 0 \text{ V}$$

$$(b) \quad v_2(t) = -15e^{-25t} + 15e^{-5t}(\cos 10t - 2 \sin 10t)$$

$$5.8 \quad (a),(b) \quad V_2(s) = \frac{2(2s^2 + 1)}{6s^3 + 4s^2 + 4s + 1}$$

$$(c) \quad v_L(t) = 0.7329e^{-0.2994t} + e^{-0.1836t}(-0.0662 \cos 0.7232t - 0.328 \sin 0.7232t)$$

$$(e) \quad V_2(s) = \frac{2(2s^2 - 1)}{6s^3 + 12s^2 + 4s + 1},$$

$$v_L(t) = 0.6567e^{-1.6586t} + e^{-0.1707t} (0.01 \cos 0.2671t - 0.9076 \sin 0.2671t)$$

$$5.9 \quad (b) \quad i_2(t) = e^{-3t/10} [\cos(t/10) - 3 \sin(t/10)] u_s(t) [\text{A}]$$

$$5.10 \quad (b) \quad i_2(t) = [-2e^{-t} + 2e^{-t/10} \cos(t/10)] u_s(t) [\text{A}]$$

$$5.11 \quad I_2 = (N_1/N_2)I_1, N_1 = 1 \text{ turn}$$

6 AC Circuits

$$6.1 \quad X_C = \sqrt{\frac{a}{1-a}}R, \quad X_L = \sqrt{a(1-a)}R$$

$$6.2 \quad \mathbf{V}_3 = 10; v_s(t) = 10 \cos(100t)$$

$$6.3 \quad \frac{-200s^2(8s^2 + 16795s + 1599500)}{(s + 100)(7s^3 + 996200s^2 + 107925 \times 10^4s + 831 \times 10^8)}$$

$$6.4 \quad (a) \quad \frac{\mathbf{V}_o(j\omega)}{\mathbf{V}_i(j\omega)} = -\frac{R_2}{R_1(1 + j\omega R_2 C)}$$

$$(b) \quad \frac{\mathbf{V}_o(j\omega)}{\mathbf{V}_i(j\omega)} = -\frac{j\omega R_2 C}{1 + j\omega R_1 C}$$

$$6.5 \quad (a) \quad V_o(s) = -\frac{1 + sR_2 C_2}{sR_2 C_2(1 + sR_1 C_1)} V_i(s)$$

$$(c) \quad C_1 \geq 10^{-6} \text{F} = 1 \mu\text{F}$$

$$6.6 \quad C_2 = 33.16 \mu\text{F}$$

$$6.7 \quad (a) \quad \mathbf{V}_2 = -1 - j$$

$$6.8 \quad \mathbf{I} = \mathbf{I}_1 - \mathbf{I}_2 = 5 - j2 \text{ A}$$

$$6.9 \quad (a) \quad \mathbf{V}_{\text{Th}} = -j10, Z_{\text{Th}} = 1 - j$$

$$(b) \quad \mathbf{V}_{\text{Th}} = 2(1 - j) Z_{\text{Th}} = 8 - j6$$

$$6.12 \quad (a) \quad R_c = 480 \Omega; L_c = 0.7351 \text{ H}$$

$$(b) \quad R_l = 0.8 \Omega; L_l = 0.0016 \text{ H}$$

$$6.13 \quad PF_0 = 0.7452, X_c = -9.298 \Omega \text{ for } Q_c = -1075.5 \text{ VAR}, P_{\text{loss}} = 6500 \text{ W} \rightarrow 4000 \text{ W}, \\ \text{Voltage variation} = 26\% \rightarrow 20.4\%, \text{ Generator voltage } |\mathbf{V}_s| = 123.1 \text{ V} \rightarrow 120.3 \text{ V}$$

$$6.14 \quad I_l : I_L = 1 : 30, P_{\text{loss}} \rightarrow (1/900)\text{times}$$

$$6.15 \quad R_s = 1.92 \Omega, L = 1.44 \mu\text{H}, C = 2 \mu\text{F}, K = 4$$

$$6.16 \quad C = \frac{\sqrt{(R_s - R_L)/R_L}}{\omega R_s}, L = R_s R_L C$$

$$6.18 \quad \mathbf{I}_1 = 2.11 \angle -71.6^\circ, \quad \mathbf{I}_2 = 1.98 \angle 63.5^\circ, \quad P_{\text{transformer}} = 44.29 \text{ W}, \quad P_{\text{load}} = 31.46 \text{ W}, \\ \text{Power efficiency} = 71\%$$

