Microelectronic Circuits
Analysis and Design
Second Edition

Muhammad H. Rashid
University of West Florida
To my parents,
my wife, Fatema,
my children, Faeza, Farzana, and Hasan
CONTENTS

Preface xiii
Teaching Plans and Suggested Course Outlines xvii
About the Author xix

Introduction to Electronics and Design

1.1 Introduction 2
1.2 History of Electronics 2
1.3 Electronic Systems 4
1.4 Electronic Signals and Notation 6
1.5 Classifications of Electronic Systems 10
1.6 Specifications of Electronic Systems 12
1.7 Types of Amplifiers 15
1.8 Design of Electronic Systems 17
1.9 Design of Electronic Circuits 20
1.10 Electronic Devices 27
1.11 Emerging Electronics 32
References 36
Problems 37

Introduction to Amplifiers and Frequency Response

2.1 Introduction 40
2.2 Amplifier Characteristics 40
2.3 Amplifier Types 50
2.4 Cascaded Amplifiers 59
2.5 Frequency Response of Amplifiers 62
2.6 Miller’s Theorem 71
2.7 Frequency Response Methods 72
2.8 PSpice/SPICE Amplifier Models 87
2.9 Amplifier Design 88
Summary 91
References 92
Review Questions 92
Problems 93

Introduction to Operational Amplifiers and Applications

3.1 Introduction 104
3.2 Characteristics of Ideal Op-Ampl 104
## Contents

3.3 Op-Amp PSpice/SPICE Models 111  
3.4 Analysis of Ideal Op-Amp Circuits 114  
3.5 Op-Amp Applications 128  
3.6 Op-Amp Circuit Design 164  
Summary 165  
References 166  
Review Questions 166  
Problems 167  

### Semiconductor Diodes

4.1 Introduction 180  
4.2 Ideal Diodes 180  
4.3 Transfer Characteristics of Diode Circuits 183  
4.4 Practical Diodes 185  
4.5 Analysis of Practical Diode Circuits 192  
4.6 Modeling of Practical Diodes 196  
4.7 Zener Diodes 208  
4.8 Light-Emitting Diodes 220  
4.9 Power Rating 220  
4.10 Diode Data Sheets 222  
Summary 226  
References 226  
Review Questions 226  
Problems 227  

### Applications of Diodes

5.1 Introduction 238  
5.2 Diode Rectifier 238  
5.3 Output Filters for Rectifiers 260  
5.4 Diode Peak Detectors and Demodulators 272  
5.5 Diode Clippers 276  
5.6 Diode Clamping Circuits 279  
5.7 Diode Voltage Multipliers 284  
5.8 Diode Function Generators 287  
Summary 290  
References 291  
Review Questions 291  
Problems 291  

### Semiconductors and pn Junction Characteristics

6.1 Introduction 300  
6.2 Semiconductor Materials 300
6.3 Zero-Biased \( pn \) Junction 307
6.4 Reverse-Biased \( pn \) Junction 314
6.5 Forward-Biased \( pn \) Junction 319
6.6 Junction Current Density 323
6.7 Temperature Dependence 325
6.8 High-Frequency AC Model 326
Summary 329
References 330
Review Questions 330
Problems 331

Metal Oxide Semiconductor Field-Effect Transistors

7.1 Introduction 336
7.2 Metal Oxide Field-Effect Transistors 336
7.3 Enhancement MOSFETs 337
7.4 Depletion MOSFETs 346
7.5 MOSFET Models and Amplifier 349
7.6 A MOSFET Switch 356
7.7 DC Biasing of MOSFETs 357
7.8 Common-Source (CS) Amplifiers 364
7.9 Common-Drain Amplifiers 375
7.10 Common-Gate Amplifiers 380
7.11 Multistage Amplifiers 383
7.12 DC Level Shifting and Amplifier 386
7.13 Frequency Response of MOSFET Amplifiers 393
7.14 Design of MOSFET Amplifiers 408
Summary 413
References 413
Review Questions 414
Problems 414

Bipolar Junction Transistors and Amplifiers

8.1 Introduction 434
8.2 Bipolar Junction Transistors 434
8.3 Principles of BJT Operation 436
8.4 Input and Output Characteristics 447
8.5 BJT Circuit Models 449
8.6 The BJT Switch 455
8.7 DC Biasing of Bipolar Junction Transistors 457
8.8 Common-Emitter Amplifiers 467
8.9 Emitter Followers 476
8.10 Common-Base Amplifiers 483
8.11 Multistage Amplifiers 488
Contents

8.12 The Darlington Pair Transistor 491
8.13 DC Level Shifting and Amplifier 495
8.14 Frequency Model and Response of Bipolar Junction Transistors 501
8.15 Frequency Response of BJT Amplifiers 508
8.16 MOSFETs versus BJTs 528
8.17 Design of Amplifiers 528
Summary 533
References 533
Review Questions 533
Problems 534

Differential Amplifiers

9.1 Introduction 554
9.2 Internal Structure of Differential Amplifiers 554
9.3 MOSFET Current Sources 558
9.4 MOS Differential Amplifiers 566
9.5 Depletion MOS Differential Amplifiers 580
9.6 BJT Current Sources 586
9.7 BJT Differential Amplifiers 602
9.8 BiCMOS Differential Amplifiers 620
9.9 Frequency Response of Differential Amplifiers 626
9.10 Design of Differential Amplifiers 628
Summary 629
References 629
Review Questions 629
Problems 630

Feedback Amplifiers

10.1 Introduction 642
10.2 Feedback 643
10.3 Characteristics of Feedback 644
10.4 Feedback Topologies 652
10.5 Analysis of Feedback Amplifiers 656
10.6 Series-Shunt Feedback 657
10.7 Series-Series Feedback 667
10.8 Shunt-Shunt Feedback 677
10.9 Shunt-Series Feedback 686
10.10 Feedback Circuit Design 692
10.11 Stability Analysis 698
10.12 Compensation Techniques 711
Summary 721
References 721
Review Questions 722
Problems 722
Power Amplifiers
11.1 Introduction 740
11.2 Classification of Power Amplifiers 740
11.3 Power Transistors 743
11.4 Class A Amplifiers 745
11.5 Class B Push-Pull Amplifiers 756
11.6 Complementary Class AB Push-Pull Amplifiers 766
11.7 Class C Amplifiers 777
11.8 Class D Amplifiers 781
11.9 Class E Amplifiers 784
11.10 Short-Circuit and Thermal Protection 786
11.11 Power Op-Amps 788
11.12 Thermal Considerations 792
11.13 Design of Power Amplifiers 796
Summary 797
References 797
Review Questions 797
Problems 798

Active Filters
12.1 Introduction 804
12.2 Active versus Passive Filters 804
12.3 Types of Active Filters 805
12.4 First-Order Filters 808
12.5 The Biquadratic Function 810
12.6 Butterworth Filters 814
12.7 Transfer Function Realization 818
12.8 Low-Pass Filters 819
12.9 High-Pass Filters 829
12.10 Band-Pass Filters 837
12.11 Band-Reject Filters 843
12.12 All-Pass Filters 848
12.13 Switched-Capacitor Filters 849
12.14 Filter Design Guidelines 854
Summary 855
References 855
Review Questions 855
Problems 856

Oscillators
13.1 Introduction 862
13.2 Principles of Oscillators 862
13.3 Audio-Frequency Oscillators 867
Operational Amplifiers

14.1 Introduction 910
14.2 Internal Structure of Op-Amps 910
14.3 Parameters and Characteristics of Practical Op-Amps 911
14.4 CMOS Op-Amps 933
14.5 BJT Op-Amps 940
14.6 Analysis of the LM741 Op-Amp 944
14.7 BiCMOS Op-Amps 962
14.8 Design of Op-Amps 974
Summary 975
References 976
Review Questions 976
Problems 977

Introduction to Digital Electronics

15.1 Introduction 982
15.2 Logic States 982
15.3 Logic Gates 983
15.4 Performance Parameters of Logic Gates 985
15.5 NMOS Inverters 996
15.6 NMOS Logic Circuits 1014
15.7 CMOS Inverters 1016
15.8 CMOS Logic Circuits 1022
15.9 Comparison of CMOS and NMOS Gates 1026
15.10 BJT Inverters 1026
15.11 Transistor-Transistor Logic Gates 1033
15.12 Emitter-Coupled Logic OR/NOR Gates 1049
15.13 BiCMOS Inverters 1057
15.14 Interfacing of Logic Gates 1060
15.15 Comparison of Logic Gates 1063
15.16 Design of Logic Circuits 1064
Summary 1068
References 1068
Review Questions 1068
Problems 1069
Integrated Analog Circuits and Applications

16.1 Introduction 1080
16.2 Circuits with Op-Amps and Diodes 1080
16.3 Comparators 1097
16.4 Zero-Crossing Detectors 1100
16.5 Schmitt Triggers 1101
16.6 Square-Wave Generators 1110
16.7 Triangular-Wave Generators 1113
16.8 Sawtooth-Wave Generators 1117
16.9 Voltage-Controlled Oscillators 1120
16.10 The 555 Timer 1126
16.11 Phase-Lock Loops 1139
16.12 Voltage-to-Frequency and Frequency-to-Voltage Converters 1147
16.13 Sample-and-Hold Circuits 1155
16.14 Digital-to-Analog Converters 1158
16.15 Analog-to-Digital Converters 1165
16.16 Circuit Design Using Analog Integrated Circuits 1169
Summary 1170
References 1170
Review Questions 1170
Problems 1171

Appendix A Introduction to OrCAD 1177
Appendix B Review of Basic Circuits 1213
Appendix C Low-Frequency Hybrid BJT Model 1261
Appendix D Ebers–Moll Model of Bipolar Junction Transistors 1267
Appendix E Passive Components 1275
Appendix F Design Problems 1281
Answer to Selected Problems A1
Index I1
Semiconductor devices and integrated circuits (ICs) are the backbone of modern technology, and thus the study of electronics—which deals with their characteristics and applications—is an integral part of the undergraduate curriculum for students majoring in electrical, electronics, or computer engineering. Traditionally, the basic course in electronics has been a one-year (two-semester) course at most universities and colleges. However, with the emergence of new technologies and university-wide general education requirements, electrical engineering departments are under pressure to reduce basic electronics to a one-semester course. This book can be used for a one-semester course as well as a two-semester course. The only prerequisite is a course in basic circuit analysis. A one-semester course would cover Chapters 1 through 8, in which the basic techniques for analyzing electronic circuits are introduced using ICs as examples. In a two-semester course, the second semester would focus on detailed analysis of devices and circuits within the ICs and their applications.

The objectives of this book are:

- To develop an understanding of the characteristics of semiconductor devices and commonly used ICs
- To develop skills in analysis and design of both analog and digital circuits
- To introduce students to the various elements of the engineering design process, including formulation of specifications, analysis of alternative solutions, synthesis, decision-making, iterations, consideration of cost factors, simulation, and tolerance issues

**Approach**

This book adopts a top-down approach to the study of electronics, rather than the traditional bottom-up approach. In the classical bottom-up approach, the characteristics of semiconductor devices and ICs are studied first, and then the applications of ICs are introduced. Such an approach generally requires a year of instruction, as it is necessary to cover all the essential materials in order to give students an overall knowledge of electronic circuits and systems. In the top-down approach used here, the ideal characteristics of IC packages are introduced to establish the design and analytical techniques, and then the characteristics and operation of devices and circuits within the ICs are studied to understand the imperfections and limitations of IC packages. This approach has the advantage of allowing the instructor to cover only the basic techniques and circuits in the first semester, without going into detail on discrete devices. If the curriculum allows, the course can continue in the second semester with detailed analysis of discrete devices and their applications.

In practice, the lectures and laboratory experiments run concurrently. If students’ experimental results differ from the ideal characteristics because of the practical limitations of IC packages, students may become concerned. This concern may be addressed by a brief explanation of the causes of discrepancies. The experimental results, however, will not differ significantly from the theoretically obtained results.

Current ABET (Accreditation Board of Engineering and Technology) criteria and other engineering criteria under the Washington Accord (http://www.washingtonaccord.org/) require the integration of design and computer usage throughout the curriculum. After students have satisfied other ABET and
accreditation requirements in math, basic science, engineering science, general education electives, and free electives, they find that not many courses are available to satisfy the design requirements. The lack of opportunities for design credits in engineering curricula is a common concern. Electronics is generally the first electrical engineering course well suited to the integration of design components and computer usage. This book is structured to permit design content to constitute at least 50% of the course, and it integrates computer usage through PSpice. Many design examples use PSpice to verify the design requirements, and the numerous computer-aided design examples illustrate the usefulness of personal computers as design tools, especially in cases in which design variables are subjected to component tolerances and variations.

New to This Edition

The second edition offers a reorganized order of chapters with the required material augmented and the nonessential topics abridged. The key changes to this edition are summarized below:

- All new chapter on MOSFETs and amplifiers
- All new chapter on semiconductors and pn junctions
- Fully revised chapter on BJTs
- More emphasis on MOSFETs and active biasing techniques to allow students to move easily on to differential amplifiers and ICs
- Extensive revision of power amplifiers to include MOSFET circuits with class C, D, and E amplifiers
- Integrated PSpice/OrCAD examples for both analysis and design verifications
- Developed Mathcad files for calculations of worked-out examples so that students can try similar problems and explore the effects of design parameters

Content and Organization

After an introduction to the design process in Chapter 1, the book may be divided into six parts:

I. Chapters 2 and 3 on characteristics of amplifiers and their frequency responses
II. Chapters 4 and 5 on diodes and applications
III. Chapters 6 to 8 and 11 on semiconductor fundamentals, transistors, and amplifiers
IV. Chapters 10, 12, and 13 on characteristics and analyses of electronic circuits
V. Chapter 15 on digital logic gates
VI. Chapters 9, 14, and 16 on integrated circuits and applications

A review of basic circuit analysis and an introduction to PSpice are included in the appendices.

Modern semiconductor technology has evolved to such an extent that many analog and digital circuits are available in the form of integrated circuit (IC) packages. Manufacturers of these packages provide application notes that can be used to implement circuit functions. Knowledge of the characteristics and operation of devices within the IC packages is essential, however, to understand the limitations of these ICs when they are interfaced as building blocks in circuit designs. Such knowledge also serves as the basis for developing future generations of IC packages. Although the trend in IC technology suggests that discrete circuit design may disappear entirely in the future, transistor amplifiers (in large-scale or...
very-large-scale integrated forms) will continue to be the building blocks of ICs. Thus, semiconductor
fundamentals and transistor amplifiers are covered in Chapters 6 to 8, after the general types and spec-
cifications of amplifiers have been introduced in Chapter 2. Because diodes are the building blocks of
many electronic circuits, and because the techniques for the analysis of diodes are similar to those for
transistor amplifiers, diodes and their applications are addressed in detail in Chapters 4 and 5.

Pedagogy and Supplements

The pedagogical approach of the first edition has been enhanced and augmented in this edition. Mathe-
matical derivations are kept to a minimum by using approximate circuit models of operational amplifiers,
transistors, and diodes. The significance of these approximations is established by computer-aided analy-
sis using PSpice. Important circuits are analyzed in worked-out examples in order to introduce the basic
techniques and emphasize the effects of parameter variations. At the end of each chapter, review questions
and problems test students’ learning of the concepts developed in the chapter. The student learning out-
comes (SLOs) are listed at the beginning of each chapter. Symbols and their meanings have been uniquely
identified at the beginning of each chapter to serve as a quick reference to the students. Every chapter
opens with an introduction that puts the content of the chapter in perspective of the field of microelec-
tronics. Solved examples carry captions that identify the objective of the example. Notes interspersed
through the text provide a link to other chapters and serve to guide students against common misconcep-
tions and mistakes. Key points of most of the sections are summarized in a box in addition to an end-of-
chapter summary. A list of references is included at the end of each chapter for those interested in further
reading. End-of-chapter exercises are divided into Review Questions and Problems. Design problems and
PSpice problems are identified by relevant symbols.

Student support from Cengage Learning is available on the book’s student website www.cengage.com/
electronics/rashid. This website contains tools that are designed to help the student learn about elec-
tronics more effectively. It includes electronic copies of all the PSpice schematics printed in this book,
and Mathcad files for all worked-out examples in the book, which can be downloaded and allow students
to work their own problems.

The student version PSpice schematics and/or OrCAD capture software can be obtained or down-
loaded from:

Cadence Design Systems, Inc.
2655 Seely Avenue
San Jose, CA 95134, USA
Websites: http://www.cadence.com
http://www.orcad.com
http://www.ema-eda.com

Support for Instructors

A solutions manual (in both print and electronic forms) and slides of the figures in this book are available on
request from Cengage Learning through the Global Engineering website www.cengage.com/engineering.

Teaching plans and suggested course outlines for one- and two-semester courses using this book are
included just after this preface.
Acknowledgments

Thanks are due to the editorial team at Cengage Learning, Chris Carson, Chris Shortt, Hilda Gowans, Swati Meherishi, and Yumnam Ojen Singh for their guidance and support.

I would also like to thank the following reviewers for their comments and suggestions on the first and the second editions:

Dr. Ezzat G. Bakhoum  
*University of West Florida*

Dr. William T. Baumann  
*Virginia Polytechnic Institute and State University*

Dr. Paul J. Benkeser  
*Georgia Institute of Technology*

Dr. Alok K. Berry  
*George Mason University*

Dr. Michael A. Bridgwood  
*Clemson University*

Dr. Nadeem N. Bunni  
*Clarkson University*

Dr. Wai-Kai Chen  
*University of Illinois at Chicago*

Dr. Shirshak K. Dhali  
*Southern Illinois University*

Dr. Constantine Hatziadoniu  
*Southern Illinois University*

Dr. Bruce P. Johnson  
*University of Nevada-Reno*

Dr. Frank Kornbaum  
*South Dakota State University*

Dr. Oguz Kucur  
*Gebze Institute of Technology, Turkey*

Dr. John A. McNeill  
*Worcester Polytechnic Institute*

Dr. Bahram Nabet  
*Drexel University*

Dr. Hemanshu R. Pota  
*Australian Defense Force Academy*

Dr. Jack R. Smith  
*University of Florida*

Dr. Robert D. Stratman  
*University of Tulsa*

Finally, thanks to my family for their patience while I was occupied with this and other projects.

Any comments and suggestions regarding this book are welcome. They should be sent to the author at mrashidfl@gmail.com.

Muhammad H. Rashid  
Web: http://uwf.edu/mrashid
As with any comprehensive microelectronics textbook, this text has more material than can be covered in two single-semester courses. Instructors are often lost on what topics to cover in two semesters of 16 weeks each.