7 Three-Phase AC Circuits

7.1 (b) $C_{AB} = 49.3 \mu\text{F}$, $C_{BC} = 23.8 \mu\text{F}$, $C_{CA} = 28.5 \mu\text{F}$

	(a)	(c)
V_A	$112 \angle -1.58^\circ$	$117 \angle -2.49^\circ$
V_B	$113 \angle -120^\circ$	$117 \angle -122^\circ$
V_C	$115 \angle 119^\circ$	$118 \angle 118^\circ$
I_a	$7.58 \angle -42.8^\circ$	$5.15 \angle -3.33^\circ$
I_b	$5.98 \angle -179^\circ$	$4.22 \angle -128^\circ$
I_c	$5.25 \angle 85.5^\circ$	$4.40 \angle 125^\circ$

7.2

	(a)	(c)
V_A	$111 \angle -1.51^\circ$	$116 \angle -2.42^\circ$
V_B	$113 \angle -121^\circ$	$117 \angle -122^\circ$
V_C	$115 \angle 119^\circ$	$119 \angle 118^\circ$
I_a	$8.12 \angle -45.2^\circ$	$5.39 \angle -10.2^\circ$
I_b	$5.94 \angle -175^\circ$	$4.57 \angle -124^\circ$
I_c	$4.93 \angle 83.4^\circ$	$4.02 \angle 126^\circ$

8 Frequency Selective Circuit – Filters

8.1 (c) $\omega_r = 1414 \text{ rad/s}$, $G_{\max} = |G(j\omega_r)| = 0.5882$, $\omega_l = 511$, $\omega_u = 3911 \text{ rad/s}$

8.4 $R_1 = 500 \Omega$, $R_4 = 200 \text{ k}\Omega$

8.5 (a) $G_a(s) = \frac{C_1}{C_2} \frac{s + 1/(R_1 C_1)}{s + 1/(R_2 C_2)}$, $G_b(s) = \frac{-R_2 + 1/(sC)}{R_1 + 1/(sC)} = -\frac{R_2}{R_1} \frac{s - 1/(R_2 C)}{s + 1/(R_1 C)}$

(b) (1) (a) Phase-lag, (2) (b) Phase-lead, (3) (c) All-pass

8.6 (a) (1): (a) LPNF, (2): (b) HPNF, (3): (c) APF

(b) (1) $|G(j\omega_p)| = 4.16$ at $\omega_p = 53.5 \text{ rad/s}$, (2) $|G(j\omega_p)| = 1.63$ at $\omega_p = 224 \text{ rad/s}$

8.7 (j) Figure P8.7(c1): LP, (d1):HP, (e1):BP, (f1):BS, (g1):LPN, (h1):HPN, (i1):AP

8.8 (b) $R = 1 \text{ k}\Omega$, $R_2 = 7 \text{ k}\Omega$, $R_B = 7 \text{ k}\Omega$

(d) (1) LP, (2) HP, (3) positive BP, (4) negative BP, (5) BS, (6) AP

8.12 (a) $C_{13} = 4.16 \mu\text{F}$, $C_{14} = 0.609 \mu\text{F}$, $C_{23} = 1.72 \mu\text{F}$, $C_{24} = 1.47 \mu\text{F}$

(b) $R_1 = 82 \Omega$, $R_2 = 34 \Omega$, $R_f = 58.5 \Omega$, $C_1 = 25.3 \mu\text{F}$

$$8.13 \quad (a) \quad R_{11} = 5000 \Omega, R_{12} = 139.2 \Omega, R_{21} = 5000 \Omega, R_{22} = 97.38 \Omega$$

9 Circuit Analysis Using Fourier Series

$$9.1 \quad (c) \quad y_c(t) = \frac{2A}{k\pi} \sum_{k=1}^{\infty} (-1)^{k+1} \sin(k\omega_0 t), \quad y_d(t) = \frac{2A}{k\pi} \sum_{k=1}^{\infty} \sin(k\omega_0 t)$$

$$9.3 \quad (c) \quad v_{o,L}(t) = \sum_{k=2m+1}^{\infty} d'_k \sin(k\omega_0 t + \varphi_k) \text{ with } d'_k = \frac{4V_m}{k\pi \sqrt{1 + (k\omega_0 RC)^2}} \text{ and } \varphi_k = -\tan^{-1} k\omega_0 RC$$

- 9.4 (a) The lower bound of the ratio is 1/9
 (b) The lower bound of the ratio is 1/27
 (c) 63 662 Ω

$$9.6 \quad (a) \quad v_i: d'_0 = 0, d'_1 = 8.11, d'_3 = 0.901, d'_5 = 0.324; \\ v_o: d'_0 = 0, d'_1 = 8.07, d'_3 = 0.0675, d'_5 = 0.0135$$

10 Two-Port Networks

$$10.1 \quad (b) \quad I_{SC} = -y_{21} = \frac{G_2 G_3 - G_1 G_4}{G_1 + G_2 + G_3 + G_4}$$

$$(c) \quad V_{OC} = \frac{R_2 R_3 - R_1 R_4}{(R_1 + R_2)(R_3 + R_4)} V_1 = V_{Th}$$

$$(d) \quad Z_{Th} = \frac{R_1 R_2}{R_1 + R_2} + \frac{R_3 R_4}{R_3 + R_4} = \text{Equation (E2.18.2)}$$

$$(e) \quad i_{RL} = 2 \text{ A}$$

$$10.2 \quad (a) \quad \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} = \begin{bmatrix} sL + R_1 & 0 \\ \beta R_2 & R_2 \end{bmatrix}$$

$$(b) \quad \begin{bmatrix} r_b + \frac{r_c r_e}{(1-\alpha)r_c + r_e} & \frac{r_e}{(1-\alpha)r_c + r_e} \\ \frac{\alpha r_c - r_e}{(1-\alpha)r_c + r_e} & \frac{1}{(1-\alpha)r_c + r_e} \end{bmatrix}$$

$$(c) \quad - \begin{bmatrix} n & nR_2 + 1/(nsC_1) \\ 0 & 1/n \end{bmatrix}$$

$$10.3 \quad A = \begin{bmatrix} 1 & 0 \\ 0 & -\frac{Z_2 Z_4}{Z_1 Z_3} \end{bmatrix}$$

$$10.4 \quad (b) \quad \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} = \begin{bmatrix} 3 & 2 \\ K + 2 & 5 \end{bmatrix}$$

$$(c) \quad \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} = \begin{bmatrix} (11 - 2K)/5 & 2/5 \\ -(K + 2)/5 & 1/5 \end{bmatrix}$$

$$10.5 \quad (b) \quad \begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} = \frac{1}{2R(sRC + 1)} \begin{bmatrix} s^2 R^2 C^2 + 4sRC + 1 & -(s^2 R^2 C^2 + 1) \\ -(s^2 R^2 C^2 + 1) & s^2 R^2 C^2 + 4sRC + 1 \end{bmatrix}$$

$$(c) \quad A_v = \frac{s^2 + 1/R^2 C^2}{s^2 + 4s/(RC) + 1/(R^2 C^2)}$$

$$10.6 \quad (a) \quad Y = \begin{bmatrix} 1 + s + 3s & -3s \\ K - 3s & 2 + 2s + 3s \end{bmatrix}$$

$$(b) \quad A = \begin{bmatrix} -\frac{5s+2}{3s} & \frac{11s^2 + (13-3K)s + 2}{3s} \\ \frac{1}{3s} & -\frac{4s+1}{3s} \end{bmatrix}$$

$$10.7 \quad (c) \quad \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} = \begin{bmatrix} 11/4 & 11/6 \\ 11/6 & 11/4 \end{bmatrix}$$

$$10.8 \quad (a) \quad Z_{in} = r_b + \frac{r_e(r_c + Z_L)}{r_e + (1-\alpha)r_c + Z_L}, \quad Z_{out} = \frac{r_c r_e + [r_e + (1-\alpha)r_c](r_b + Z_s)}{r_b + r_e + Z_s}$$

$$(b) \quad Z_{in} = -\frac{Z_1 Z_3}{Z_2 Z_4} Z_L$$

$$10.9 \quad (b) \quad \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = \begin{bmatrix} 1 - 1/(\omega^2 LC) & 1/(j\omega C) \\ 1/(j\omega L) & 1 \end{bmatrix}, \quad C = \frac{L}{R_s R_L}, \quad L = \frac{R_L}{\omega \sqrt{(R_L - R_s)/R_s}}$$

$$10.10 \quad (a) \quad Z_a = \frac{1}{2s} \begin{bmatrix} s^2 + 1 & s^2 - 1 \\ s^2 - 1 & s^2 + 1 \end{bmatrix}$$