The book covers diodes after op-amp circuits so that the complete coverage of op-amp circuits including nonlinear circuits cannot be included in the same chapter. However, if nonlinear op-amp circuits are not to be covered in the course, then the op-amp circuits can be covered at the beginning, after Chapter 2. Most of the materials in Chapter 2 on introduction to amplifiers and in Chapter 5 on applications of diodes can, however, be skipped in a first course. Some approaches to typical first and second electronics courses are delineated below.

**First Electronics Course**

This course usually covers (a) characteristics and models of amplifiers and their frequency responses; (b) IC op-amps and their applications; (c) physical operation, characteristics, and modeling of diodes, which form the basis for understanding small-signal operation and modeling of transistors; (d) the operation, characteristics, modeling, and biasing of transistors; (e) the fundamentals of active sources and differential amplifiers, which are generally used in IC amplifiers; and (f) understanding of frequency responses of electronic circuits. These can be covered by one of the following two approaches. The suggested sequences of course topics are shown in Tables 1 and 2, respectively.

**TABLE 1  Suggested topics for first electronics course—Approach A**

<table>
<thead>
<tr>
<th>Number of Weeks</th>
<th>Topics</th>
<th>Chapters</th>
<th>Sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Introduction to Electronics and Design</td>
<td>1</td>
<td>1.3–1.9</td>
</tr>
<tr>
<td>1</td>
<td>Introduction to Amplifiers and Frequency Responses</td>
<td>2</td>
<td>2.1–2.7</td>
</tr>
<tr>
<td>2</td>
<td>Introduction to Op-Amps</td>
<td>3</td>
<td>3.1–3.4, 3.5.1–5.6</td>
</tr>
<tr>
<td>2</td>
<td>Diodes</td>
<td>4</td>
<td>4.1–4.7</td>
</tr>
<tr>
<td>1</td>
<td>Applications of Diodes</td>
<td>5</td>
<td>3.1–3.3</td>
</tr>
<tr>
<td>1</td>
<td>Semiconductors and \textit{pn} Junctions</td>
<td>6</td>
<td>6.1–6.4</td>
</tr>
<tr>
<td>3</td>
<td>MOSFETs and Amplifiers</td>
<td>7</td>
<td>7.1–7.9</td>
</tr>
<tr>
<td>3</td>
<td>BJTs and Amplifiers</td>
<td>8</td>
<td>8.1–8.9</td>
</tr>
<tr>
<td>1</td>
<td>Differential Amplifiers</td>
<td>9</td>
<td>9.1–9.5</td>
</tr>
<tr>
<td>1</td>
<td>Exams</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
TABLE 2  Suggested topics for first electronics course—Approach B

<table>
<thead>
<tr>
<th>Number of Weeks</th>
<th>Topics</th>
<th>Chapters</th>
<th>Sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Introduction to Electronics and Design</td>
<td>1</td>
<td>1.3–1.9</td>
</tr>
<tr>
<td>2</td>
<td>Diodes</td>
<td>4</td>
<td>4.1–4.7</td>
</tr>
<tr>
<td>1</td>
<td>Applications of Diodes</td>
<td>5</td>
<td>3.1–3.3</td>
</tr>
<tr>
<td>1</td>
<td>Semiconductors and pn Junctions</td>
<td>6</td>
<td>6.1–6.4</td>
</tr>
<tr>
<td>3</td>
<td>MOSFETs and Amplifiers</td>
<td>7</td>
<td>7.1–7.9</td>
</tr>
<tr>
<td>3</td>
<td>BJTs and Amplifiers</td>
<td>8</td>
<td>8.1–8.9</td>
</tr>
<tr>
<td>1</td>
<td>Introduction to Amplifiers and</td>
<td>2</td>
<td>2.1–2.7</td>
</tr>
<tr>
<td></td>
<td>Frequency Responses</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Introduction to Op-Amps</td>
<td>3</td>
<td>3.1–3.4, 3.5.1–3.5.6</td>
</tr>
<tr>
<td>1</td>
<td>Differential Amplifiers</td>
<td>9</td>
<td>9.1–9.5</td>
</tr>
<tr>
<td>1</td>
<td>Exams</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Approach A: Op-amps are covered before diodes in which the course is not expected to cover nonlinear op-amp circuits (using diodes). Since op-amps are the building blocks of many electronic circuits, the analyses of simple op-amp circuits are often covered in the first Basic Circuit Analysis course, which is generally a prerequisite for the electronics course. This approach has the advantage of continuity with the circuits course and is more of a systems-based approach. This approach may be viewed as a top-down approach.

Approach B: Op-amps are covered after diodes, so that students can work on nonlinear op-amp circuits (using diodes) as design projects. This has the advantage of logical progression from the devices (diodes and transistors) to op-amp amplifiers.

Second Electronics Course

This course covers the characteristics and applications of amplifiers. The course usually covers (a) the frequency response of amplifiers; (b) introduction to active filters; (c) feedback amplifiers; (d) oscillators; (e) differential amplifiers with active current sources; (f) power amplifies; (g) op-amps; and (h) IC applications. The sequence of course topics is shown in Table 3.

TABLE 3  Suggested topics for second electronics course

<table>
<thead>
<tr>
<th>Number of Weeks</th>
<th>Topics</th>
<th>Chapters</th>
<th>Sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Frequency Response of Amplifiers</td>
<td>3, 7, 8</td>
<td>2.7, 7.13, 8.15</td>
</tr>
<tr>
<td>1</td>
<td>Differential Amplifiers</td>
<td>9</td>
<td>9.1–9.5</td>
</tr>
<tr>
<td>2</td>
<td>Feedback Amplifiers</td>
<td>10</td>
<td>10.1–10.8, 10.14</td>
</tr>
<tr>
<td>2</td>
<td>Power Amplifiers</td>
<td>11</td>
<td>11.1–11.9</td>
</tr>
<tr>
<td>2</td>
<td>Active Filters</td>
<td>12</td>
<td>12.1–12.9, 12.14</td>
</tr>
<tr>
<td>2</td>
<td>Oscillators</td>
<td>13</td>
<td>13.1–13.7</td>
</tr>
<tr>
<td>2</td>
<td>Introduction to Digital Electronics</td>
<td>15</td>
<td>15.1–15.4, 15.7–15.8</td>
</tr>
<tr>
<td>1</td>
<td>Operational Amplifiers</td>
<td>14</td>
<td>14.1–14.4</td>
</tr>
<tr>
<td>2</td>
<td>IC Applications</td>
<td>16</td>
<td>16.1, 16.2, 16.5–6.8</td>
</tr>
<tr>
<td>1</td>
<td>Exams</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Muhammad H. Rashid is currently Professor (and past Director from 1997 to 2007) of Electrical and Computer Engineering at the University of West Florida. Dr. Rashid received his B.Sc. degree in Electrical Engineering from Bangladesh University of Engineering and Technology, and M.Sc. and Ph.D. degrees from the University of Birmingham in the UK. Previously, he worked as Professor of Electrical Engineering and the Chair of the Engineering Department at Indiana University-Purdue University at Fort Wayne. He has also served as Visiting Assistant Professor of Electrical Engineering at the University of Connecticut, Associate Professor of Electrical Engineering at Concordia University (Montreal, Canada), Professor of Electrical Engineering at Purdue University Calumet, Visiting Professor of Electrical Engineering at King Fahd University of Petroleum and Minerals (Saudi Arabia), design and development engineer with Brush Electrical Machines Ltd. (England, UK), Research Engineer with Lucas Group Research Centre (England), and Lecturer and Head of Control Engineering Department at the Higher Institute of Electronics (Malta).

Dr. Rashid is actively involved in teaching, researching, and lecturing, especially in the area of power electronics. He has published 16 books and more than 130 technical papers. His books are adopted as textbooks all over the world. His books have been translated into several world languages, including Spanish, Portuguese, Indonesian, Korean, and Persian.

Dr. Rashid was a registered Professional Engineer in the Province of Ontario (Canada), and a registered Chartered Engineer (UK). He is a Fellow of the Institution of Electrical Engineers (IEE, UK) and a Fellow of the Institute of Electrical and Electronics Engineers (IEEE, USA). He was elected as an IEEE Fellow with the citation “Leadership in power electronics education and contributions to the analysis and design methodologies of solid-state power converters.” Dr. Rashid is the recipient of the 1991 Outstanding Engineer Award from the IEEE. He received the 2002 IEEE Educational Activity Board (EAB) Meritorious Achievement Award in Continuing Education with the following citation “For contributions to the design and delivery of continuing education in power electronics and computer-aided simulation.” He is the recipient of the 2008 IEEE Undergraduate Teaching Award with the citation “For his distinguished leadership and dedication to quality undergraduate electrical engineering education, motivating students and publication of outstanding textbooks.”

Dr. Rashid was an ABET program evaluator for electrical engineering from 1995 to 2000 and an engineering evaluator for the Southern Association of Colleges and Schools (SACS, USA). He has been elected as an IEEE Industry Applications Society (IAS) Distinguished Lecturer and Speaker. He is the Series Editors of *Power Electronics and Applications* and *Nanotechnology and Applications* with the CRC Press. He serves as the Editorial Advisor of *Electric Power and Energy* with Elsevier Publishing. He lectures and conducts workshops on outcome-based education (OBE) and its implementations including assessments.
Learning Outcomes

After completing this chapter, students should be able to do the following:

• Describe the historical development of electronics.
• List electronic systems and their classifications.
• List the types of electronic amplifiers.
• Describe what constitutes engineering design.
• Describe the design process of electronic circuits and systems.
• List some electronic devices and describe their basic input and output characteristics.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_{V}, A_v$</td>
<td>DC and small-signal voltage gains</td>
</tr>
<tr>
<td>$BW, A_{PB}$</td>
<td>Bandwidth and pass-band voltage gain</td>
</tr>
<tr>
<td>$f_H, f_L$</td>
<td>High and low cutoff frequencies</td>
</tr>
<tr>
<td>$t_d, t_r, t_f, t_{on}, t_{off}$</td>
<td>Delay, rise, fall, on, and off times</td>
</tr>
<tr>
<td>$T_s$</td>
<td>Sampling time</td>
</tr>
<tr>
<td>$T, f$</td>
<td>Period and frequency of a signal</td>
</tr>
<tr>
<td>$v_i(t), v_o(t)$</td>
<td>Instantaneous input and output voltages</td>
</tr>
<tr>
<td>$V_i, V_o$</td>
<td>rms (root mean square) input and output voltages</td>
</tr>
</tbody>
</table>
1.1 Introduction

We encounter electronics in our daily life in the form of telephones, radios, televisions, audio equipments, home appliances, computers, and equipments for industrial control and automation. Electronics have become the stimuli for and an integral part of modern technological growth and development. The field of electronics deals with the design and applications of electronic devices. This chapter serves as an introduction to electronics.

1.2 History of Electronics

The age of electronics began with the invention of the first amplifying device, the triode vacuum tube, by Fleming in 1904. This invention was followed by the development of the solid-state point-contact diode (silicon) by Pickard in 1906, the first radio circuits from diodes and triodes between 1907 and 1927, the super heterodyne receiver by Armstrong in 1920, demonstration of television in 1925, the field-effect device by Lilienfield in 1925, frequency modulation (FM) by Armstrong in 1933, and radar in 1940.

The first electronics revolution began in 1947 with the invention of the silicon transistor by Bardeen, Bratian, and Shockley at Bell Telephone Laboratories. Most of today’s advanced electronic technologies are traceable to that one invention. This revolution was followed by the first demonstration of color television in 1950 and the invention of the unipolar field-effect transistor by Shockley in 1952.

The next breakthrough came in 1956, when Bell Laboratories developed the pn-pn triggering transistor, also known as a thyristor or a silicon-controlled rectifier (SCR). The second electronics revolution began with the development of a commercial thyristor by General Electric Company in 1958. That was the beginning of a new era for applications of electronics in power processing or conditioning, called power electronics. Since then, many different types of power semiconductor devices and conversion techniques have been developed.

The first integrated circuit (IC) was developed in 1958 simultaneously by Kilby at Texas Instruments and Noyce and Moore at Fairchild Semiconductor, marking the beginning of a new phase in the microelectronics revolution. This invention was followed by development of the first commercial IC operational amplifier, the μA709, by Fairchild Semiconductor in 1968; the 4004 microprocessor by Intel in 1971; the 8-bit microprocessor by Intel in 1972; and the gigabit memory chip by Intel in 1995. The progression from vacuum tubes to microelectronics is shown in Fig. 1.1. Integrated circuit development

![FIGURE 1.1 Progression from vacuum tubes to microelectronics](image)
TABLE 1.1  Levels of integration

<table>
<thead>
<tr>
<th>Date</th>
<th>Degree of Integration</th>
<th>Number of Components per Chip</th>
</tr>
</thead>
<tbody>
<tr>
<td>1950s</td>
<td>Discrete components</td>
<td>1 to 2</td>
</tr>
<tr>
<td>1960s</td>
<td>Small-scale integration (SSI)</td>
<td>Fewer than $10^2$</td>
</tr>
<tr>
<td>1966</td>
<td>Medium-scale integration (MSI)</td>
<td>From $10^2$ to $10^3$</td>
</tr>
<tr>
<td>1969</td>
<td>Large-scale integration (LSI)</td>
<td>From $10^3$ to $10^4$</td>
</tr>
<tr>
<td>1975</td>
<td>Very-large-scale integration (VLSI)</td>
<td>From $10^4$ to $10^9$</td>
</tr>
<tr>
<td>1990s</td>
<td>Ultra-large-scale integration (ULSI)</td>
<td>More than $10^9$</td>
</tr>
</tbody>
</table>

continues today in an effort to achieve higher-density chips with lower power dissipation; historical levels of integration in circuits are shown in Table 1.1.

The degree of device integration continues to follow Moore’s law, which is an observation made by Gordon E. Moore that the number of transistors inside an IC could be doubled every 24 months at a density that also minimizes the cost of a transistor [1]. Figure 1.2(a) shows the growth in the number of transistors on ICs over the years. Figure 1.2(b) shows the generations of microelectronics technology [2].

![Figure 1.2](http://commons.wikimedia.org/wiki/File:Moore_Law_diagram_(2004).jpg) and generations of microelectronic technology (Continued)

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
Since the invention of the first amplifying device, the vacuum tube, in 1904, the field of electronics has evolved rapidly. Today ultra-large-scale integrated (ULSI) circuits have more than $10^9$ components per chip.

1.3 Electronic Systems

An electronic system is an arrangement of electronic devices and components with a defined set of inputs and outputs. Using transistors (trans-resistors) as devices, it takes in information in the form of input signals (or simply inputs), performs operations on them, and then produces output signals (or outputs). Electronic systems may be categorized according to the type of application, such as communication system, medical electronics, instrumentation, control system, or computer system.
A block diagram of an FM radio receiver is shown in Fig. 1.3(a). The antenna acts as the sensor. The input signal from the antenna is small, usually in the microvolt range; its amplitude and power level are amplified by the electronic system before the signal is fed into the speaker. A block diagram of a temperature display instrument is shown in Fig. 1.3(b). The output drives the display instrument. The temperature sensor produces a small voltage, usually in millivolts per unit temperature rise above 0°C (e.g., 1 mV/°C). Both systems take an input from a sensor, process it, and produce an output to drive an actuator.

An electronic system must communicate with input and output devices. In general, the inputs and outputs are in the form of electrical signals. The input signals may be derived from the measurement of physical qualities such as temperature or liquid level, and the outputs may be used to vary other physical qualities such as those of display and heating elements. Electronic systems often use sensors to sense external input qualities and actuators to control external output qualities. Sensors and actuators are often called transducers. The loudspeaker is an example of a transducer that converts an electronic signal into sound.

### 1.3.1 Sensors

There are many types of sensors, including the following:
- Thermistors and thermocouples to measure temperature
- Phototransistors and photodiodes to measure light
- Strain gauges and piezoelectric materials to measure force
- Potentiometers, inductive sensors, and absolute position encoders to measure displacement
- Tachogenerators, accelerometers, and Doppler effect sensors to measure motion
- Microphones to measure sound
- Anemometer to measure the wind speed

### 1.3.2 Actuators

Actuators produce a nonelectrical output from an electrical signal. There are many types of actuators, including the following:
- Resistive heaters to produce heat
- Light-emitting diodes (LEDs) and light dimmers to control the amount of light
- Solenoids to produce force
Microelectronic Circuits: Analysis and Design

- Meters to indicate displacement
- Electric motors to produce motion or speed
- Speakers and ultrasonic transducers to produce sound

### KEY POINTS OF SECTION 1.3

- An electronic system consists of electronic devices and components. It processes electronic signals, acting as an interface between sensors on the input side and as actuators on the output side.
- Sensors convert physical qualities to electrical signals, whereas actuators convert electrical signals to physical qualities. Sensors and actuators are often called transducers.

### 1.4 Electronic Signals and Notation

Electronic signals can be categorized into two types: analog and digital. An analog signal has a continuous range of amplitudes over time, as shown in Fig. 1.4(a). Figure 1.4(b) is the sampled form of the input signal in Fig. 1.4(a). A digital signal assumes only discrete voltage values over time, as shown in Fig. 1.4(c). A digital signal has only two values, representing binary logic state 1 (for high level) and binary logic state 0 (for low level). To accommodate variations in component values, temperature, and noise (or extraneous
signals), logic state 1 is usually assigned to any voltage between 2 V and 5 V. Logic state 0 may be assigned to any voltage between 0 and 0.8 V.

The output signal of a sensor is usually of the analog type, and actuators often require analog input to produce the desired output. An analog signal can be converted to digital form and vice versa. The electronic circuits that perform these conversions are called analog-to-digital (A/D) and digital-to-analog (D/A) converters.

1.4.1 Analog-to-Digital Converters

An A/D converter converts an analog signal to digital form and provides an interface between analog and digital signals. Consider the analog input voltage shown in Fig. 1.5(a). The input signal is sampled at periodic intervals determined by the sampling time $T_s$, and an $n$-bit binary number $(b_1b_2\ldots b_n)$ is assigned to each sample, as shown in Fig. 1.5(b) for $n = 3$. The $n$-bit binary number is a binary fraction

![Analog-to-digital conversion](Image)

**FIGURE 1.5** Analog-to-digital conversion
that represents the ratio between the unknown input voltage \( v_I \) and the full-scale voltage \( V_{FS} \) of the converter. For \( n = 3 \), each binary fraction is \( V_{FS}/2^n = V_{FS}/8 \). The output voltage of a 3-bit A/D converter is shown in Fig. 1.5(c).