$$(b) \quad A_1 = \frac{1}{s^2 - 1} \begin{bmatrix} s^2 + 1 & 2s \\ 2s & s^2 + 1 \end{bmatrix}$$

$$(c) \quad Z = \frac{1}{2s} \begin{bmatrix} s^2 + 2s + 3 & s^2 + 1 \\ s^2 + 1 & s^2 + 2s + 3 \end{bmatrix}$$

$$(d) \quad G(s) = A_i = \frac{-I_2(s)}{I_1(s)} = \frac{s^2 + 1}{s^2 + 4s + 3}$$

$$(i) \quad G(s) = A_i = \frac{2s^5 + 3s^4 + 6s^3 + 4s^2 + 4s + 1}{2s^5 + 7s^4 + 10s^3 + 9s^2 + 6s + 1}$$

$$10.11 \quad (a) \quad \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = -\begin{bmatrix} 0.5000157 & 0.0001501 \\ 0.0005000 & 0.0000005 \end{bmatrix}, \quad Z_i = 0.0212, \quad Z_o = 0.00300,$$

$$A_v = V_o/V_s = -2$$

$$(b) \quad Z_i = 9994, \quad A_v = \frac{V_o}{V_s} = -\frac{4.95s}{s + 100}$$

10.12 (a) $Z_i = 687, Z_o = 17.1, A_v = -152$

(b) $Z_{\text{out}} = \frac{90.195R_{12} + 72\,175}{R_{12} + 12\,781} = 16\,\Omega$

10.13 $\frac{V_o}{I_i} = \frac{(1 - g_m R_f) R_s r_{be} R_C}{r_{be} R_C + r_{be} R_f + r_{be} R_s + R_s R_C + R_s R_f + g_m R_s r_{be} R_C}$

References

- [F-1] Franco, Sergio, *Electric Circuits Fundamentals*, Saunders College Publishing, Philadelphia, Pennsylvania, 1995.
- [H-1] Herrick, Robert J., *DC/AC Circuits and Electronics: Principles and Applications*, Thomson Delmar Learning, Inc., New York, 2003.
- [H-2] Huelsman, L.P., *Basic Circuit Theory*, 3rd edition, Prentice-Hall, Inc., Englewood Clipp, New Jersey, 1998.
- [I-1] Irwin, J. David and Chwan-Hwa Wu, *Basic Engineering Circuit Analysis*, 6th edition, Prentice-Hall, Inc., Englewood Clipp, New Jersey, 1999.
- [K-1] Keown, John, *OrCAD PSpice and Circuit Analysis*, 4th edition, Prentice-Hall, Inc., Englewood Clipp, New Jersey, 2001.
- [K-2] Kreyszig, Erwin, *Advanced Engineering Mathematics*, 8th edition, John Wiley & Sons, Inc., New York, 1999.
- [K-3] Kuc, Roman, *Introduction to Digital Signal Processing*, McGraw-Hill, New York, 1988.
- [N-1] Nilsson, James W. and Susan A. Riedel, *Electric Circuits*, 5th edition, Addison-Wesley Publishing Company, Reading, Massachusetts, 1996.
- [R-1] Rashid, Muhammad H., *Microelectronic Circuits: Analysis and Design*, PWS Publishing Company, Boston, Massachusetts, 1999.
- [R-2] Rashid, Muhammad H., *Introduction to PSpice Using OrCAD for Circuits and Electronics*, 3rd edition, Pearson Education, Inc., New Jersey, 2004.
- [R-3] Reed, Michael L. and Ron A. Rohrer, *Applied Introductory Circuit Analysis for Electrical and Computer Engineers*, Prentice-Hall, Inc., Englewood Clipp, New Jersey, 1999.
- [S-1] Sedra, Adel S. and Kenneth C. Smith, *Microelectronic Circuits*, Saunders College Publishing, Philadelphia, Pennsylvania, 1991.
- [T-1] Thomas, Roland E. and Albert J. Rosa, *The Analysis and Design of Linear Circuits*, Prentice-Hall, Inc., Englewood Clipp, New Jersey, 1994.
- [T-2] Thomas, Roland E. and Albert J. Rosa, *Circuits and Signals: An Introduction to Linear and Interface Circuit*, John Wiley & Sons, Inc., New Jersey, 1984.
- [W-1] Web site (http://en.wikipedia.org/wiki/Main_Page)
- [W-2] Web site (<http://www.mathworks.com/>)
- [W-3] Web site (<http://mathworld.wolfram.com/>)
- [W-4] Web site (<http://www.uoguelph.ca/~antoon/gadgets/555/555.html>)
- [W-5] Web site (<http://inventors.about.com/library/inventors/bledison.htm>)
- [W-6] Web site (<http://whatis.techtarget.com/>)
- [W-7] Web site (http://www.datasheetcatalog.com/datasheets_pdf/U/A/7/4/UA741.shtml)
- [W-8] Web site (<http://www.alldatasheet.com/>)
- [W-9] Web site (<http://www.k-wz.de/physik/threephasegenerator.html>)
- [Y-1] Yang, Won Y., Wenwu Cao, Tae-Sang Chung, and John Morris, *Applied Numerical Methods Using MATLAB*, John Wiley & Sons, Inc., New Jersey, 2005.

Index

- ABCD parameter, 402
- AC (alternating current), 10
- AC admittance, 262
- AC-excited first-order circuit, 136
- AC impedance, 261–262
- AC steady-state response, 260
- AC sweep analysis, 498, 508
- active element, 9
- active filter, 336–340
- active power, 275, 394
- admittance, 41, 120, 262
- admittance parameter, 401
- admittance triangle, 264–265
- all-pass filter, 358
- Ampere's right-hand rule, 225
- analog computer, 169
- analog filter design, 341–354
- analogy, 205
- α -parameter, 402, 438
- apparent power, 277, 394
- arc, 112, 188, 190
- astable, 143
- autotransformer, 241–242
- average power, 275, 394

- balanced, 299–300, 302, 305
- bandpass filter (BPF), 323, 336, 344, 370
- bandstop filter (BSF), 330, 337, 345
- bandwidth, 320, 324, 331
- bias point, 493, 504
- biquad circuit, 362
- bistable multivibrator, 80
- bode(), 357, 366
- Bode diagram, 357
- bouncing, 163
- b -parameter, 402, 438
- Branch, 14, 18
- bridge balance condition, 274
- bridge circuit, 65, 267, 274, 292, 363, 440
- butter(), 342–344, 368
- Butterworth filter, 343, 347

- capacitance, 7–8, 116–117
- capacitance multiplier, 171
- capacitive, 264, 265
- capacitive reactance, 264
- capacitor, 7–8, 113, 116–117, 262
- Capture (CIS) window, 481
- cascade, 343, 369
- cascade connection, 416, 448
- causal, 212
- CCCS (current-controlled current source), 9–10
- CCVS (current-controlled voltage source), 9–10
- center frequency, 323, 324, 330
- characteristic equation, 178, 182
- characteristic root, 178
- charge conservation, 114, 147, 148
- cheby1(), 342, 344
- cheby2(), 342, 345
- Chebyshev filter, 342, 244, 245, 370
- closed-loop gain, 74, 75, 435
- coefficient of coupling, 225, 236
- complex power, 277
- conductance, 5, 36
- continuity rule of capacitor voltage, 114, 124
- continuity rule of inductor current, 111–112, 121, 252
- controlled source, 9
- controlling variable, 9
- convolution, 210, 455
- convolution property, 210, 452, 455
- coupled coils, 225–236, 243–254
- coupling, 225, 236
- CRC circuit, 146
- critically damped, 178, 179, 184, 194
- CtFS_trigonometric(), 380
- current, 2
- current divider, 38, 97
- current gain, 82, 241, 243, 423, 425
- current magnification ratio, 326
- current source, 9
- current transformer, 252
- Cursor, 490