The input–output relation shown in Fig. 1.5(c) indicates that as the input voltage increases from 0 to full-scale voltage, the binary output steps up from 000 to 111. However, the binary number remains constant for an input voltage range of \( V_{FS}/2^n \) (for \( n = 3 \)), which is equal to one least significant bit (LSB) of the A/D converter. Thus as the input voltage increases, the binary output will give first a negative error and then a positive error, as shown in Fig. 1.5(d). This error, called the quantization error, can be reduced by increasing the number of bits \( n \). Thus, the quantization error may be defined as the smallest voltage that can change the LSB of the binary output from 0 to 1. The quantization error is also called the resolution of the converter, and it can be found from

\[
V_{LSB} = V_{error} = \frac{V_{FS}}{2^n} \tag{1.1}
\]

where \( V_{FS} \) is the full-scale voltage of the converter. For example, \( V_{LSB} \) for an 8-bit converter of \( V_{FS} = 5 \) V is

\[
V_{LSB} = \frac{V_{FS}}{2^n} = \frac{5}{2^8} = 19.53 \text{ mV} \approx 20 \text{ mV}
\]

### 1.4.2 Digital-to-Analog Converters

A D/A converter takes an input signal in binary form and produces an output voltage or current in an analog (or continuous) form. A block diagram of an \( n \)-bit D/A converter consisting of binary digits \((b_1b_2 \ldots b_n)\) is shown in Fig. 1.6. It is assumed that the converter generates the binary fraction, which is multiplied by the full-scale voltage \( V_{FS} \) to give the output voltage, expressed by

\[
V_O = (b_12^{-1} + b_22^{-2} + b_32^{-3} + \cdots + b_n2^{-n})V_{FS} \tag{1.2}
\]

where the \( i \)th binary digit is either \( b_i = 0 \) or \( b_i = 1 \) and \( b_1 \) is the most significant bit (MSB). For example, for \( V_{FS} = 5 \) V, \( n = 3 \), and a binary word \( b_1b_2b_3 = 110 \), Eq. (1.2) gives

\[
V_O = (1 \times 2^{-1} + 1 \times 2^{-2} + 0 \times 2^{-3}) \times 5 = 3.75 \text{ V}
\]

### 1.4.3 Notation

An analog signal is normally represented by a symbol with a subscript. The symbol and the subscript can be either uppercase or lowercase, according to the conventions shown in Table 1.2. For example, consider the circuit in Fig. 1.7(a), whose input consists of a DC voltage \( V_{DC} = 5 \text{ V} \) and an AC voltage \( v_{ab} = 2 \sin \omega t \).
The instantaneous voltages are shown in Fig. 1.7(b). The definitions of voltage and current symbols are as follows:

1. $V_{\text{DC}}$ and $I_{\text{DC}}$ are DC values: uppercase variables and uppercase subscripts.
   
   \[ V_{\text{DC}} = 5 \text{ V} \]
   \[ I_{\text{DC}} = \frac{V_{\text{DC}}}{R_L} = 5 \text{ mA} \]

2. $v_{\text{ab}}$ and $i_{\text{a}}$ are instantaneous AC values: lowercase variables and lowercase subscripts.
   
   \[ v_{\text{ab}} = 2 \sin \omega t \]
   \[ i_{\text{a}} = 2 \sin \omega t \text{ mA} \quad \text{(for } R_L = 1 \text{ k}\Omega) \]

3. $v_{\text{AB}}$ and $i_{\text{A}}$ are total instantaneous values: lowercase variables and uppercase subscripts.
   
   \[ v_{\text{AB}} = V_{\text{DC}} + v_{\text{ab}} = 5 + 2 \sin \omega t \]
   \[ i_{\text{A}} = I_{\text{DC}} + i_{\text{a}} = 5 \text{ mA} + 2 \sin \omega t \text{ mA} \quad \text{(for } R_L = 1 \text{ k}\Omega) \]

4. $V_{\text{ab}}$ and $I_{\text{a}}$ are total rms values: uppercase variables and lowercase subscripts.
   
   \[ V_{\text{ab}} = \sqrt{5^2 + \left( \frac{2}{\sqrt{2}} \right)^2} = 5.20 \text{ V} \]
   \[ I_{\text{a}} = \sqrt{5^2 + \left( \frac{2}{\sqrt{2}} \right)^2} = 5.20 \text{ mA} \quad \text{(} \sqrt{2} \text{ factor is used to convert the peak value to a rms value)} \]

\[ i_{\text{A}} = I_{\text{DC}} + i_{\text{a}} \]
\[ V_{\text{AB}} = V_{\text{DC}} + v_{\text{AB}} \]

\[ v_{\text{ab}} = 2 \sin \omega t \]
\[ i_{\text{a}} = 2 \sin \omega t \text{ mA} \quad \text{(for } R_L = 1 \text{ k}\Omega) \]

\[ v_{\text{AB}} = V_{\text{DC}} + v_{\text{ab}} \]

\[ I_{\text{a}} = \sqrt{5^2 + \left( \frac{2}{\sqrt{2}} \right)^2} = 5.20 \text{ mA} \quad \text{(} \sqrt{2} \text{ factor is used to convert the peak value to a rms value)} \]

\[ i_{\text{A}} = I_{\text{DC}} + i_{\text{a}} \]
\[ v_{\text{AB}} = V_{\text{DC}} + v_{\text{AB}} \]

\[ v_{\text{AB}} = V_{\text{DC}} + v_{\text{ab}} \]

\[ I_{\text{a}} = \sqrt{5^2 + \left( \frac{2}{\sqrt{2}} \right)^2} = 5.20 \text{ mA} \quad \text{(} \sqrt{2} \text{ factor is used to convert the peak value to a rms value)} \]
1.5 Classifications of Electronic Systems

The form of signal processing carried out by an electronic system depends on the nature of the input signals, the output requirements of the actuators, and the overall functional requirement. However, certain functions are common to a large number of systems. These include amplification, addition and subtraction of signals, integration and differentiation of signals, and filtering. Some systems require a sequence of operations such as counting, timing, setting, resetting, and decision making. Also, it may be necessary to generate sinusoidal or other signals within a system.

Electronic systems find applications in automobiles, home entertainment, office and communication equipments, and medicines, among other areas, and help us maintain our high-tech lifestyles. Electronic systems are often classified according to the type of application:

- Automobile electronics
- Communication electronics
- Consumer electronics
- Industrial electronics
- Instrumentation electronics
- Mechatronics
- Medical electronics
- Office electronics

The field of electronics is divided into three distinct areas, depending on the type of signals and processing required by the electronic systems.

**Analog electronics** deals primarily with the operation and applications of transistors as amplifying devices. The input and output signals take on a continuous range of amplitude values over time. The function of analog electronics is to transport and process the information contained in an analog input signal with a minimum amount of distortion.

**Digital electronics** deals primarily with the operation and applications of transistors as “on” and “off” switching devices. Both input and output signals are discontinuous pulse signals that occur at uniformly spaced points in time. The function of digital electronics is to transport and process the information contained in a digital input signal with a minimum amount of error at the fastest speed.

**Power electronics** deals with the operation and applications of power semiconductor devices, including power transistors, as “on” and “off” switches for the control and conversion of electric power. Analog and/or digital electronics are used to generate control signals for the switching power devices in order to obtain the desired conversion strategies (AC/DC, AC/AC, DC/AC, or DC/DC) with the maximum conversion efficiency and the minimum amount of waveform distortion. The input to a power electronic system is a DC or an AC power supply voltage (or current). Power electronics is primarily concerned with power content and quality rather than the information contained in a signal. For example, a power...
electronic circuit can provide a stable DC power supply, say 12 V to an analog system and 5 V to a digital system, from an AC supply of 120 V at 60 Hz.

Microelectronics has given us the ability to generate and process control signals at an incredible speed. Power electronics has given us the ability to shape and control large amounts of power with a high efficiency—between 94% and 99%. Many potential applications of power electronics are now arising from the marriage of power electronics—the muscle—with microelectronics—the brain. Also, power electronics has emerged as a distinct discipline and is revolutionizing the concept of power processing and conditioning for industrial power control and automation.

Many electronic systems use both analog and digital techniques. Each method of implementation has advantages and disadvantages, summarized in the following list:

- Noise is usually present in electronic circuits. It is defined as the extraneous signal that arises from the thermal agitation of electrons in a resistor, the inductive or capacitive coupling of signals from other systems, or other sources. Noise is added directly to analog signals and hence affects the signals, as shown in Fig. 1.8(a). Thus noise is amplified by the subsequent amplification stages. Since digital signals have only two levels (high or low), noise will not affect the digital output, shown in Fig. 1.8(b), and can effectively be removed from digital signals.
- An analog circuit requires fewer individual components than a digital circuit to perform a given function. However, an analog circuit often requires large capacitors or inductors that cannot be manufactured in ICs.
- A digital circuit tends to be easier to implement than an analog circuit in ICs, although it can be more complex than an analog circuit. Digital circuits, however, generally offer much higher quality and speed of signal processing.
- Analog systems are designed to perform specific functions or operations, whereas digital systems are adaptable to a variety of tasks or uses.
- Signals from sensors and to actuators in electronic systems are generally analog. If an input signal has a low magnitude and must be processed at very high frequencies, then the analog technique is required. For optimal performance and design, both analog and digital approaches are often used.

**KEY POINT OF SECTION 1.5**

- Electronics can be classified into three areas: analog, digital, and power electronics. The classification is based primarily on the type of signal processing. Electronic systems are often classified according to the type of application such as medical electronics and consumer electronics.
1.6 Specifications of Electronic Systems

An electronic system is normally designed to perform certain functions or operations. The performance of an electronic system is specified or evaluated in terms of voltage, current, impedance, power, time, and frequency at the input and output of the system. The performance parameters include transient specifications, distortion, frequency specifications, and DC and small-signal specifications.

1.6.1 Transient Specifications

Transient specifications refer to the output signal of a circuit generated in response to a specified input signal, usually a repetitive pulse signal, as shown in Fig. 1.9(a). The output signal usually goes through a delay time $t_d$, rise time $t_r$, on time $t_{on}$, fall time $t_f$, and off time $t_{off}$ in every cycle, as shown in Fig. 1.9(b). Depending on the damping factor of the circuit, the response may exhibit an overshoot before settling into the steady-state condition, as shown by the dashed curve in Fig. 1.9(b). The times associated with an output signal are defined as follows:

- **Delay time** $t_d$ is the time before the circuit can respond to any input signal.
- **Rise time** $t_r$ is the time required for the output to rise from 10% to 90% of its final (high) value.
- **On time** $t_{on}$ is the time during which the circuit is fully turned on and is functioning in its normal mode.
- **Fall time** $t_f$ is the time required for the output to decrease from 90% to 10% of its initial (high) value.
- **Off time** $t_{off}$ is the time during which the circuit is completely off, not operating.

Thus, the switching period $T$ is

$$T \approx t_d + t_r + t_{on} + t_f + t_{off}$$  \hspace{1cm} (1.3)

and the switching frequency is $f = 1/T$. These times limit the maximum switching speed $f_{\text{max}}$ of a circuit. For example, the maximum switching frequency of a circuit with $t_d = 1 \mu s$ and $t_r = t_f = 2 \mu s$ is

$$f_{\text{max}} = \frac{1}{(t_d + t_r + t_f)} = \frac{1}{5 \mu s} = 200 \text{ kHz}$$

![FIGURE 1.9 Pulse response of a circuit](image-url)
1.6.2 Distortion

While passing through different stages within an electronic system, a signal often gets distorted. Distortion may take many forms and can alter the shape, amplitude, frequency, or phase of a signal. Some examples of distortion are shown in Fig. 1.10: part (b) shows clipping of the original sine wave in part (a) due to the power supply limit, part (c) shows crossover distortion due to ineffectiveness of the circuit near zero crossing, and part (d) shows harmonic distortion due to nonlinear characteristics of electronic devices. A sinusoidal input signal of a specified frequency is usually applied to the input of a circuit, and then the fundamental and harmonic components of the output signal are measured. The amount of distortion is specified as the total harmonic distortion (THD), which is the ratio of the rms value of the harmonic component to the rms value of the fundamental component (at the frequency of the sinusoidal input). The THD should be as low as possible.

1.6.3 Frequency Specifications

The range of signal frequencies of electronic signals varies widely, depending on the application, as shown in Table 1.3. The frequency specifications refer to the plot of the output signal as a function of the input signal frequency. A typical plot for a system such as the one in Fig. 1.11(a) is shown in Fig. 1.11(b). For frequencies less than $f_L$ and greater than $f_H$, the output is attenuated. But for frequencies between $f_L$ and $f_H$, the output remains almost constant. The frequency range from $f_L$ to $f_H$ is called the bandwidth BW of the circuit. That is, $BW = f_H - f_L$. A system with a bandwidth like the one shown in Fig. 1.11(b) is said to have a band-pass characteristic. If $f_L = 0$, the system is said to have a low-pass characteristic. If $f_H = \infty$, the system is said to have a high-pass characteristic.
For an operating frequency within the bandwidth or pass-band range, the voltage gain is defined as

\[ A_{PB} = \frac{V_o}{V_i} \]  

(1.4)

where \( V_i \) and \( V_o \) are the rms values of the input and output voltages, respectively. The input impedance is defined as

\[ Z_i = \frac{V_i}{I_i} \]  

(1.5)

where \( I_i \) is the rms value of the input current of the circuit. \( Z_i \) is often referred to as the small-signal input resistance \( R_i \) because the output is almost independent of the frequency in the midband range. Ideally, \( R_i \) should tend to infinity. Thevenin’s equivalent resistance seen from the output side is specified as the output impedance \( Z_o \) or the output resistance \( R_o \), which should ideally be zero.

### 1.6.4 DC and Small-Signal Specifications

The DC and small-signal specifications include the DC power supply \( V_{CC} \), DC biasing currents (required to activate and operate internal transistors), and power dissipation \( P_D \) (power requirement from the DC power supply). The voltage gain (the ratio of the output voltage \( V_o \) to the input voltage \( V_i \)) is

<table>
<thead>
<tr>
<th>Signal Type</th>
<th>Bandwidth</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seismic signals</td>
<td>1 Hz to 200 Hz</td>
</tr>
<tr>
<td>Electrocardiograms</td>
<td>0.05 Hz to 100 Hz</td>
</tr>
<tr>
<td>Audio signals</td>
<td>20 Hz to 15 kHz</td>
</tr>
<tr>
<td>Video signals</td>
<td>DC to 4.2 MHz</td>
</tr>
<tr>
<td>AM radio signals</td>
<td>540 kHz to 1600 kHz</td>
</tr>
<tr>
<td>Radar signals</td>
<td>1 MHz to 100 MHz</td>
</tr>
<tr>
<td>VHF TV signals</td>
<td>54 MHz to 60 MHz</td>
</tr>
<tr>
<td>FM radio signals</td>
<td>88 MHz to 806 MHz</td>
</tr>
<tr>
<td>UHF TV signals</td>
<td>470 MHz to 806 MHz</td>
</tr>
<tr>
<td>Cellular telephone signals</td>
<td>824 MHz to 891.5 MHz</td>
</tr>
<tr>
<td>Satellite TV signals</td>
<td>3.7 GHz to 4.2 GHz</td>
</tr>
<tr>
<td>Microwave communication signals</td>
<td>1 GHz to 50 GHz</td>
</tr>
</tbody>
</table>

For an operating frequency within the bandwidth or pass-band range, the voltage gain is defined as

\[ A_{PB} = \frac{V_o}{V_i} \]

where \( V_i \) and \( V_o \) are the rms values of the input and output voltages, respectively. The input impedance is defined as

\[ Z_i = \frac{V_i}{I_i} \]

where \( I_i \) is the rms value of the input current of the circuit. \( Z_i \) is often referred to as the small-signal input resistance \( R_i \) because the output is almost independent of the frequency in the midband range. Ideally, \( R_i \) should tend to infinity. Thevenin’s equivalent resistance seen from the output side is specified as the output impedance \( Z_o \) or the output resistance \( R_o \), which should ideally be zero.

### 1.6.4 DC and Small-Signal Specifications

The DC and small-signal specifications include the DC power supply \( V_{CC} \), DC biasing currents (required to activate and operate internal transistors), and power dissipation \( P_D \) (power requirement from the DC power supply). The voltage gain (the ratio of the output voltage \( V_o \) to the input voltage \( V_i \)) is

<table>
<thead>
<tr>
<th>Signal Type</th>
<th>Bandwidth</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seismic signals</td>
<td>1 Hz to 200 Hz</td>
</tr>
<tr>
<td>Electrocardiograms</td>
<td>0.05 Hz to 100 Hz</td>
</tr>
<tr>
<td>Audio signals</td>
<td>20 Hz to 15 kHz</td>
</tr>
<tr>
<td>Video signals</td>
<td>DC to 4.2 MHz</td>
</tr>
<tr>
<td>AM radio signals</td>
<td>540 kHz to 1600 kHz</td>
</tr>
<tr>
<td>Radar signals</td>
<td>1 MHz to 100 MHz</td>
</tr>
<tr>
<td>VHF TV signals</td>
<td>54 MHz to 60 MHz</td>
</tr>
<tr>
<td>FM radio signals</td>
<td>88 MHz to 806 MHz</td>
</tr>
<tr>
<td>UHF TV signals</td>
<td>470 MHz to 806 MHz</td>
</tr>
<tr>
<td>Cellular telephone signals</td>
<td>824 MHz to 891.5 MHz</td>
</tr>
<tr>
<td>Satellite TV signals</td>
<td>3.7 GHz to 4.2 GHz</td>
</tr>
<tr>
<td>Microwave communication signals</td>
<td>1 GHz to 50 GHz</td>
</tr>
</tbody>
</table>

For an operating frequency within the bandwidth or pass-band range, the voltage gain is defined as

\[ A_{PB} = \frac{V_o}{V_i} \]

(1.4)

where \( V_i \) and \( V_o \) are the rms values of the input and output voltages, respectively. The input impedance is defined as

\[ Z_i = \frac{V_i}{I_i} \]

(1.5)

where \( I_i \) is the rms value of the input current of the circuit. \( Z_i \) is often referred to as the small-signal input resistance \( R_i \) because the output is almost independent of the frequency in the midband range. Ideally, \( R_i \) should tend to infinity. Thevenin’s equivalent resistance seen from the output side is specified as the output impedance \( Z_o \) or the output resistance \( R_o \), which should ideally be zero.