- cutoff frequency, 320, 321, 322
- cutset, 15, 18
- DAC, *see* Digital-to-Analog converter
- damped frequency, 180
- damping constant, 180
- damping ratio, 180
- DC (direct current), 10
- DC path to ground, 485
- DC sensitivity analysis, 504
- DC steady state, 113, 114, 115
- DC sweep analysis, 494
- deactivation (removal) of sources, 21
- debounce, 95, 163
- degenerate circuit, 144
- Δ (delta)-Y (II-T) conversion, 40, 266
- dependent source, 9
- design_combiner(), 104
- destabilization effect of positive feedback, 80
- diff(), 473
- difference amplifier, 91, 102, 103
- differential equation, 459, 469, 472
- differential input voltage, 11
- differentiation property, 122, 184, 452–454
- differentiator, 139, 170
- Digital-to-Analog converter (DAC), 102
- distortion factor, 394
- dot convention, 226
- driving-point impedance, 411
- dsolve(), 175, 472
- dual circuit, 206
- duality, 206
- dy_conversion(), 266
- dynamic resistance, 85–86
- effective value, 256, 393
- electric field energy (of capacitor), 8, 147
- electric potential, 2
- electric power, 3
- electric shock, 310
- electromotive force (emf), 2
- ellip(), 342, 346
- emf, *see* electromotive force
- equivalent, 21–22
- 555 timer/oscillator, 152–155, 173
- Faraday's law, 6, 224
- feedback amplifier, 430–434
- filter, 319
- filter design, 341, 343, 367, 370
- final state, 113, 114, 115
- final value theorem, 452, 457
- first-order circuit, 111
- first-order OP Amp circuit, 138–143
- flux, 6, 225
- flux linkage, 6, 224–225
- flux linkage conservation, 112, 145, 148, 252
- forced response, 124
- Fourier_analysis(), 384–385, 399
- Fourier analysis using Pspice, 381–384, 501
- Fourier series, 373–375
- Fourier series and Laplace transform, 387–390
- free-wheeling diode, 149
- frequency response, 211–212, 263, 319, 338–341, 347, 379
- frequency response scaling, 338
- frequency scaling, 338
- frequency selective circuit, 319
- fsolve(), 157, 162, 174, 175, 468
- full-wave rectified cosine wave, 385
- fundamental frequency, 373
- geometric series, 474
- GFI, *see* ground fault interrupter *g*-parameter, 402, 438
- ground fault interrupter (GFI), 311–313
- ground (node), 40, 484, 485
- grounding, 310–312
- half-power frequency, *see* cutoff frequency
- half-wave rectified cosine wave, 378
- half-wave rectifier, 161
- half-wave (symmetric), 375
- highpass filter (HPF), 321, 335, 336, 346
- highpass notch filter (HPNF), 358
- Howland circuit, 107
- h*-parameter, 402, 438
- hybrid parameter, 402
- hysteresis characteristic, 80
- ideal OP Amp, 12
- ideal source, 10
- ideal transformer, 237–239
- ilaplace(), 187, 234, 459–460
- ilaplace_my(), 249, 446
- immittance parameter, 401
- impedance, 49, 120, 262
- impedance angle, 265
- impedance matching, 283–284, 296, 444, 448
- impedance parameter, 401
- impedance scaling (transformation), 238
- impedance transformation (scaling), 238
- impedance triangle, 263, 265
- impulse function, 453
- impulse response, 207–210
- independent source, 9
- inductance, 6–7, 116, 224
- inductance emulator, 172
- inductive, 264, 265
- inductive reactance, 264
- inductor, 6–7, 111, 115–116, 261–262

- initial state, 113, 114, 115
- initial transient bias point, 493
- initial value theorem, 452, 456
- in phase, 265, 323
- input impedance, 11, 77, 82, 422, 424, 425
- input resistance, 82, 100
- input resistance of OP Amp, 12
- instantaneous power, 275, 303
- int(), 473
- integration property, 452, 454
- integrator, 138–139, 169, 170, 219
- inverse Laplace transform, 457–460
- inverse matrix, 462–463
- inverse phasor transform, 259
- inverting OP Amp circuit, 72–73
- inverting positive-feedback OP Amp, 78–79

- jacob(), 469

- KCL (Kirchhoff's current law), 15–16, 27–30, 40, 75–76
- KCL equation, 18
- Kerwin-Huelsman-Newcomb (KHN) circuit, 362
- KVL (Kirchhoff's voltage law) 16–17, 27–30, 48
- KVL equation, 18

- ladder network, 27, 87
- lagging PF, 278
- Laplace transform, 118, 260, 390, 451–460
- Laplace transform table, 452
- LC tank, 327
- leading PF, 278
- Lenz's law, 224
- limit on output current of OP Amp, 109
- limit on output voltage of OP Amp, 109
- linear, 5, 8, 71, 208
- linear region, 11
- linear time-invariant, 208–209
- linear transformer, 240
- line voltage, 301
- load current controller, 107
- load flow, 285
- loading effect, 38, 77, 81–82
- load line analysis, 82–86, 110
- loop, 14–18
- loop analysis, *see* mesh analysis
- lowpass filter (LPF), 320, 334, 336
- lowpass notch filter (LPNF), 358
- LRL circuit, 144–145