### 1.6.4 DC and Small-Signal Specifications

The DC and small-signal specifications include the DC power supply \( V_{CC} \), DC biasing currents (required to activate and operate internal transistors), and power dissipation \( P_D \) (power requirement from the DC power supply). The voltage gain (the ratio of the output voltage \( V_o \) to the input voltage \( V_i \)) is

<table>
<thead>
<tr>
<th>Signal Type</th>
<th>Bandwidth</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seismic signals</td>
<td>1 Hz to 200 Hz</td>
</tr>
<tr>
<td>Electrocardiograms</td>
<td>0.05 Hz to 100 Hz</td>
</tr>
<tr>
<td>Audio signals</td>
<td>20 Hz to 15 kHz</td>
</tr>
<tr>
<td>Video signals</td>
<td>DC to 4.2 MHz</td>
</tr>
<tr>
<td>AM radio signals</td>
<td>540 kHz to 1600 kHz</td>
</tr>
<tr>
<td>Radar signals</td>
<td>1 MHz to 100 MHz</td>
</tr>
<tr>
<td>VHF TV signals</td>
<td>54 MHz to 60 MHz</td>
</tr>
<tr>
<td>FM radio signals</td>
<td>88 MHz to 806 MHz</td>
</tr>
<tr>
<td>UHF TV signals</td>
<td>470 MHz to 806 MHz</td>
</tr>
<tr>
<td>Cellular telephone signals</td>
<td>824 MHz to 891.5 MHz</td>
</tr>
<tr>
<td>Satellite TV signals</td>
<td>3.7 GHz to 4.2 GHz</td>
</tr>
<tr>
<td>Microwave communication signals</td>
<td>1 GHz to 50 GHz</td>
</tr>
</tbody>
</table>
often specified. If the $v_O-v_I$ relationship is linear, as shown in Fig. 1.12(a), and the circuit operates at a quiescent point $Q$, the voltage gain is given by

$$A_V = \frac{V_O}{V_I}$$

(a) Linear relationship

Electronic circuits, especially amplifiers, are normally operated over a practically linear range of the characteristic. For an operating frequency within the BW of the circuit, $A_V \equiv A_{PB}$, where $A_{PB}$ is the pass-band or midfrequency gain of the amplifier.

**KEY POINT OF SECTION 1.6**

- The parameters that describe the performance of electronic circuits and systems usually include transient specifications, distortion, frequency specifications, and large- and small-signal specifications.

### 1.7 Types of Amplifiers

There are many types of amplifiers, which can be classified according to the type of signal amplification, the function, the type of interstage coupling, the frequency range, and the type of load.

*Signal amplification types* are classified by the types of input and output signals:

1. A voltage amplifier produces an amplified output voltage in response to an input voltage signal.
2. A transconductance amplifier produces an amplified output current in response to an input voltage signal.
3. A current amplifier produces an amplified output current in response to an input current signal.
4. An impedance amplifier produces an amplified output voltage in response to an input current signal.
5. A power amplifier produces an amplified output voltage and delivers power to a low resistance load in response to an input voltage signal.

*Functional types* are classified by their function or output characteristics:

1. A linear amplifier produces an output signal in response to an input signal without introducing significant distortion on the output signal, whereas a nonlinear amplifier does introduce distortion.
2. An audio amplifier is a power amplifier in the audio frequency (AF) range.
3. An operation amplifier performs some mathematical functions for instruments and for signal processing.
4. A wideband amplifier amplifies an input signal over a wide range of frequencies to boost signal levels, whereas a narrowband amplifier amplifies a signal over a specific narrow range of frequencies.
5. A radio frequency (RF) amplifier amplifies a signal for use over the RF range.
6. A servo amplifier uses a feedback loop to control the output at a desired level.

*Interstage coupling types* are classified by the coupling method of the signal at the input, at the output, or between stages:

1. An RC-coupled amplifier uses a network of resistors and capacitors to connect it to the following and preceding amplifier stages.
2. An LC-coupled amplifier uses a network of inductors and capacitors to connect it to the following and preceding amplifier stages.
3. A transformer-coupled amplifier uses transformers to match impedances to the load side and input side.
4. A direct-coupled amplifier uses no interstage elements, and each stage is connected directly to the following and preceding amplifier stages.

*Frequency types* are classified in accordance to the frequency range:

1. A DC amplifier is capable of amplifying signals from zero frequency (DC) and above.
2. An AF amplifier is capable of amplifying signals from 20 Hz to 20 kHz.
3. A video amplifier (VA) is capable of amplifying signals up to a few hundred megahertz (<10 MHz for TV).
4. An ultra-high-frequency (UHF) amplifier is capable of amplifying signals up to a few gigahertz.

*Load types* are classified in accordance to the type of load:

1. An audio amplifier has an audio type of load.
2. A video amplifier has a video type of load.
3. A tuned amplifier amplifies a single RF or band of frequencies.

**KEY POINT OF SECTION 1.7**

- Amplifiers can be classified according to the type of signal amplification, the function, the type of interstage coupling, the frequency range, and the type of load.
1.8 Design of Electronic Systems

Engineering systems are becoming increasingly complex. Thus, it is highly desirable that engineers have the skills needed to analyze, synthesize, and design complex systems. A design transforms specifications into circuits that satisfy those specifications. Designing a system is a challenging task involving many variables. One can use different approaches to implement the same specifications, and hence many decisions must be made in implementing the specifications.

In practical design work, the most challenging tasks are attacked first, and then the simple tasks are tackled. That way, if an acceptable solution cannot be found to the difficult problems, time and money are not wasted on solving easier problems. Thus, the engineering design process follows a hierarchy in which systems are designed first through functional block diagrams, after which circuits and then devices are designed. This approach is the opposite of what is normally taught in academic courses. The system-level design is conceptualized and expressed in terms of functional blocks and system integration [3]. The major steps in the design process, shown in Fig. 1.13, are as follows:

1. General product description
2. Definition of specifications/requirements

---

**FIGURE 1.13** System-level design process
3. System design through functional block diagrams
4. Definition of specifications of functional blocks for circuit-level synthesis and implementation
5. System integration
6. Simulation or modeling
7. Testing and verification

The system-level solution to designing the radio receiver in Fig. 1.3(a) is shown in Fig. 1.14. It includes RF, intermediate frequency (IF), and AF amplifiers. The local oscillator tunes the radio receiver to receive the signal of a desired station.

Only the broad outlines of the design process are given here. The details depend on the type of system being designed. The design process may be viewed as a means to accomplish the following [4]:

1. Identify needs.
2. Generate ideas for meeting the needs.
3. Refine the ideas.
4. Analyze all possible solutions.
5. Decide on the action to be taken.
6. Implement the decision.

These steps are shown in Fig. 1.15. The steps are repeated until the desired specifications have been satisfied. Each of these six steps can be subdivided, as shown in Fig. 1.16. As the figure suggests, engineering
design involves many disciplines, and a design engineer must be able to function in a multidisciplinary team and communicate effectively with other team members.

**KEY POINT OF SECTION 1.8**

- In practical design work, the most challenging tasks are attacked before the simple tasks. Thus the engineering design process follows a hierarchy in which systems are designed first through functional block diagrams, after which circuits, and then devices are designed.
1.9 Design of Electronic Circuits

A circuit-level design is implemented and expressed in terms of components, devices, and voltage–current relationships. The lowest level is device-level design, which involves selecting types of devices. Before starting this level of design, you must have some knowledge of electronic devices and their characteristics, parameters, and models.

1.9.1 Analysis versus Design

Analysis is the process of finding the unique specifications or properties of a given circuit. Design, on the other hand, is the creative process of developing a solution to a problem. We start with a desired set of specifications or properties and find a circuit that satisfies them. The solution is not unique, and finding it requires synthesis. For example, the current flowing from a 12-V battery to a 5-Ω load resistance is simply 2.4 A. However, if you were asked to arrange a load that would draw 2.4 A from a battery of 12 V, you could use many possible combinations of series and parallel resistors. Figure 1.17 shows a comparison of analysis and design.

1.9.2 Definition of Engineering Design

What is engineering design? If you asked several different engineers, you would probably get several different definitions. The Accreditation Board for Engineering and Technology (ABET) provides the following broad definition [5]:

Engineering design is the process of devising a system, component, or process to meet desired needs. It is a decision-making process (often iterative), in which the basic sciences and mathematics and engineering sciences are applied to convert resources optimally to meet these stated needs. Among the fundamental elements of the design process are the establishment of objectives and criteria, synthesis, analysis, construction if feasible, testing, and evaluation. The engineering design component of a curriculum must include most of the following features: development of student creativity, use of open-ended problems, development and use of design theory and methodology, formulation of design problem statements and specifications, consideration of alternative solutions, feasibility considerations, production processes, concurrent engineering design, and detailed system descriptions. Further, it is essential to incorporate appropriate engineering standards and include multiple realistic constraints such as economic, environmental, social, political, ethical, health and safety, manufacturability, and sustainability.

![FIGURE 1.17 Analysis versus design](image-url)
1.9.3 The Circuit-Level Design Process

The major steps in the circuit-level design process, shown in Fig. 1.18, are as follows:

Step 1. Study the design problem.
Step 2. Define the design objectives—that is, establish the design’s performance requirements.
Step 3. Establish the design strategy, and find the functional block diagram solution.
Step 4. Select the circuit topology or configuration after evaluating alternative solutions.
Step 5. Select the component values and devices. Analysis and synthesis may be required to find the component values. Use simple device models to simplify the analytical derivations.
Step 6. Evaluate your design, and predict its performance. Modify your design values if necessary.
**EXAMPLE 1.1**

**Carrying out the design process**  Design a circuit to measure DC voltage in the range from 0 to 20 V. For a full-scale deflection, the indicating meter draws 100 μA at a voltage of 1 V across it. The current drawn from the DC supply should not exceed 1 mA.

**SOLUTION**

**Step 1.** Study the design problem carefully so that you can define the design objectives succinctly in engineering terms.

**Step 2.** Define the design objectives by means of a design statement, performance requirements, design constraints, and design criteria.

- **The design statement** expresses the objective in a single sentence with few or no numbers—for example,
  
  Design of a DC indicating meter

- **The performance requirements** must be specific and related to the required performance characteristics in terms of voltage, current, impedance, power, time, frequency, and so on. The values refer to the input and output terminals of the circuit and are normally expressed in mathematical inequalities—for example,
  
  Meter current \( I_M \leq 100 \, \mu A \)

  DC supply voltage \( V_{DC} = 0 \) to 20 V

  DC supply current \( I_{DC} \leq 1 \) mA

  Meter voltage \( V_M = 1 \) V

  The designer, in general, has no flexibility to modify these constraints.

- **The design constraints** are the limitations imposed by the system-level design process—for example,

  - The excess of \( I_M \) over 100 μA should be a minimum, say 5%:
    
    \( \Delta I_M = 5 \, \mu A \)

  - The value of \( I_{DC} \) under 1 mA should be a maximum, say within 15%:
    
    \( \Delta I_{DC} = 150 \, \mu A \)

  - The cost must be kept to a minimum.
Step 3. Establish the design strategy, and find the functional block diagram solution. This solution is shown in Fig. 1.19.

Step 4. Select the circuit configuration after evaluating alternative solutions. Many different circuit configurations (such as using zener diodes) could perform the function of providing the meter current at a specific voltage. For this example we will use a simple circuit that utilizes the principle of voltage division. This circuit is shown in Fig. 1.20. Note that this is not a unique arrangement. We could remove $R_2$ and still satisfy the specifications.

Step 5. Select the component values after analyzing the circuit. The meter can be represented by a resistance $R_M$:

$$R_M = \frac{V_M}{I_M} = \frac{1 \text{ V}}{100 \mu\text{A}} = 10 \text{ k}\Omega$$

The value of resistance $R_1$ can be found from

$$R_1 = \frac{(V_{DC} - V_M)}{I_{DC}} = \frac{(20 - 1)}{1 \text{ mA}} = 19 \text{ k}\Omega$$

To keep the cost down, we will use a 5% carbon resistor. From the tables of available resistors in Appendix E, we find that 20 k\Omega is the nearest higher value for a 5% carbon resistor. That is,

$$R_1 = 20 \text{ k}\Omega \pm 5\%$$

From the voltage divider rule, $V_M$ is related to $V_{DC}$ by

$$V_M = \left( \frac{R_M \parallel R_2}{R_1 + (R_M \parallel R_2)} \right) V_{DC} \quad (1.8)$$

which, for $V_M = 1 \text{ V}$, $V_{DC} = 20 \text{ V}$, $R_1 = 20 \text{ k}\Omega$, and $R_M = 10 \text{ k}\Omega$, gives $R_2 = 1.18 \text{ k}\Omega$. We find that 1.2 k\Omega is the nearest higher value for a 5% carbon resistor. That is, $R_2 = 1.2 \text{ k}\Omega \pm 5\%$.

Step 6. Evaluate your design, and predict its performance. Using Eq. (1.8), we get

$$V_M = \left( \frac{R_M \parallel R_2}{R_1 + (R_M \parallel R_2)} \right) V_{DC} = 1.01695 \text{ V}$$

FIGURE 1.19 Block diagram solution of the design for Example 1.1

FIGURE 1.20 Proposed circuit configuration for Example 1.1
which in turn gives $I_M = 1.01695/10 \, \text{k}\Omega = 102 \, \mu\text{A}$, which falls within the specified criteria.

$$I_{DC} = \frac{(V_{DC} - V_M)}{R_1} = \frac{(20 - 1.01695)}{20 \, \text{k}\Omega} = 949 \, \mu\text{A}$$

This also falls within the specified criteria. The power rating of $R_1$ is

$$P_{R_1} = (V_{DC} - V_M) \times I_{DC} = (20 - 1.01695) \times 949 \, \mu\text{A} = 18 \, \text{mW}$$

We choose the lowest power rating, $1/8$ W. The power rating of $R_2$ is

$$P_{R_2} = \frac{V_M^2}{R_2} = \frac{1.01695^2}{1.2 \, \text{k}\Omega} = 862 \, \mu\text{W}$$

We choose the next larger power rating, $1/8$ W.

**NOTE:** At this step we may need to modify our design because we did not consider the effects of resistor tolerance on performance.

**Step 7.** Simulate the circuit. We will use PSpice to find the meter current. The complete circuit for PSpice simulation is shown in Fig. 1.21. $R_{\text{break}}$ is the model name for resistors to assign tolerance to resistors. Refer to Appendix A, “Introduction to OrCAD,” and to Rashid [6].

The PSpice plot of the meter current $I(R_m)$ versus the DC supply voltage $V_{DC}$ is shown in Fig. 1.22. The design meets the specifications under nominal values, but under worst-case conditions the design falls short of specifications. Fine-tuning and several modifications will be required to find the final solution. This is generally true for open-ended design problems. We could try changing the value of $R_1$ or $R_2$ so that the specifications are met under worst-case conditions. Of course we could meet the specifications easily, but at a higher cost, if we chose resistors with 1% tolerance. The second cursor of the first plot is used to measure the current of the second plot.

**Step 8.** Get a cost estimate. From the resistor cost tables provided by suppliers, we find

- $R_1$: 20 k\,\Omega, $1/8$ W $\quad$ $0.20$ (approx.)
- $R_2$: 1.2 k\,\Omega, $1/8$ W $\quad$ $0.15$ (approx.)

which gives a total component cost of $0.35. Note that this estimate does not take into account costs usually associated with production, manufacturing, or company overhead.
1.9.4 Benefits of Studying from a Design Perspective

On a concrete level, the use of the design process helps you translate complex design tasks into simple circuits in a systematic way. Also, it helps you learn to establish procedures for tackling the system blocks, circuits, or subcircuits and to find, use, and integrate information from various sources, such as manufacturers’ data sheets, modeling, and simulations.

On a more abstract level, the design requires decision making on trade-offs and alternative solutions and challenges you to select the best answer from a large number of acceptable ones. Thus, it strengthens decision-making skills and develops judgment, as well as fosters self-confidence and expertise in applying theory to solve real problems. Also, it lets you solve problems in your own unique way; thus, it motivates and develops creativity as well as critical thinking skills. Creativity is important to the design process, which requires you to go beyond what you learn in classroom lectures.

Because it integrates different topics in electronics as well as material from other courses on basic circuits, physics, mathematics, simulation and modeling, and laboratory techniques, a design perspective emphasizes that broad general knowledge is essential for engineering design. The design at the system level requires knowledge of system methodologies such as analysis and top-down design and system characteristics such as safety and reliability. It also requires communication skills to prepare reports and present data and management skills to coordinate product development and discover why designs fail to meet performance specifications.

1.9.5 Types of Design Projects

Engineering design involves open-ended problems whose objectives are only partially defined. Problem definition and identification of constraints are needed to achieve a satisfactory solution. Often considerable ingenuity is required to find an acceptable solution from alternative pathways, and generally several iterations are needed to reach the solution. Also, it is necessary to verify the solution by simulation and/or testing to ensure that the design objectives have been satisfied. The design projects that appear in this text vary in complexity; the time required to complete them varies from 1 h to 1 month. The design projects can be classified into four categories, depending on the time required: short design projects, mini design projects, medium design projects, and large design projects. Electrical Engineering Design Compendium [7] is an excellent source of other open-ended problems.

Short Design Projects

Short design projects can be completed in 1 or 2 h. Some of the problems at the end of each chapter fall into this category, including the following:

1. Defining the specifications of rectifier circuits (Chapter 5)
2. Defining the specifications of amplifiers (Chapter 2)
3. Design of transistor biasing circuits (Chapters 7 and 8)
4. Design of simple operational amplifier (op-amp) circuits (Chapter 3)
5. Design for offset minimization of op-amp circuits (Chapter 14)
6. Design of simple current source biasing circuits (Chapter 9)
Mini Design Projects

Mini design projects can be completed in approximately 1 week. Following are some of the end-of-chapter problems in the text that fall into this category:

1. Design of half-wave and full-wave rectifiers with an output filter (Chapter 5)
2. Design of diode wave-shaping circuits (Chapter 5)
3. Design of op-amp differentiators or integrators (Chapter 3)
4. Design of power amplifiers (Chapter 11)
5. Design of Schmitt trigger circuits (Chapter 16)
6. Design of sample and hold circuits (Chapter 16)
7. Design of timing circuits (Chapter 16)

Medium Design Projects

Medium design projects can be completed in approximately 2 to 3 weeks. Following are some of the end-of-chapter problems in the text that fall into this category:

1. Design of single-stage transistor amplifiers (Chapters 7 and 8)
2. Design of active filters (Chapter 12)
3. Design of instrumentation amplifiers (Chapter 3)
4. Design of transistor amplifiers to meet frequency specifications (Chapters 7 and 8)
5. Design of single-stage feedback amplifiers (Chapter 10)
6. Design of oscillators (Chapter 13)
7. Design of active current sources (Chapter 9)
8. Design of differential amplifiers with current source biasing (Chapter 9)
9. Design of electronic circuits using A/D and D/A converters (Chapter 15)
10. Design of electronic circuits using ICs of phase-lock loop (PLL) and voltage-controlled oscillators (VCO) (Chapter 16)

Large Design Projects

Large design projects can be completed in approximately 4 to 5 weeks. Following are some of the end-of-chapter problems in the text that fall into this category:

1. Design of multistage amplifiers (Chapters 7 and 8)
2. Design of higher-order active filters (Chapter 12)
3. Design of power amplifiers with current source biasing (Chapter 11)
4. Design of operational amplifiers (Chapter 14)
5. Design of multistage feedback amplifiers (Chapter 10)
6. Design of logic gates (Chapter 15)

1.9.6 Design Report

It is recommended that in your design reports you do the following:

- Give the complete design, including the ratings and values of each component.
- Justify the use of a particular circuit topology.
• Verify your design objectives by simulating your circuit using PSpice/SPICE/OrCAD or Electronics Workbench. Include a worst-case analysis (with 10% tolerances for all passive components, unless specified).
• Give a cost estimate. The project should be as inexpensive as possible.

A suggested format for design reports is as follows:

1. Title page (including your name, the course number, and the date)
   Include the following statement on the cover sheet:
   I/we certify that this assignment is the result of my/our own efforts.
   Signature(s):   Date:
2. Design objectives and specifications
3. Design steps (including the circuit topology)
4. Design modifications
5. Computer simulation and design verification
6. Components and costs
7. Flowchart of the design process
8. Costs versus reliability and safety considerations
9. Conclusions

KEY POINTS OF SECTION 1.9

■ Design is the creative process of developing a solution to an open-ended problem. Use of the engineering design process has many benefits. It develops creativity as well as critical thinking skills, and it promotes decision making and develops judgment.
■ Following the design process helps in translating complex design tasks into simple circuits in a systematic way.

1.10 Electronic Devices

Electronic devices constitute the heart of electronics. The many types of devices can be classified into three categories: semiconductor diodes, bipolar junction transistors (BJTs), and field-effect transistors (FETs). All are nonlinear devices.

1.10.1 Semiconductor Diodes

A diode is a two-terminal semiconductor device. It offers a low resistance in the forward direction and a high resistance in the reverse direction. Thus a diode permits easy current flow in only one direction. The symbol for a diode is shown in Fig. 1.23(a). The arrow indicates the direction of current flow. If the anode–cathode voltage $v_D$ is greater than zero, a diode is like a short circuit; if the voltage $v_D$ is less than zero, the diode is like an open circuit. Thus, a diode is a logic device and can be represented by a controlled switch, as shown in Fig. 1.23(b). We will study the characteristic and modeling of diodes in Chapter 4 and the applications of diodes in Chapter 5.
1.10.2 Bipolar Junction Transistors

Bipolar junction transistors (BJTs), developed in the 1950s, are the oldest devices for amplification of signals. There are two types of transistors: npn and pnp. Their symbols are shown in Fig. 1.24(a) and (b). A BJT has three terminals: the emitter (E), the base (B), and the collector (C). The arrowhead on the emitter identifies the transistor as an npn or a pnp transistor. Voltages \( V_{BE} \) and \( V_{CC} \) are required to activate and bias the transistors appropriately in their normal operating modes.

A BJT is a current-dependent device, and its collector (output) current \( i_C \) depends on the base current \( i_B \), as shown in Fig. 1.24(c). The base emitter behaves like a diode and can be represented by a diode. Thus, a small change in the base current \( i_b \) causes an amplified change in the collector current \( i_C \). That is,

\[
\frac{i_C}{i_b} = \beta_f \quad \text{(1.9)}
\]

where \( \beta_f \) is called the forward current gain of the transistor. The small-signal model of a BJT in response to a small input base current \( i_b \) is shown in Fig. 1.24(d). In Chapter 8, we will study the characteristics and modeling of bipolar transistors.
1.10.3 Field-Effect Transistors

Field-effect transistors (FETs) are the next generation of transistors after BJTs. An FET has three terminals: the drain (D), the gate (G), and the source (S). The output current of an FET depends on an electric field that depends on a gate control voltage. An FET operates as a voltage-dependent device. That is, the drain (output) current depends on the input gate voltage. There are three types of FETs: enhancement metal oxide semiconductor field-effect transistors (enhancement MOSFETs), depletion metal oxide semiconductor field-effect transistors (depletion MOSFETs), and junction field-effect transistors (JFETs). In Chapter 7, we will study the characteristics and modeling of FETs.

Enhancement MOSFETs

There are two types of enhancement MOSFETs: n-channel and p-channel. Their symbols are shown in Fig. 1.25 [(a) and (b)]. The arrowhead on the substrate indicates the type, either p or n. The substrate B is normally connected to the source terminal. A channel is induced under the influence of electric field action. As shown by the break in the lines from the drain to the source, there is no physical channel between the drain and the source. Voltages $v_{GS} = V_{GS}$ and $V_{DD}$ are required to activate and bias the FETs appropriately in their normal operating modes. The gate current $i_G$ is very small, practically zero. The drain (output) current $i_D$ depends on the gate–source voltage $v_{GS}$, as shown in Fig. 1.25(c), and it is given by

$$i_D = K_p(v_{GS} - V_t)^2 \quad \text{for } |v_{GS}| \geq |V_t|$$

(1.10)

where $K_p$ is the MOSFET constant, in ampere per square volts $V_t$ is the threshold voltage of the MOSFET, in volts. $v_{GS}$ and $V_t$ are positive for n-type enhancement MOSFETs and negative for p-type enhancement MOSFETs. The value of $v_{GS}$ must exceed the value of $V_t$ for any drain current to flow. That is,
|v_{GS}| > |V_t|. For example, if \( K_p = 2 \text{ mA/V}^2 \), \( V_t = 1.5 \text{ V} \) (for an n-channel MOSFET), and \( v_{GS} = 3 \text{ V} \), Eq. (1.10) gives

\[
i_D = \frac{2 \text{ mA}}{V^2} \times (3 - 1.5)^2 \text{ V}^2 = 4.5 \text{ mA}
\]

**Depletion MOSFETs**

There are two types of depletion MOSFETs: n-channel and p-channel. Their symbols are shown in Fig. 1.26[(a) and (b)]. As shown by the continuous lines from the drain to the source, there is a physical channel between the drain and the source. However, the channel can be enhanced or depleted by the influence of electric field action. Voltages \( v_{GS} (= V_{GS}) \) and \( V_{DD} \) bias the FETs appropriately in their normal operating modes. The gate current \( i_G \) is practically zero. The drain (output) current \( i_D \) depends on the gate–source voltage \( v_{GS} \), as shown in Fig. 1.26(c), and it is given by

\[
i_D = K_p(v_{GS} - V_p)^2
\]

\[
i_D = I_{DSS} \left(1 - \frac{v_{GS}}{V_p}\right)^2
\]  

where

\( K_p = \text{MOSFET constant, A/V}^2 \)

\( I_{DSS} = \frac{K_p}{V_p^2} \), the steady-state drain current at \( v_{GS} = 0 \), A

\( V_p = \text{pinch-off voltage of the MOSFET, V} \)

\( V_p \) is the voltage at which the drain–source channel is effectively pinched off and no drain current flows. \( V_p \) is negative for n-type depletion MOSFETs and positive for p-type depletion MOSFETs. \( v_{GS} \) can be either positive or negative, but its magnitude cannot exceed \( |V_p| \). That is, \( v_{GS} > V_p \) for n-channel and \( v_{GS} > -V_p \) for p-channel. For example, if \( I_{DSS} = 10 \text{ mA} \), \( v_{GS} = -2 \text{ V} \) and \( V_p = -4 \text{ V} \), Eq. (1.12) gives

\[
i_D = 10 \text{ mA} \times \left(1 - \frac{2}{4}\right)^2 = 2.5 \text{ mA}
\]

**FIGURE 1.26** Depletion MOSFETs
Junction FETs

Depletion MOSFETs can perform the same functions as JFETs [8] and have advantages over JFETs. As a result, JFETs are being phased out and are not covered further in this text. There are two types of JFETs: $n$-channel and $p$-channel. Their symbols are shown in Fig. 1.27(a) and (b). The gate–source junction is reverse biased so that it behaves as a reverse-biased diode. As shown by the continuous lines from the drain to the source, there is a physical channel between the drain and the source. The drain current is controlled by the influence of electric field action. Voltages $v_{GS} (= V_{GS})$ and $V_{DD}$ bias the JFETs appropriately in their normal operating modes. There is a small gate current $i_G$ on the order of microamperes. The drain (output) current $i_D$ depends on the gate–source voltage $v_{GS}$, as shown in Fig. 1.27(c), and it is given by

$$i_D = I_{DSS} \left(1 - \frac{v_{GS}}{V_p}\right)^2 \text{ for } |v_{GS}| \leq |V_p|$$

where $I_{DSS}$ is drain current at $v_{GS} = 0$, in amperes and $V_p$ is pinch-off voltage of the JFET, in volts. $V_p$ is the voltage at which the drain–source channel is effectively pinched off and no drain current flows. $V_p$ is negative for $n$-type JFETs and positive for $p$-type JFETs. $v_{GS}$ is negative for $n$-type JFETs and positive for $p$-type JFETs, but its magnitude cannot exceed $V_p$. For example, if $I_{DSS} = 20 \text{ mA}$, $V_p = -3 \text{ V}$ (for an $n$-channel JFET), and $v_{GS} = -1.5 \text{ V}$, Eq. (1.13) gives

$$i_D = 20 \text{ mA} \times (1 - 1.5/3)^2 = 5 \text{ mA}$$

KEY POINTS OF SECTION 1.10

- Electronic devices constitute the heart of electronics. There are three categories of devices: semiconductor diodes, bipolar junction transistors (BJTs), and field-effect transistors (FETs).
- A diode acts as a switch, which may be on or off depending on the voltage across its terminals. A BJT is a current-dependent device that can be operated as a switch or as an amplifying device. An FET is a voltage-dependent device that can be operated as a switch or as an amplifying device.
1.11 Emerging Electronics

Traditional electronics use inorganic conductors. The inorganic materials such as silicon, gallium arsenide semiconductors, silicon dioxide insulators, and metals such as aluminum and copper have been the backbone of the modern semiconductor industry. Organic materials are being used as conductors in organic and biomedical electronics. A new type of element known as memristor may lead to the invention of a new class of high-density devices.

1.11.1 Memristor

From the circuit point of view, the circuit elements are defined in terms of four circuit variables: the current \( i \), the voltage \( v \), the charge \( q \), and the magnetic flux \( \phi \). Therefore, there are four possible combinations of these variables: \( i \) and \( v \), \( v \) and \( q \), \( q \) and \( \phi \), and \( \phi \) and \( i \). Three of these relationships describe the three basic circuit elements: the resistance \( R \), the inductance \( L \), and the capacitance \( C \). The fourth relationship between \( q \) and \( \phi \) was discovered and described by Chua [9] as the fourth circuit element, known as memristor. Thus, the four circuit elements are related to the circuit variables by

\[
\text{Resistance: } R(t) = \frac{dv}{di} \tag{1.14}
\]

\[
\text{Inductance: } L(i) = \frac{d\phi}{di} \tag{1.15}
\]

\[
\text{Capacitance: } C(q) = \frac{dq}{dv} \tag{1.16}
\]

\[
\text{Memristance: } M(q) = \frac{d\phi}{dq} \tag{1.17}
\]

If the charge \( q \) and the flux \( \phi \) vary with the time \( t \), the current \( i \) and the voltage \( v \) are given by

\[
\text{Current: } i(t) = \frac{dq}{dt} \tag{1.18}
\]

\[
\text{Voltage: } v(t) = \frac{d\phi}{dt} \tag{1.19}
\]

Therefore, if the charge \( q \) and the flux \( \phi \) vary, the ratio of \( v(t) \) and \( i(t) \) gives the memristance \( M(q) = v(t)i(t) = d\phi/dq \). The memristor is a circuit element in which the magnetic flux \( \phi \) is a function of the accumulated charge; its symbol is shown in Fig. 1.28(a). The memristance is the rate of change of flux
with the charge as shown in Fig. 1.28(b). Since \( M(q) \) has the unit of resistance, it is the incremental memristance whose value at any time \( t_0 \) depends on the time integral of the memristor current from \( t = - \infty \) to \( t = t_0 \).

Thus, a memristor behaves like an ordinary resistor at any given instant of time \( t_0 \), but its resistance depends on the history of its current or its accumulated charge that has flowed through the device in the same way that the voltage of capacitors does. The name memristor is a portmanteau of memory resistor. A memristor (one for which \( M \) varies) is a nonlinear resistor. A linear memristor (one for which \( M \) is constant) would be similar to an ordinary linear resistor (one for which \( R \) is constant), with \( M = R \). It is important to note that unlike a capacitor that stores charge, a memristor does not use magnetic flux to achieve its nonlinear characteristic; rather it achieves the characteristic of a resistor whose value depends on the history of the current using the chemical mechanism of charge changes.

A memristor is capable of “remembering” how much electrical charge most recently passed through it in a particular direction, which is reversible by changing the direction of current [10]. The memristor has unique analog properties that may lead to the invention of a new class of high-density devices having the characteristics of low heat generation. These include applications in high-density nonvolatile digital memory [10, 11, 12], programmable logic, signal processing, neural networks, and control systems.

### 1.11.2 Organic Electronics

There has been a growing research effort in “organic electronics” to improve the semiconducting, conducting, and light-emitting properties of organics (polymers, oligomers) and hybrids (organic–inorganic composites) [13, 14]. Organic electronics, also called plastic electronics, use organic material that has semiconductor properties [15]. A semiconductor is any compound whose electrical conductivity is between that of typical metals and that of insulating compounds. Carbon-based conductive polymers and molecules, which are similar to the molecules of living cells, are used as conductors exhibiting semiconductor properties. Single-molecule organic semiconductors are of two types: short-chain (oligomers) and long-chain (polymers). Like inorganic semiconductors, organic semiconductors can be doped to obtain an electron donor molecule or an electron acceptor molecule.

Conductive polymers are lighter, more flexible, and less expensive than inorganic conductors. They are finding new applications as smart windows and electronic papers, and they are expected to play an important role in the emerging area of molecular computers. Conductive polymers, however, have a higher resistance and therefore conduct electricity poorly and inefficiently compared to inorganic conductors. The organic materials, which are used as either sacrificial stencils (photoresists) or passive insulators, do not take any active role in the electronic functioning of a device. They neither conduct current to act as switches or wires nor emit light. Organic electronics fall broadly into two categories:

1. **Organic thin-film transistors (OTFTs)** [16]: These offer unique advantages over inorganic transistors and are cheaper to produce. They find applications such as a flat screen that can be rolled up and put into a jacket pocket. They can be constructed on flexible surfaces, such as plastic film, making organic circuits ideal for portable and mobile devices. However, organic transistors have a big disadvantage of much higher energy consumption. OTFTs also have low charge mobilities, which limit their operation at high switching frequencies.

2. **Organic light-emitting diodes (OLEDs)**: These are breaking into the consumer electronics market and replacing the small liquid crystal displays (LCDs) [17] found in music players, cameras, and mobile phones [18]. The production of flexible OLED screens will allow roll-up screens for easy transport of unique, large-screen personal digital assistants (PDAs) [19] and flexible and impact-resistant display units.
Here are some advantages of organic electronics:

- Organic electronics have photoconductive properties, which makes them suitable in applications such as liquid crystal displays, light-emitting diodes, solar cells, and even superconductors.
- They are lighter, more flexible, and cheaper to manufacture than inorganic electronics—for example, in making radio frequency identification (RFID) tags.
- Environment-polluting chemicals are not needed in manufacturing organic circuits because the circuits can be printed from organic “ink.” Silicon components, in contrast, require a long, tedious manufacturing process that requires dangerous chemicals such as arsenic, lead, and mercury, along with millions of gallons of water and solvent.

The disadvantages of organic electronics include the following:

- Their major disadvantage is their relatively higher energy consumption.
- The mobility of their electrons is much lower than that of silicon. This affects the speed at which the transistors or other devices can switch on and off.

After the disadvantages are overcome, organic electronics are expected to take over a major share of the silicon-based component marketplace.

1.11.3 Bioelectronics

Electronic principles find applications in many biomedical devices. Bioelectronics use biomaterials as conductors and apply the principle of intermolecular electron transfer in physiological processes. Biocompatible materials are used to construct artificial organs, rehabilitation devices, or prostheses and replace natural body tissues. Materials that are used and adapted for medical applications are essentially biomaterials. Therefore, a biomaterial is any material, natural or human-made, that makes up all or part of a living structure or biomedical device that performs, augments, or replaces a natural function. Applications include but are not limited to cartilage replacement, bone plates, organ replacement, blood vessel prostheses, skin repair devices, and contact lenses.

There are many different types of bioelectronics, depending on the types of applications. These include sensors that can distribute medications and supplements according to the user’s needs, nanowires capable of sending information and power, biofuel cells capable of producing reasonable power, solar energy, bioelectronic noses, medical devices, prostheses, and electronic biosensors. An implant biosensor can restore partial sight.

Nanowires [20] could yield optical chips and sensors for biological and chemical molecules. The wires are about a thousandth the width of a human hair and function with minimal signal loss. Bioelectronic principles include concepts for curing diseases, DNA-based nanofabrication, fabricating nanoelectronic devices using self-assembling protein molecules, cell restoration, organ prostheses, and nanoelectronic circuitry.

Some applications and terminology of bioelectronics fall broadly into 1 of 12 categories:

1. Biotechnology [21] uses microorganisms, such as bacteria or yeasts, or biological substances, such as enzymes, to perform specific industrial or manufacturing processes.
2. Nanotechnology [22] refers to a field of applied science concerning the control of matter and construction of devices on the atomic and molecular scale. Nanofabrication based on DNA architecture can synthesize and program reliably self-organizing building blocks for any desired structure—such as metals, semiconductor nanoparticles, carbon nanotubes, and biomolecules.
3. **Nanomedicine** is a specific application of nanotechnology. Nanomedicine is being used for drug delivery: Medicines are delivered by basic nanites that can identify particular cells that need a medicine and release the drug into the cells. This greatly increases the efficiency of the medication. Nanotechnology is also currently used to augment the quality of MRI scans by providing greater contrast with nanoparticle agents. Nanites can help identify certain cancers and can sometimes destroy the cancers at a cellular level. Nanoscopic biosensors can be injected to provide medical personnel more complete information about patients [22].