- magnetically coupled, 225–236, 243–254
- magnetic field energy (of inductor), 7, 145
- magnetic reluctance, 7, 223
- magnetomotive force (mmf), 6, 223
- magnitude scaling, 338
- marginally stable, 181
- Marker, 486
- maximum output voltage of OP Amp, 11, 78–80
- maximum power transfer, 283–284, 296
- mesh, 14, 17
- mesh (loop) analysis, 48–56
- mesh equation, 49
- mesh impedance matrix, 49
- MFB (multiple feedback) circuit, 335–337, 353
- Millman's theorem, 98
- missing DC path to ground, 504
- mmf, *see* magnetomotive force
- multi-scale ammeter, 96
- multi-scale voltmeter, 96
- mutual inductance, 225–230, 246

- natural frequency, 125
- natural response, 124–125
- negative feedback, 12–13, 73–75, 138–143
- negative resistance, 108
- netlist file, 493
- neutral, 301
- neutrally stable, 181
- newtons(), 157, 469
- node, 14, 18
- node admittance matrix, 41
- node analysis, 40–48, 56
- node equation, 41
- non-inverting OP Amp circuit, 74–75
- non-inverting positive-feedback OP Amp, 79–80
- nonlinear equation, 468
- nonlinear resistor circuit, 82–86, 110
- nonlinear RL circuit, 175
- Norton equivalent, 63–64
- notch frequency, 330

- OP Amp (operational amplifier), 10–12, 77–78
- OP Amp circuit, 72–80, 91–94, 138–143, 168–174, 204, 333–340
- open-loop gain, 11–12, 435
- operating point, 83–86, 175, 493
- output impedance, 11, 77, 78, 82, 424, 429
- output resistance, 82, 100
- output resistance of OP Amp, 12
- overdamped, 178, 183

- parallel_comb(), 36, 266
- parallel, 25, 116, 117, 343, 369, 414, 432
- parallel(-parallel) connection, 414, 432
- parallel combination of capacitors, 117
- parallel combination of inductors, 116
- parallel combination of resistors, 36–37, 95
- parallel duplication of voltage source, 19, 22
- parallel resonance, 326, 327
- parallel RLC circuit, 192, 197, 215, 264, 326, 332

- parallel-series, 415, 433
- PARAMETERS, 287–288, 500–501
- parameter conversion, 406–410
- Parametric Sweep, 287–288, 500–501
- Passband, 324, 341
- passive element, 5
- passive sign convention, 4
- peak frequency, 323, 326
- periodic switching, 165
- permeability, 7, 223
- permeance, 6–7, 223, 237
- PF_correction(), 282
- PF (power factor) correction, 279–282, 295
- phase, 256, 466
- phase lag, 265
- phase lead, 265
- phase voltage, 301
- phasor, 256
- phasor diagram, 263–264, 280, 293, 301, 303
- phasor method, *see* phasor transform
- phasor transform, 259–260
- $\Pi(\pi)$ -equivalent, 229
- port condition, 401, 416, 444
- port_conversion(), 410, 419, 439, 449
- port current requirement, *see* port condition
- port_property(), 427, 439, 449
- positive feedback, 13, 78–80, 95, 140–143
- potential, 2
- power, 3, 228, 275–278, 303, 394
- power conservation, 277
- power factor (PF), 278–286, 302–303, 394
- power factor (PF) angle, 265, 278
- power transmission, 240, 317
- power triangle, 277–278
- practical analysis rule of OP Amp circuit, 76
- practical source, 10
- Property Editor spreadsheet, 94, 152, 237, 288, 484, 497, 500
- pulse response, 208
- quality factor, 324, 326, 331
- RC circuit, 123, 134, 150, 156, 160, 164–167, 381
- RC OP Amp circuit, 139, 169, 173
- reactance, 262, 263
- reactive power, 276
- real power, 275
- reciprocal, 412
- rectangular (or square) wave, 142, 376, 381
- rectangular wave generator, 140–143, 167–168, 174
- reference direction, 4
- reference node, 40
- reference polarity, 3
- reflected impedance, 239, 297
- rejection frequency, 330
- relative winding direction, 226, 230, 235
- relaxation oscillator, 143
- relay, 159
- removal (deactivation) of sources, 21
- residue(), 458–460
- resistance, 5, 35
- resistivity, 5, 25
- resistor, 5, 119, 261
- resonance, 325, 326, 329
- resonance condition, 329
- resonant frequency, 323, 326
- RL circuit, 121, 134, 152, 156, 159, 166, 173, 176
- rms (root-mean-square), 256, 393
- Sallen-Key circuit, 334, 353
- saturation output voltage of OP Amp, 11, 78, 108, 220, 291
- saturation (nonlinear) region, 11
- saw-tooth function, 395
- Schmitt trigger, 80, 94–95, 163
- s -domain (transformed) equivalent, 119–120
- second-order active filter, 334–338
- second-order circuit, 177
- second-order OP Amp circuit, 204, 213, 222
- selectivity, 324, 331
- self inductance, 224
- sequential switching, 133, 166–167
- series, 25, 116, 117, 414
- series combination of capacitors, 117
- series combination of inductors, 116
- series combination of resistors, 36–37, 95
- series duplication of current source, 20, 23
- series-parallel, 415, 431
- series resonance, 325
- series RLC circuit, 181, 187, 214, 263, 323, 330
- series(-series), 414, 432
- simulation profile, 486–487
- Simulation Settings dialog box, 237, 288, 487–488
- singular circuit, 144
- SKIPBP, 236, 248, 250
- solve(), 472
- source transformation, 21–24, 31
- source transformation method, 43, 45, 47, 52, 55, 56
- square wave, *see* rectangular wave
- stable, 180–181, 212
- stabilization effect of negative feedback, 12
- standard values of capacitors, 477
- standard values of inductors, 478
- standard values of resistors, 476
- state equation, 470
- static resistance, 85–86
- steady-state response, 125, 211–212
- step function, 451
- step response, 327
- stopband, 331, 341