4. **Biosensors** [23–26] are devices that detect, record, and transmit information about a physiological change or process. They use biological materials to monitor the presence of various chemicals in a substance. A biosensor consists of three elements: the sensitive biological element, the transducer or the detector element, and the user interface. Blood glucose sensors are the most widespread example of a commercial biosensor. Other instances of biosensor development include electronic noses [24] and eyes [27].

5. **Medical telesensors** [28] are fingertip chips that measure and transmit data about human body temperature, blood pressure, oxygen level, and pulse rate. This type of medical telesensor can report measurements of vital functions to remote recorders.

6. **Microcantilevers** can measure the presence of substances by nonoptical methods and are alternatives to optical fiber. They can act as physical, chemical, or biological sensors by detecting changes in cantilever bending or vibrational frequency. Microcantilevers are a million times smaller than the molecules they detect; but molecules adsorbed on a microcantilever cause vibrational frequency changes. Viscosity, density, and flow rate can also be measured through these changes in vibrational frequency.

7. **Bioreporters** are a type of biosensor based on detection of light emitted by a specially engineered microorganism that is involved in bioremediation. The light originates from a particular protein that has been installed in certain bacteria by modern molecular genetic methods. This type of biosensor can find application within the fields of pollution cleanup and detection and reporting of biodegradation.

8. **Microchip implants** [23] in animal or human brains are linked to a communication device on a computer. This process, called brain–computer interface, is linked directly to the neural part of the brain. The computer can either send or receive signals from the microchip—but not both. The computer can send signals to the brain to help control and improve eyesight, hearing, or movement. The microchip can also send signals to a computer system to warn of potential health problems of an animal or human.

9. **Biofuel cells** [23] are microorganisms when consumed by a substrate such as sugar in an aerobic condition, that is, a solution involving air, usually oxygen, produce carbon dioxide and water. However, when oxygen is not present, they produce electrons, thus providing current for power generation. Therefore, a biofuel cell uses living organisms to produce electricity. Two types are microbial fuel cells and enzymatic biofuel cells. Microbial fuel cells use living microorganisms. Enzymatic biofuel cells use enzymes derived from organisms.

10. **Microbial fuel cells** [23, 29] convert chemical energy to electrical energy by the catalytic reaction of microorganisms. They have a number of potential uses. The most obvious is harvesting the electricity produced; virtually any organic material could be used to “feed” the fuel cells. Microorganisms known as *electricigens* can efficiently convert organic wastes, renewable biomass, and even mud into electricity and harmless by-products. This capability offers the potential for using microbes (or their components) to generate electricity at low cost while transforming industrial, domestic, and farm wastes.

11. **Nanobiotechnology** refers to the use of nanotechnology to further the goals of biotechnology.

12. **Nanofabrication** refers to any technique used to create objects or mechanisms on the scale of nanotechnology.
References


Problems

1.1 Design a circuit to measure a DC voltage in the range from 0 to 400 V. For a full-scale deflection, the indicating meter draws 100 μA at a voltage of 1 V (DC) across it. The accuracy should be better than 2%. The input current from the source should be less than 1 mA.

1.2 Design a circuit to supply 6 V to a load from a DC supply of 24 V. The load current should be 5 A. The accuracy should be better than 5%.

1.3 Design a circuit to provide 60 W at 50 V to a resistive lamp from an AC supply of 120 V ± 10% at 60 Hz. The circuit should be energy efficient—that is, it should consume the least power possible.

1.4 Design a circuit to charge a 2-μF capacitor from a DC supply of 24 V. The charging current should be limited to 1 mA. The accuracy should be better than 5%.

1.5 The input and output voltages of an amplifier are

\[ v_i = 5 \sin(1000 \pi t + 30^\circ) \text{ mV} \quad \text{and} \quad v_o = 400 \sin(1000 \pi t + 90^\circ) \text{ mV} \]

Find the magnitude and phase of the voltage gain of the amplifier.

1.6 An n-channel MOSFET has \( K_p = 20 \text{ mA/V}^2 \) and \( V_t = 1.5 \text{ V} \). If the gate–source voltage is \( v_{GS} = 3 \text{ V} \), find the small-signal transconductance \( g_m \) of the MOSFET.

\[
g_m = \frac{dI_D}{dV_{GS}}
\]

1.7 An n-channel JFET has \( I_{DSS} = 20 \text{ mA} \) and \( V_p = 23 \text{ V} \). If the gate–source voltage is \( v_{GS} = -1.5 \text{ V} \), find the small-signal transconductance \( g_m \) of the JFET. Assume the devices are in saturation.

\[
g_m = \frac{dI_D}{dV_{GS}}
\]

1.8 The base current of a bipolar transistor is \( i_B = 2(1 + \sin 2000 \pi t) \text{ mA} \) and the current gain of the transistor is \( \beta_F = 100 \). What are \( I_B, i_b, I_C, \) and \( i_c \)?

1.9 For a bipolar transistor, the collector–emitter voltages are \( V_{CE} = 6 \text{ V} \) and \( v_{ce} = -100 \sin(2000 \pi t) \text{ mV} \), and the base–emitter voltages are \( V_{BE} = 0.7 \text{ V} \) and \( v_{be} = 1 \sin(2000 \pi t) \text{ mV} \).

a. What are the expressions for \( v_{CE} \) and \( v_{BE} \)?

b. Find the small-signal voltage gain \( A_v \).

1.10 For an FET, the drain–source voltages are \( V_{DS} = 6 \text{ V} \) and \( v_{ds} = -50 \sin(1000 \pi t) \text{ mV} \), and the gate–source voltages are \( V_{GS} = 3 \text{ V} \) and \( v_{gs} = 2 \sin(1000\pi t) \text{ mV} \).

a. What are the expressions for \( v_{DS} \) and \( v_{GS} \)?

b. Find the small-signal voltage gain \( A_v \).
Learning Outcomes

After completing this chapter, students should be able to do the following:

- Describe the characteristics, types, circuit models, and applications of amplifiers.
- Determine the design specifications of an amplifier in order to meet the input and output requirements of an application.
- Describe the frequency behavior of amplifiers.
- Apply the frequency response techniques to determine the low and high cutoff frequencies of amplifiers.
- Describe the external characteristics and circuit models of op-amps.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_v$, $G_m$, $Z_m$, $A_i$</td>
<td>Voltage, transconductance, transimpedance, and current gain</td>
</tr>
<tr>
<td>$A_{vo}$, $G_{mo}$, $Z_{mo}$, $A_{is}$</td>
<td>Open-circuit voltage, transconductance, transimpedance, and current gain</td>
</tr>
</tbody>
</table>
2.1 Introduction

The output signals from transducers are weak (in the range of microvolts [μV] or millivolts [mV]) and possess a very small amount of energy. These signals are generally too small to be processed reliably to perform any useful function. Signal processing is much easier if the magnitude of the signal is large (in the range of volts). Amplifiers are used in almost every electronic system to increase the strength of a weak signal. An amplifier consists of one or more amplifying devices. The complexity of an amplifier depends on the number of amplifying devices. To analyze a complex circuit consisting of a number of amplifiers, a model representing the terminal behavior of each amplifier is often necessary.

2.2 Amplifier Characteristics

An amplifier may be considered as a two-port network with an input port and an output port. It is represented by the circuit symbol shown in Fig. 2.1, which indicates the direction of signal flow from the input side to the output side. Normally, one of the input terminals is connected to one of the output terminals to form a common ground. The output voltage (or current) is related to the input voltage (or current) by a gain parameter. If the output signal is directly proportional to the input signal, the amplifier is said to be a linear amplifier. If there is any change in the output waveform, it is considered to have distortion, which is undesirable. The amplifier is then said to

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>BW, $A_{PB}$</td>
<td>Bandwidth and pass-band voltage gain</td>
</tr>
<tr>
<td>CMRR</td>
<td>Common-mode rejection ratio</td>
</tr>
<tr>
<td>$f_H$, $f_L$</td>
<td>High and low cutoff frequencies</td>
</tr>
<tr>
<td>$i_s$, $i_i$, $i_o$</td>
<td>Small-signal source and input and output currents</td>
</tr>
<tr>
<td>$R_i$, $R_o$</td>
<td>Input and output resistances</td>
</tr>
<tr>
<td>$R_{CI}$, $τ_{CI}$</td>
<td>Equivalent resistance of an RC circuit and the RC time constant</td>
</tr>
<tr>
<td>$v_s(t)$, $v_i(t)$, $v_o(t)$</td>
<td>Instantaneous input signal and input and output voltages</td>
</tr>
<tr>
<td>$V_s$, $V_o$</td>
<td>DC input signal and output voltages</td>
</tr>
</tbody>
</table>

*Figure 2.1  Symbol for an amplifier*
be a **nonlinear amplifier**. The characteristics of amplifiers are defined by a number of parameters, which are described in the sections that follow.

### 2.2.1 Voltage Gain

If the input signal voltage to a linear amplifier is $v_S$, the amplifier will provide an output voltage $v_O$, which will be a magnified facsimile of $v_S$. This situation is shown in Fig. 2.2(a) for an amplifier with a load resistance of $R_L$. The *voltage gain* $A_V$ of the amplifier is defined by

$$ A_V = \frac{v_O}{v_S} $$

The transfer characteristic, shown in Fig. 2.2(b), will be a straight line with a slope of $A_V$. Thus, if we apply a DC input signal of $v_S = V_S$, the DC output voltage will be $v_O = V_O = A_V V_S$ and the amplifier will operate at point $Q$. The DC voltage gain then becomes $A_V = \frac{V_O}{V_S}$. However, if we superimpose a small sinusoidal signal $v_s = V_m \sin \omega t$ on $V_S$, as shown in Fig. 2.2(c), the output voltage becomes $v_O = V_O + v_o$. 

---

**FIGURE 2.2** Voltage amplifier

---

---
The small-signal AC voltage gain becomes $A_v = \frac{\Delta v_o}{\Delta v_s} = \frac{v_o}{v_s}$. Thus, a small-signal input voltage $v_s = V_m \sin \omega t$ will give a corresponding small-signal output voltage $v_o = A_v V_m \sin \omega t$ such that $v_o = A_v V_s + A_i v_s = V_o + A_v V_m \sin \omega t$. This is shown in Fig. 2.2(d). Therefore, we face two voltage gains: a DC gain and a small-signal gain. For a linear amplifier, the two gains are equal. That is, $A_V = A_v$, and the small-signal gain is referred to simply as the voltage gain.

### 2.2.2 Current Gain

If $i_s$ is the current the amplifier draws from the signal source and $i_O$ is the current the amplifier delivers to the load $R_L$, then the current gain $A_i$ of the amplifier is defined by

$$A_i = \frac{\text{Load current } i_O}{\text{Input current } i_S} (A/A)$$

The transfer characteristic will be similar to that shown in Fig. 2.2(b). For a linear amplifier, the DC gain equals the small-signal current gain: $A_i = \frac{\Delta i_O}{\Delta i_S} = \frac{i_o}{i_s}$. That is, $A_i = A_i$, and the small-signal gain is referred to simply as the current gain.

### 2.2.3 Power Gain

An amplifier provides the load with greater power than it receives from the signal source. Thus, an amplifier has a power gain $A_p$, which is defined by

$$A_p = \frac{\text{Load power } P_L}{\text{Input power } P_i}$$

$$= \frac{v_o^2}{v_s^2} (W/W)$$

After substitution of $A_v = \frac{v_o}{v_s}$ and $A_i = \frac{i_o}{i_s}$, Eq. (2.4) can be written as

$$A_p = A_v A_i$$

Thus, the power gain is the product of the voltage gain and the current gain.

### 2.2.4 Logarithmic Gain

The gains of amplifiers can be expressed either as dimensionless quantities or with units ($V/V$ for a voltage gain, $A/A$ for a current gain, or $W/W$ for a power gain). Their values are usually very large and extend over several orders of magnitude. It is not convenient to plot such large numbers against other parameters. Gains are normally expressed in terms of logarithms, as follows:

$$\text{Power gain in decibels (dB) } = 10 \log A_p = 10 \log_{10} \left( \frac{P_L}{P_i} \right) = 10 \log_{10} \left( \frac{v_o^2}{R_L} \right)$$

$$= 20 \log_{10} \left( \frac{v_o}{v_s} \right) + 10 \log_{10} \left( \frac{R_i}{R_L} \right)$$
where $R_i$ is the input resistance of the amplifier, which is seen as a load by the signal source $v_s$. The term $20 \log_{10} (v_o/v_s)$ is referred to as the voltage gain of the amplifier in decibels. That is,

$$\text{Voltage gain in dB} = 20 \log |A_v| \quad \text{for } R_i = R_L$$

The power gain can also be expressed in terms of the input and output current:

$$\text{Power gain in dB} = 10 \log A_P = 10 \log_{10} \left( \frac{P_L}{P_i} \right) = 10 \log_{10} \left( \frac{i_o^2 R_L}{i_s^2 R_i} \right)$$

$$= 20 \log_{10} \left( \frac{i_o}{i_s} \right) + 10 \log_{10} \left( \frac{R_L}{R_i} \right)$$

The term $20 \log_{10} (i_o/i_s)$ is referred to as the current gain of the amplifier in decibels. That is,

$$\text{Current gain in dB} = 20 \log |A_i|$$

If $R_i = R_L$, the power gain in decibels is equal to the voltage and current gains in decibels. That is,

$$\text{Power gain in dB} = \text{Voltage gain in dB} = \text{Current gain in dB}$$

Some amplifiers, such as operational amplifiers (op-amps), have a very high voltage gain, which is quoted in decibels. For example, rather than writing $A_v = 10^5 \text{ V/V}$, it is common to write 100 dB, which equals $20 \log 10^5$.

### NOTES

1. If there is a phase difference of $180^\circ$ between the input and output voltages (or currents), the voltage gain $A_v$ (or current gain $A_i$) will be negative. Therefore, the absolute value of $A_v$ (or $A_i$) must be used for calculating the gain in decibels. However, the power gain $A_p$ is always positive.

2. If the absolute value of the voltage (or current) gain is less than 1, the output is said to be attenuated rather than amplified, and the gain in decibels will be negative.

#### 2.2.5 Input and Output Resistances

**Input resistance** $R_i$ is a measure of the current drawn by the amplifier. It is a ratio of the input signal voltage to the input current:

$$R_i = \frac{v_s}{i_s} \quad (2.6)$$

**Output resistance** $R_o$ is the internal resistance seen from the output terminals of an amplifier—that is, Thevenin’s equivalent resistance.

#### 2.2.6 Amplifier Saturation

An amplifier needs a DC power supply (or supplies) so that an operating $Q$-point can be established, as shown in Fig. 2.2(b), that allows variation in the output signal in response to a small change in the input signal. The DC supply (or supplies) provides the power delivered to the load, as well as any power that is dissipated as heat within the amplifier itself. An amplifier with two power supplies, $V_{CC}$ and $V_{EE}$, is shown in
Fig. 2.3(a). $I_{CC}$ and $I_{EE}$ are the currents drawn from the DC supplies $V_{CC}$ and $V_{EE}$, respectively. Terminal A is connected to the positive side of the DC source $V_{CC}$, and terminal B is connected to the negative side of the DC source $V_{EE}$. The output voltage of the amplifier cannot exceed the positive saturation limit $V_{O,max}$ and cannot decrease below the negative saturation limit $-V_{O,min}$.

Each of the two saturation limits is usually within 1 V or 2 V of the corresponding power supply. This fact is a consequence of the internal circuity of the amplifiers and the nonlinear behavior of the amplifying devices. Therefore, to avoid distortion of the output voltage as shown in Fig. 2.3(b), the input voltage must be kept within the range defined by

$$-V_{O,min} \leq v_s \leq V_{O,max}$$

As long as the amplifier operates with the saturation limits, the voltage gain can normally be assumed to be linear. The power delivered by the DC supplies will be

$$P_{dc} = V_{CC}I_{CC} + V_{EE}I_{EE}$$

and the power delivered $P_i$ by the input signal will be small compared to $P_{dc}$. Therefore, the efficiency $\eta$ of an amplifier is defined by

$$\text{Amplifier efficiency } \eta = \frac{\text{Load power } P_L}{\text{Power delivered by DC supplies } P_{dc}}$$

The efficiency of an amplifier ranges from 25% to 80%, depending on the type of amplifier. For amplifiers with a very low input signal (millivolts or microvolts), the voltage gain rather than the efficiency is the prime consideration. On the other hand, for power amplifiers (covered in Chapter 11), efficiency is the major consideration because the amplifier should supply the maximum power to the load (such as the speakers of an audio amplifier).
**EXAMPLE 2.1**

**Finding amplifier parameters** The measured small-signal values of the linear amplifier in Fig. 2.3(a) are \( v_s = 20 \sin 400t \) (mV), \( i_s = 1 \sin 400t \) (\( \mu \)A), \( v_o = 7.5 \sin 400t \) (V), and \( R_L = 0.5 \) k\( \Omega \). The DC values are \( V_{CC} = V_{EE} = 12 \) V and \( I_{CC} = I_{EE} = 10 \) mA. Find (a) the values of amplifier parameters \( A_v, A_i, A_p, \) and \( R_i \); (b) the power delivered by DC supplies \( P_{dc} \) and the power efficiency \( \eta \); and (c) the maximum value of the input voltage so that the amplifier operates within the saturation limits.

**SOLUTION**

\[ v_{s(peak)} = 20 \text{ mV}, \quad v_{o(peak)} = 7.5 \text{ V}, \text{ and } i_{s(peak)} = 1 \text{ \( \mu \)A}. \]

(a) The load current is

\[ i_o = \frac{v_o}{R_L} = \frac{7.5 \sin 400t \text{ (V)}}{0.5 \text{ k}\Omega} = 15 \times 10^{-3} \sin 400t = 15 \sin 400t \text{ (mA)} \]

The voltage gain is

\[ A_v = \frac{v_{o(peak)}}{v_{s(peak)}} = \frac{7.5 \text{ V}}{20 \text{ mV}} = 375 \text{ V/V} \quad [\text{or } 20 \log (375) = 51.48 \text{ dB}] \]

The current gain is

\[ A_i = \frac{i_{o(peak)}}{i_{s(peak)}} = \frac{15 \text{ mA}}{1 \text{ \( \mu \)A}} = 15 \text{ kA/A} \quad [\text{or } 20 \log (15 \text{ k}) = 83.52 \text{ dB}] \]

The power gain is

\[ A_p = A_v A_i = 375 \times 15 \text{ k} = 5625 \text{ kW/W} \quad [\text{or } 10 \log (5625 \text{ k}) = 67.5 \text{ dB}] \]

The input resistance is

\[ R_i = \frac{v_{s(peak)}}{i_{s(peak)}} = \frac{20 \text{ mV}}{1 \text{ \( \mu \)A}} = 20 \text{ k}\Omega \]

(b) The power delivered by the DC supplies is

\[ P_{dc} = V_{CC}I_{CC} + V_{EE}I_{EE} = 2 \times 12 \text{ V} \times 10 \text{ mA} = 240 \text{ mW} \]

The load power is

\[ P_L = \left( \frac{v_{o(peak)}}{\sqrt{2}} \right) \left( \frac{i_{o(peak)}}{\sqrt{2}} \right) = \left( \frac{7.5 \text{ V}}{\sqrt{2}} \right) \left( \frac{15 \text{ mA}}{\sqrt{2}} \right) = 56.25 \text{ mW} \]

By using \( \sqrt{2} \) factor for converting a peak value to a rms value, the input power is

\[ P_i = \left( \frac{v_{s(peak)}}{\sqrt{2}} \right) \left( \frac{i_{s(peak)}}{\sqrt{2}} \right) = \left( \frac{20 \text{ mV}}{\sqrt{2}} \right) \left( \frac{1 \text{ \( \mu \)A}}{\sqrt{2}} \right) = 10 \text{ mW} \]

The power efficiency is

\[ \eta = \frac{P_L}{P_{dc} + P_i} = \frac{56.25 \text{ mW}}{250 \text{ mW}} = 22.5\% \]
2.2.7 Amplifier Nonlinearity

Practical amplifiers exhibit a nonlinear characteristic, which is caused by nonlinear devices such as transistors (discussed in Chapters 7 and 8). For the amplifier shown in Fig. 2.4(a) with one DC supply, its nonlinear characteristic is shown in Fig. 2.4(b). Fortunately there is a region in the midrange of the output voltage where the gain remains almost constant. If the amplifier can be made to operate in this region, a small variation in the input voltage will cause an almost linear variation in the output voltage, and the gain will remain approximately constant. This goal is accomplished by biasing the amplifier to operate at a quiescent point, generally called the $Q$-point, having a DC input voltage $V_S$ and a corresponding DC output voltage $V_O$. If a small instantaneous input voltage $v_s(t) = V_m \sin \omega t$ is superimposed on the DC input voltage $V_S$, as shown in Fig. 2.4(b), the total instantaneous input voltage becomes

$$v_s(t) = V_S + v_s(t) = V_S + V_m \sin \omega t$$

which will cause the operating point to move up and down along the transfer characteristic around the $Q$-point. This movement will cause a corresponding time-varying output voltage

$$v_O(t) = V_O + v_o(t)$$

If $v_s(t)$ is sufficiently small, then $v_o(t)$ will be directly proportional to $v_s(t)$; so

$$v_o(t) = A_v v_s(t) = A_v V_m \sin \omega t$$

where $A_v$ is the slope of the transfer characteristic at the $Q$-point. That is,

$$A_v = \left. \frac{dv_O}{dv_S} \right|_{Q\text{-point}}$$

(c) Since

$$A_v v_s(max) = v_o(max) = V_{CC} = V_{EE} \quad \text{or} \quad v_s(max) = \frac{12V}{375} = 32 \text{ mV}$$

the limit of the maximum input voltage is $0 \leq v_{s(max)} \leq 32 \text{ mV}$, and the limit of the minimum input voltage is $v_{s(min)} = -v_{s(max)} = -32 \text{ mV}$.

![Figure 2.4](image)

**FIGURE 2.4** Amplifier nonlinearity
Therefore, as long as the input signal is kept sufficiently small, the amplifier will exhibit an almost linear characteristic. However, increasing the magnitude of the input signal is expected to cause distortion of the output voltage and may even cause saturation. $A_v$ is known as the *small-signal voltage gain* (or simply the *voltage gain*) of the amplifier; it should not be confused with the *DC gain*, which is defined by

$$A_{dc} = A_V = \frac{v_O}{v_S} \bigg|_{\text{at Q-point}} = \frac{V_O}{V_S} \quad (2.11)$$

Thus, we can conclude that the analysis and the design of a nonlinear amplifier involve two signals: a DC signal and an AC signal. However, the characteristics of an amplifier are described by its behavior in response to a small AC input signal.

**NOTE** In practical amplifiers, the Q-point is set internally and the amplifiers operate from a small input signal. The input signal $v_s$ is superimposed on the Q-point (which consists of $V_O$ and $V_S$) to produce a small-signal output voltage $v_o$.

### EXAMPLE 2.2

**Finding the limiting parameters of a nonlinear amplifier** The measured values of the nonlinear amplifier in Fig. 2.4(a) are $v_O = 4.3 \text{ V}$ at $v_S = 18 \text{ mV}$, $v_O = 5 \text{ V}$ at $v_S = 20 \text{ mV}$, and $v_O = 5.8 \text{ V}$ at $v_S = 22 \text{ mV}$. The DC supply voltage is $V_{CC} = 9 \text{ V}$, and the saturation limits are $2 \text{ V} \leq v_O \leq 8 \text{ V}$.

(a) Determine the small-signal voltage gain $A_v$.

(b) Determine the DC voltage gain $A_{dc}$.

(c) Determine the limits of input voltage $v_S$.

**SOLUTION**

Let $v_O = 5 \text{ V}$ at $v_S = 20 \text{ mV}$ be the Q-point. Then

$$\Delta v_O = v_O(\text{at } v_S = 22 \text{ mV}) - v_O(\text{at } v_S = 18 \text{ mV}) = 5.8 - 4.3 = 1.5 \text{ V}$$

$$\Delta v_S = v_S(\text{at } v_O = 5.8 \text{ V}) - v_S(\text{at } v_O = 4.3 \text{ V}) = 22 - 18 = 4 \text{ mV}$$

(a) The small-signal voltage gain is

$$A_v = \frac{\Delta v_O}{\Delta v_S} = \frac{1.5 \text{ V}}{4 \text{ mV}} = 375 \text{ V/V} \quad \text{(or 51.48 dB)}$$

(b) The DC voltage gain is

$$A_{dc} = A_V = \frac{v_O}{v_S} = \frac{5 \text{ V}}{20 \text{ mV}} = 250 \text{ V/V} \quad \text{(or 47.96 dB)}$$

(c) The limits of input voltage $v_S$ are

$$\frac{(v_O - v_O(\text{min}))}{A_v} \leq v_S \leq v_O(\text{max}) - v_O \left( \frac{v_O(\text{max}) - v_O}{A_v} \right)$$

That is, $-\frac{(5 - 2)}{A_V} \leq v_S \leq \frac{(5.8 - 2)}{A_V}$, or $-8 \text{ V} \leq v_S \leq 8 \text{ V}$, which gives $12 \text{ mV} \leq v_S \leq 28 \text{ mV}$.
2.2.8 Rise Time

The rise time is the time required for the output voltage to rise from 10% to 90% of the steady-state value. If the voltage gain of the amplifier is assumed to be unity, the output voltage due to a step input voltage $V_S$ can be expressed as

$$v_O = V_S(1 - e^{-t/\tau}) \quad (2.12)$$

where $\tau$ is the time constant due to the internal resistance and capacitance of the amplifier. From Eq. (B.40) in Appendix B, the time rise is related to the time constant $\tau$ by

$$t_r = 2.2\tau \quad (2.13)$$

The typical value of rise time is 0.3 $\mu$s for the $\mu$A741 op-amp. Note that a linear operation was assumed in deriving Eq. (2.13) and the effect of slew rate (discussed in Sec. 2.2.9) was ignored.

2.2.9 Slew Rate

The slew rate (SR) is the maximum rate of rise of the output voltage per unit time, and it is measured in volts per microsecond. If a sharp step input voltage is applied to an amplifier, the output will not rise as quickly as the input because the internal capacitors require time to charge to the output voltage level. SR is a measure of how quickly the output of an amplifier can change in response to a change of input frequency. The slew rate depends on the voltage gain, but it is normally specified at unity gain. SR for the LF411 op-amp is 10 V/$\mu$s, whereas it is 0.5 V/$\mu$s for the $\mu$A741C op-amp. The output response due to a step input is shown in Fig. 2.5(a). The output, which follows the slew rate of the op-amp, will be distorted because the op-amp output cannot rise as fast as the input voltage.

With a unity-gain amplifier, the rate of rise of the output voltage for a step signal $V_S$ can be found from Eq. (2.12) to be

$$\frac{dv_O}{dt} = \frac{V_S}{\tau} e^{-t/\tau} \quad (2.14)$$

![FIGURE 2.5 Effect of slew rate on amplifier response]
which becomes maximum at \( t = 0 \) and gives the slew rate SR as

\[
SR = \frac{d v_O}{dt} \bigg|_{t=0} = \frac{V_S}{\tau}
\]

The 3-dB frequency can be related to the time constant \( \tau \) or to the rise time \( t_r \) by

\[
f = \frac{1}{2\pi \tau} = \frac{2.2}{2\pi t_r} = \frac{0.35}{t_r}
\]

Thus, the frequency response is inversely proportional to the rise time \( t_r \). The input signal frequency \( f_s \) should be less than the maximum op-amp frequency; otherwise the output voltage will be distorted. For example, if the rise time of an input signal is \( t_r = 0.1 \) \( \mu \)s, its corresponding input frequency is \( f_s = 0.35/0.1 \) \( \mu \)s = 3.5 MHz, and the output voltage will be distorted in an op-amp unity-gain bandwidth of \( f_{bw} = 1 \) MHz.

Substituting \( \tau \) from Eq. (2.13) and \( t_r \) from Eq. (2.15) gives

\[
SR = \frac{2.2V_S}{t_r} = \frac{2.2V_S f}{0.35} = 6.286V_S f
\]

For a sinusoidal input voltage with a unity gain and without limiting by the slew rate, the output voltage becomes

\[
v_O = V_m \sin \omega t
\]

\[
\frac{dv_O}{dt} = \omega V_m \cos \omega t
\]

which becomes maximum at \( \omega t = 0 \), and SR is given by

\[
SR = \frac{d v_O}{dt} \bigg|_{t=0} = \omega V_m = 2\pi f V_m
\]

which gives the maximum frequency \( f_{s(max)} \) of the sinusoidal input voltage as

\[
f_{s(max)} = \frac{SR}{2\pi V_m}
\]

Slew rate can introduce a significant error if the rate of change of the input voltage is more than the SR of the amplifier. Note that the rate of change of the input voltage rather than the change indicates how fast the input can rise. For example, if the rate of change of a sinusoidal input voltage is very high compared to the SR of the amplifier, the output will be highly distorted and will tend to have a triangular waveform. This situation is shown in Fig. 2.5(b) for a sinusoidal input voltage.
2.3 Amplifier Types

The input signal to an amplifier can be either a voltage source or a current source. The output of an amplifier can be either a voltage source or a current source. Therefore, there are four possible input and output combinations: \(v-v\), \(i-i\), \(v-i\), and \(i-v\). Based on the input and output relationships, amplifiers can be classified into four types: voltage amplifiers, current amplifiers, transconductance amplifiers, and transimpedance amplifiers.

**EXAMPLE 2.3**

**Finding the effect of slew rate on the input frequency**  The slew rate of a unity-gain amplifier is \(SR = 0.7 \text{ V/\mu s}\). The frequency of the input signal is \(f_s = 300\ \text{kHz}\). Calculate (a) the peak sinusoidal input voltage \(V_m\) that will give an output without any distortion and (b) the maximum input frequency \(f_s(\text{max})\) that will avoid distortion if the input has a peak sinusoidal voltage of \(V_m = 5\ \text{V}\).

**SOLUTION**

SR = \(0.7 \text{ V/\mu s} = 0.7 \times 10^6\ \text{V/s}\), \(f = f_s = 300\ \text{kHz}\).

(a) Using Eq. (2.17), we find that the peak value of input voltage is

\[
V_m = \frac{SR}{2\pi f_s} = \frac{0.7 \times 10^6}{2\pi \times 300 \times 10^3} = 371.4\ \text{mV}
\]

(b) From Eq. (2.18), the maximum frequency \(f_s(\text{max})\) becomes

\[
\frac{f_s(\text{max})}{2\pi V_m} = \frac{SR}{2\pi V_m} = \frac{0.7 \times 10^6}{2\pi \times 5} = 22.28\ \text{kHz}
\]

**KEY POINTS OF SECTION 2.2**

- The performance of an amplifier is described by its voltage gain, current gain, power gain, input resistance, and output resistance.
- The gains of an amplifier have high magnitudes and are quoted generally in decibels (dB).
- The power gain is very large because the signal power is very low. The DC power supply (or supplies) provides the load power.
- A DC power supply (or supplies) is needed to establish a \(Q\)-point. The small-signal source is then superimposed on the DC input so that the operating point can move up and down around the \(Q\)-point, and a magnified replica of the signal source is obtained on the output. As long as the signal source is sufficiently small, a nonlinear amplifier exhibits an almost linear characteristic.
- The DC power supply (or supplies) sets the saturation limit(s) of an amplifier.
- There are two types of gain: a DC gain and a small-signal AC gain. The small-signal gain is normally quoted as the gain of the amplifier.
2.3.1 Voltage Amplifiers

An amplifier whose output voltage is proportional to its input voltage is known as a voltage amplifier. The input signal is a voltage source, and the output of the amplifier is also a voltage source. Such an amplifier is referred to as a voltage-controlled voltage source (VCVS); an example is shown in Fig. 2.6(a). The amplifier is connected between a voltage source $v_s$ and a load resistance $R_L$. $R_s$ is the source resistance. $A_{vo}$ is the voltage gain with load resistance $R_L$ disconnected, and it is known as the open-circuit voltage gain. $R_o$ is the output resistance of the amplifier.

The output voltage of a voltage amplifier can be obtained by using the voltage divider rule:

$$v_o = i_o R_L = A_{vo} v_i \frac{R_L}{R_L + R_o} \quad (2.19)$$

From the voltage divider rule, the input voltage $v_i$ to the amplifier is related to the signal voltage $v_s$ by

$$v_i = \frac{R_i}{R_i + R_s} v_s \quad (2.20)$$

Substituting $v_i$ from Eq. (2.20) into Eq. (2.19), we get the effective voltage gain $A_v$, which is defined as the ratio of $v_o$ to $v_s$. That is,

$$A_v = \frac{v_o}{v_s} = \frac{v_o}{v_i} \frac{v_i}{v_s} = \frac{A_{vo} R_i R_L}{(R_i + R_s)(R_L + R_o)} = \frac{A_{vo}}{(1 + R_s/R_i)(1 + R_o/R_L)} \quad (2.21)$$

The current gain $A_i$, which is defined as the ratio of the output current $i_o$ to the input current $i_s$, is given by

$$A_i = \frac{i_o}{i_s} = \frac{A_{vo} v_i}{R_L + R_o} \times \frac{1}{v_i/R_i} = \frac{A_{vo} R_i}{R_L + R_o} \quad (2.22)$$

The power gain will be the product of the voltage gain and the current. That is,

$$A_p = A_v A_i \quad (2.23)$$
Notice from Eq. (2.21) that source resistance \( R_s \) and output resistance \( R_o \) reduce the effective voltage gain \( A_v \). A voltage amplifier must be designed to have an input resistance \( R_i \) much greater than the source resistance \( R_s \) so that \( R_s \ll R_i \). The reduction in gain can also be minimized by designing an amplifier with a very small value of \( R_o \) such that \( R_o \ll R_L \). An ideal voltage amplifier has \( R_o = 0 \) and \( R_i = \infty \) so that there is no reduction in the voltage gain. That is, \( A_v = A_{vo} \), and Eq. (2.21) becomes

\[
v_o = A_{vo} v_s
\]

In most practical implementations of voltage amplifiers in integrated circuits, a differential input is desirable from the viewpoint of performance. The implementation of a VCVS normally begins with a differential input to give a high input resistance and then has an output stage to give a low output resistance. This arrangement is shown in Fig. 2.6(b). If sufficient gain is available from the differential amplifier, only the output stage is required to give a low output resistance. If more gain is required, a second stage will be needed. The specifications of the VCVS and the judgment of the circuit designer will be the major factors in the choice of the implementation.

**EXAMPLE 2.4**

**Determining the design specifications of a voltage amplifier** A voltage amplifier is required to amplify the output signal from a communication receiver that produces a voltage signal of \( v_s = 20 \text{ mV} \) with an internal resistance of \( R_s = 1.5 \text{ k}\Omega \). The load resistance is \( R_L = 15 \text{ k}\Omega \). The desired output voltage is \( v_o \geq 10 \text{ V} \). The amplifier must not draw more than \( 1 \mu\text{A} \) from the receiver. The variation in output voltage when the load is disconnected should be less than 0.5%. Determine the design specifications of the voltage amplifier.

**SOLUTION**

Since the input current is \( i_s \leq 1 \mu\text{A} \), the input resistance of the amplifier can be found from

\[
R_s + R_i = \frac{v_s}{i_s} \geq \frac{20 \text{ mV}}{1 \mu\text{A}} = 20 \text{ k}\Omega
\]

which gives

\[
R_i \geq 20 \text{ k}\Omega - R_s = 20 \text{ k}\Omega - 1.5 \text{ k}\Omega = 18.5 \text{ k}\Omega
\]

The variation in output voltage, which depends on the ratio \( R_o/R_L \), can be found from

\[
\frac{\Delta v_o}{v_o} = \frac{R_o}{R_L + R_o}
\]

which, for \( \Delta v_o/v_o \leq 0.5\% \) and \( R_L = 15 \text{ k}\Omega \), gives \( R_o \leq 75 \text{ \Omega} \). The desired effective voltage gain is \( A_v = v_o/v_s \geq 10 \text{ V}/20 \text{ mV} = 500 \text{ V/V} \) (or 53.98 dB). The open-circuit voltage gain can be found from Eq. (2.21):

\[
500 \leq \frac{A_{vo}}{(1 + R_s/R_i)(1 + R_o/R_L)} = \frac{A_{vo}}{(1 + 1.5 \text{ k}/18.5 \text{ k})(1 + 75/15 \text{ k})}
\]

or \( A_{vo} \geq 543 \text{ V/V} \) (or 54.7 dB)

The amplifier specifications are \( R_i \geq 18.5 \text{ k}\Omega \), \( R_o \leq 75 \text{ \Omega} \), and \( A_{vo} \geq 543 \text{ V/V} \) (or 54.7 dB).
2.3.2 Current Amplifiers

An amplifier whose output current is proportional to its input current is called a current amplifier. Its input is a current source, as shown in Fig. 2.7(a), with a load resistance $R_L$. A current amplifier is represented by a current-controlled current source (CCCS), as shown in Fig. 2.7(b). $A_{is}$ is called the short-circuit current gain (or simply the current gain) with output terminals shorted. $R_i$ is the input resistance, and $R_o$ is the output resistance. A current amplifier is normally used to provide a modest voltage gain but a substantial current gain so that it draws little power from the signal source and delivers a large amount of power to the load. Such an amplifier is often known as a power amplifier.