- summing amplifier, 91
- supermesh, 14, 17
- supermesh method, 50, 51, 54
- supernode, 14, 16
- supernode method, 42, 44, 46, 59, 61
- superposition principle, 71, 208
- susceptance, 262, 264
- symbolic computation, 471
- symmetrical, 412

- 3dB frequency, 320, 326, 331
- Tellegen's theorem, 28
- T-equivalent, 229, 245, 248, 250
- tf2par_s(), 349
- tf2sos(), 344
- THD, *see* total harmonic distortion
- Thevenin equivalent, 63–70, 99–102, 203, 239, 270, 273, 424
- three-phase power, 303–308
- time constant, 125–128, 180, 321
- time differentiation property, 452, 454
- time-invariant, 208
- time shifting property, 452, 455
- total harmonic distortion (THD), 394
- Tow-Thomas circuit, 362
- Trace, 490
- transfer function, 178, 205, 207, 210
- transfer function analysis, 504
- transfer impedance, 411
- transformed (*s*-domain) equivalent, 119–120
- transformer, 237–243, 246, 284, 294
- transient response, 125
- transmission loss, 280
- transmission parameter, 402
- tree, 18
- triangular wave, 376, 399
- triangular wave generator, 140–143, 167–168
- trigonometric formulas, 474
- tuning, 339
- two-port network, 401, 420
- two-port network property, 427

- undamped, 186, 195
- undamped resonant frequency, 180, 181

- underdamped, 179, 185
- unit impulse function, 453
- units, 1
- unit step function, 451
- unbalanced three-phase power system, 314
- unstable, 143, 180, 181

- VA (Volt-Ampere), 277
- VAC (AC voltage source), 483
- VAR (Volt-Ampere Reactive), 277
- VCCS (voltage-controlled current source), 9–10
- VCR (voltage-current relationship), 41, 48
- VCVS (voltage-controlled voltage source), 9–10
- virtual ground, 73
- virtual open principle, 13, 76
- virtual short principle, 12–13, 76
- voltage, 2
- voltage divider, 37, 88, 97
- voltage follower, 76–77
- voltage gain, 81, 241, 243, 423, 425
- voltage magnification ratio, 324
- voltage source, 9
- voltage-to-current converter, 107
- VPULSE, 382, 484
- VPWL, 484
- VSIN, 483, 484

- War of Currents, 223
- wC_for_PF_correction(), 282
- Wien bridge oscillator, 221
- window defroster, 90

- y-parameter, 401, 438
- yd_conversion(), 266
- Y- Δ (T-II) conversion, 40, 266
- Y- Δ /Y connection, 307
- y_d(), 316
- y_dy(), 307, 314
- Y-Y connection, 304
- y_y(), 305, 314

- zero-input response, 124
- zero-state response, 124
- z-parameter, 401, 438