The output current $i_o$ of the amplifier can be obtained by using the current divider rule:

$$i_o = A_{is} i_i \frac{R_o}{R_o + R_L} \tag{2.26}$$

The input current $i_i$ of the amplifier is related to the signal source current $i_s$ by

$$i_i = \frac{R_s}{R_s + R_i} i_s \tag{2.27}$$

Substituting $i_i$ from Eq. (2.27) into Eq. (2.26), we get the effective current gain $A_i$, which is defined as the ratio of $i_o$ to $i_s$. That is,

$$A_i = \frac{i_o}{i_s} = \frac{i_o}{i_i} \times \frac{i_i}{i_s} = \frac{A_{is} R_s R_o}{(R_s + R_i)(R_o + R_L)} = \frac{A_{is}}{(1 + R_i/R_s)(1 + R_L/R_o)} \tag{2.28}$$

The voltage gain $A_v$, which is defined as the ratio of the output voltage $v_o$ to the input voltage $v_s$, is given by

$$A_v = \frac{v_o}{v_s} = \frac{i_o R_L}{i_s R_s} = A_i \frac{R_L}{R_s} \tag{2.29}$$

The power gain is the product of the voltage gain and the current gain. That is,

$$A_p = A_v A_i \tag{2.30}$$
Notice from Eq. (2.28) that larger values of input resistance \(R_i\) and load resistance \(R_L\) reduce the effective current gain \(A_i\). A current amplifier should have an input resistance \(R_i\) much smaller than the source resistance \(R_s\) so that \(R_i << R_s\). The reduction in gain can also be minimized by designing an amplifier so that the ratio \(R_L/R_o\) is very small—that is, \(R_o >> R_L\). Therefore, an ideal current amplifier has \(R_o = \infty\) and \(R_i = 0\) so that there is no reduction in the current gain. That is, \(A_i = A_{is}\), and Eq. (2.28) becomes

\[
i_o = A_{is}i_s
\]

(2.31)

The implementation of a CCCS can begin with a differential input, as shown in Fig. 2.7(c). A second stage will be necessary because the current differential amplifier will have a low current gain. An output stage may be necessary to give a high output resistance.

**EXAMPLE 2.5**

**Determining the design specifications of a current amplifier**  
A current amplifier is required to amplify the output signal from a transducer that produces a constant current of \(i_s = 1 \text{ mA}\) at an internal resistance varying from \(R_s = 1.5 \text{ k}\Omega\) to \(R_s = 10 \text{ k}\Omega\). The desired output current is \(i_o = 0.5 \text{ A}\) at a load resistance varying from \(R_L = 10 \Omega\) to \(R_L = 120 \Omega\). The variation in output current should be kept within \(\pm 3\%\). Determine the design specifications of the current amplifier.

**SOLUTION**

Since the variation in output current should be kept within \(\pm 3\%\), the variation in the effective current gain \(A_i\) should also be limited to \(\pm 3\%\). According to Eq. (2.28), the variation in \(A_i\) will be contributed by \(A_{is}\), \(R_s\), and \(R_L\). Let us assume that each of them contributes equally to the variation—that is, each contributes \(\pm 1\%\). The nominal short-circuit current gain is \(A_{is} = i_o/i_s = 0.5 \text{ A}/1 \text{ mA} = 500 \text{ A}/\text{A}\). Thus, the value of \(R_o\) that will keep the variation in positive current gain within \(1\%\) for variation in \(R_L\) from 10 \(\Omega\) to 120 \(\Omega\) can be found approximately from

\[
0.99 \frac{R_o}{R_o + 10} = \frac{R_o}{R_o + 120}
\]

which gives \(R_o \approx 10.88 \text{ k}\Omega\) when solved for \(R_o\). Similarly, the value of \(R_i\) that will keep the variation in current gain within \(1\%\) for variation in \(R_i\) from 1.5 \(\text{k}\Omega\) to 10 \(\text{k}\Omega\) can be found approximately from

\[
0.99 \frac{10 \text{ k}}{10 \text{ k} + R_i} = \frac{1.5 \text{ k}}{1.5 \text{ k} + R_i}
\]

which gives \(R_i \approx 17.86 \Omega\) when solved for \(R_i\). Thus, the amplifier specifications are \(A_{is} = 500 \text{ A}/\text{A} \pm 1\%\), \(R_o \geq 10.88 \text{ k}\Omega\), and \(R_i \geq 17.86 \Omega\).

The exact change in \(\Delta A_i\) can be found from

\[
\frac{\Delta A_i}{A_i} = \frac{\Delta A_{is}}{A_{is}} + \frac{1}{1 + R_{s1}/R_i} \times \frac{\Delta R_{s1}}{R_{s1}} - \frac{1}{1 + R_o/R_{L1}} \times \frac{\Delta R_{L1}}{R_{L1}}
\]

\[
= 1\% + \frac{1}{1 + 1.5 \text{k}/17.86} \times 1\% - \frac{1}{1 + 10.88 \text{k}/10} \times 1\% = 1\%
\]
2.3.3 Transconductance Amplifiers

An amplifier that receives a voltage signal as input and provides a current signal as output is called a transconductance amplifier; an example is shown in Fig. 2.8(a). It can be represented by a voltage-controlled current source (VCCS), as shown in Fig. 2.8(b). The amplifier is connected between a voltage source \( v_s \) and a load resistance \( R_L \). Gain parameter \( G_{ms} \), which is the ratio of the short-circuit output current to the input voltage, is called the short-circuit transconductance. From the current divider rule, the output current \( i_o \) is

\[
i_o = G_{ms} v_i \frac{R_o}{R_o + R_L}
\]  

(2.32)

The input voltage \( v_i \) of the amplifier is related to source voltage \( v_s \) by

\[
v_i = \frac{R_i}{R_i + R_s} v_s
\]  

(2.33)

Substituting \( v_i \) from Eq. (2.33) into Eq. (2.32) gives the effective transconductance gain \( G_m \) as

\[
G_m = \frac{i_o}{v_s} = \frac{G_{ms}R_oR_i}{(R_o + R_L)(R_i + R_s)} = \frac{G_{ms}}{1 + R_L/R_o}(1 + R_s/R_i)
\]  

(2.34)

The effective voltage gain \( A_v \) can be found from

\[
A_v = \frac{v_o}{v_s} = \frac{i_oR_L}{v_i} = \frac{v_o}{v_i} \times \frac{R_o}{v_s} = \frac{G_{ms}R_oR_LR_i}{(R_o + R_L)(R_i + R_s)} = \frac{G_{ms}R_L}{1 + R_L/R_o}(1 + R_s/R_i)
\]  

(2.35)

Notice from Eq. (2.34) that the source resistance \( R_s \) and the load resistance \( R_L \) reduce the effective transconductance gain \( G_m \). A transconductance amplifier should have a high input resistance \( R_i \) so that \( R_i >> R_s \) and a very high output resistance \( R_o \) so that \( R_o >> R_L \). Therefore, an ideal transconductance amplifier has \( R_o = \infty \) and \( R_i = \infty \) so that there is no reduction in the voltage gain. That is, \( G_m = G_{ms} \), and Eq. (2.34) becomes

\[
i_o = G_{ms} v_s
\]  

(2.36)

The implementation of a VCCS can begin with a differential input, as shown in Fig. 2.8(c). Since the output resistance of a differential amplifier is reasonably high, one differential stage should be adequate. If more gain is needed, however, a second stage can be added.
A transconductance amplifier can be used to eliminate interaction between two circuits, as shown in Fig. 2.9. The amplifier is connected between the meter and the peak detector. The amplifier should offer a very high resistance to the detector; at the same time, the meter current will be proportional to the peak voltage. The capacitor will continuously monitor the peak value $V_m$ of the input signal. This peak value is indicated by the meter, whose reading depends on the current flowing through it. This technique is often used in electronic circuits to isolate two circuits from each other.

**EXAMPLE 2.6**

**Determining the design specifications of a transconductance amplifier** A transconductance amplifier is needed to record the peak voltage of the circuit in Fig. 2.9. The output recorder needs 10 mA for a reading of 1 cm, and it should read 10 cm / (100 V) for a peak input voltage of 100 V. The internal resistance of the recorder varies from $R_L/100$ to $R_L/500$. The frequency of the input voltage is $f_s = 1$ kHz.

(a) Determine the value of capacitance $C$.

(b) Determine the design specifications of the transconductance amplifier.

**SOLUTION**

(a) The capacitor $C$ will charge to the peak input voltage when the diode conducts, and it will discharge through the amplifier when the diode is off. Let us assume that the discharging time constant $\tau = CR_i$ is related to the input frequency by $\tau = 10/f_s$. For $f_s = 1$ kHz, $CR_i = 10/(1$ kHz) = 10 ms. Let us choose $C = 0.01 \mu F$. Then $R_i = 10$ ms/0.01 $\mu F = 1$ M$\Omega$.

(b) Since the output variation should be kept within $\pm 2\%$, the variation in the effective transconductance $G_m$ should also be limited to $\pm 2\%$. According to Eq. (2.34), the variation in $G_m$ will be contributed by $G_{ms}$ and $R_L$. Let us assume that each of them contributes equally to the variation—that is, each contributes $\pm 1\%$. Note that there is no source resistance: $R_s = 0$. The nominal transconductance gain is

$$G_{ms} = \frac{i_o}{V_s} = \frac{10 \text{ cm}}{100 \text{ V}} \left(\frac{10 \text{ mA}}{1 \text{ cm}}\right) = 1 \text{ mA/V} \pm 1\%$$

Thus, the value of $R_o$ that will keep the gain variation within $1\%$ for variation in $R_L$ from 100 $\Omega$ to 500 $\Omega$ can be found from

$$0.99 \frac{R_o}{R_o + 100} = \frac{R_o}{R_o + 500}$$

which gives $R_o \approx 39.5$ k$\Omega$ when solved for $R_o$. Thus, the amplifier specifications are $G_{ms} = 1 \text{ mA/V} \pm 1\%$, $R_i \approx 1$ M$\Omega$, and $R_o \approx 39.5$ k$\Omega$. 

![Diagram](image-url)
2.3.4 Transimpedance Amplifiers

The input signal to a transimpedance amplifier is a current source, and its output is a voltage source. Such an amplifier can be represented as a current-controlled voltage source (CCVS), as shown in Fig. 2.10(a).

The gain parameter $Z_{mo}$ is the ratio of the open-circuit output voltage to the input current, and it is called the open-circuit transimpedance (or simply the transimpedance).

The output voltage $v_o$ is related to $i_i$ by

$$v_o = \frac{Z_{mo} i_i R_L}{R_L + R_o} \quad (2.37)$$

The input current $i_i$ of the amplifier is related to $i_s$ as follows:

$$i_i = \frac{R_s}{R_s + R_i} i_s \quad (2.38)$$

Substituting $i_i$ from Eq. (2.38) into Eq. (2.37) gives the effective transimpedance $Z_m$:

$$Z_m = \frac{v_o}{i_s} = \frac{Z_{mo} R_L R_s}{(R_L + R_o)(R_s + R_i)} = \frac{Z_{mo}}{(1 + R_o/R_L)(1 + R_i/R_s)} \quad (2.39)$$

The effective voltage gain $A_v$ is given by

$$A_v = \frac{v_o}{v_s} = \frac{i_o R_L}{i_s R_s} = \frac{Z_{mo} R_L}{(R_s + R_i)(R_L + R_o)} \quad (2.40)$$

A transimpedance amplifier must have an input resistance $R_i$ much smaller than the source resistance $R_s$ and an output resistance $R_o$ much smaller than the load resistance $R_L$. An ideal transimpedance amplifier has $R_i = 0$ and $R_o = 0$. That is,

$$v_o = Z_{mo} i_s \quad (2.41)$$

The implementation of a CCVS can begin with a current differential input, as shown in Fig. 2.10(b). If the output stage has a high input resistance and a low output resistance and the gain is adequate, a second stage may not be necessary.
EXAMPLE 2.7

Determining the design specifications of a transimpedance amplifier  A transimpedance amplifier is used to record the short-circuit current of a transducer of unknown internal resistance; the recorder requires 10 V for a reading of 1 cm. The recorder should read 10 cm ± 2% for an input current of 1 A. The input resistance of the recorder varies from $R_L = 5 \, \text{k}\Omega$ to $R_L = 20 \, \text{k}\Omega$. Determine the design specifications of the transimpedance amplifier.

SOLUTION

Since the output variation should be kept within ±2%, the variation of the effective transimpedance $Z_m$ should also be limited to ±2%. According to Eq. (2.39), the variation of $Z_m$ will be contributed by $Z_{mo}$ and $R_L$. Let us assume that each of them contributes equally to the variation—that is, each contributes ±1%. Since the source resistance is unknown, we will assume that the input resistance is very small, tending to zero (say, $R_i = 10 \, \Omega$).

The nominal transimpedance gain is

$$Z_{mo} = \frac{v_o}{i} = \left( \frac{10 \, \text{V}}{1 \, \text{cm}} \right) \left( \frac{1 \, \text{A}}{10 \, \text{cm}} \right) = 100 \, \text{V/A} \pm 1\%$$

Thus, the value of $R_o$ that will keep the gain variation within 1% for variation in $R_L$ from 5 kΩ to 20 kΩ can be found from

$$0.99 \frac{20 \, \text{k}}{20 \, \text{k} + R_o} = \frac{5 \, \text{k}}{5 \, \text{k} + R_o}$$

which gives $R_o \approx 67.6 \, \Omega$. Therefore, the amplifier specifications are $Z_{mo} = 100 \, \text{V/A} \pm 1\%$, $R_o \approx 67.6 \, \Omega$, and $R_i \leq 10 \, \Omega$.

KEY POINTS OF SECTION 2.3

- Amplifiers can be classified into four types: voltage, current, transconductance, and transimpedance. Their characteristics are summarized in Table 2.1.
- Amplifiers are used in such applications as capacitance multiplication, creating negative resistance, and inductance simulation.
- Establishing the design specifications of an amplifier requires identifying the gain, the input resistance, and the output resistance.

TABLE 2.1  Characteristics of ideal amplifiers

<table>
<thead>
<tr>
<th>Amplifier Type</th>
<th>Gain</th>
<th>Input Resistance $R_i$</th>
<th>Output Resistance $R_o$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Voltage</td>
<td>$A_{vo}$ (V/V)</td>
<td>$\infty$</td>
<td>0</td>
</tr>
<tr>
<td>Current</td>
<td>$A_{i}$ (A/A)</td>
<td>0</td>
<td>$\infty$</td>
</tr>
<tr>
<td>Transconductance</td>
<td>$G_{ms}$ (A/V)</td>
<td>$\infty$</td>
<td>$\infty$</td>
</tr>
<tr>
<td>Transimpedance</td>
<td>$Z_{mo}$ (V/A)</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
2.4 Cascaded Amplifiers

Normally one amplifier alone cannot meet the specifications for gain, input resistance, and output resistance. To satisfy the specifications, two or more amplifiers are often cascaded. Any combination of the four types of amplifiers can be used. As illustrations, we will discuss cascaded voltage amplifiers and cascaded current amplifiers.

2.4.1 Cascaded Voltage Amplifiers

Voltage amplifiers are cascaded to increase the overall voltage gain. Consider three cascaded voltage amplifiers, as shown in Fig. 2.11(a). The overall open-circuit voltage gain \( A_{vo} \) of the cascaded amplifiers can be found from

\[
A_{vo} = \frac{v_o}{v_{i1}} = \frac{v_{i2}}{v_{i1}} \times \frac{v_{i3}}{v_{i2}} \times \frac{v_o}{v_{i3}} \quad (2.42)
\]

If \( A_{vo1}, A_{vo2}, \) and \( A_{vo3} \) are the voltage gains of stages 1, 2, and 3, respectively, such that \( v_{i2} = A_{vo1}v_{i1}, \)
\( v_{i3} = A_{vo2}v_{i2}, \) and \( v_o = A_{vo3}v_{i3}, \) then Eq. (2.42) becomes

\[
A_{vo} = A_{vo1}A_{vo2}A_{vo3} \quad (2.43)
\]

![Cascaded voltage amplifiers](image)

**FIGURE 2.11** Cascaded voltage amplifiers
which indicates that the overall open-circuit voltage gain is the product of the individual gain of each stage. The voltage gains $A_{v1}$, $A_{v2}$, and $A_{v3}$ are related to the no-load voltage gains $A_{vo1}$, $A_{vo2}$, and $A_{vo3}$ by

$$A_{v1} = \frac{R_{i2}}{R_{o1} + R_{i2}} A_{vo1}$$

$$A_{v2} = \frac{R_{i3}}{R_{o2} + R_{i3}} A_{vo2}$$

$$A_{v3} = \frac{R_{i3}}{R_{o3} + R_{L}} A_{vo3}$$

If the output resistance of each stage is negligible so that

$$R_{o1} = R_{o2} = R_{o3} = 0$$

then the voltage gain of each stage becomes the same as its open-circuit voltage gain. That is,

$$v_{i2} = A_{vo1} v_{i1} \quad v_{i3} = A_{vo2} v_{i2} \quad v_o = A_{vo3} v_{i3}$$

The overall open-circuit voltage gain $A_{vo}$ in Eq. (2.43) is then given by

$$A_{vo} = A_{vo1} A_{vo2} A_{vo3} \quad (2.44)$$

Therefore, the three voltage amplifiers can be represented by an equivalent single voltage amplifier with a voltage gain of $A_{vo}$, $R_o = R_{o3}$, and $R_1 = R_{i1}$, as shown in Fig. 2.11(b).

### 2.4.2 Cascaded Current Amplifiers

Current amplifiers can be connected to increase the effective current gain. Consider three cascaded current amplifiers, as shown in Fig. 2.12(a). The overall short-circuit current gain $A_{is}$ of the cascaded current amplifiers can be found from

$$A_{is} = \frac{i_o}{i_{i1}} = \frac{i_2}{i_{i1}} \times \frac{i_3}{i_{i2}} \times \frac{i_o}{i_{i3}} \quad (2.45)$$

If $A_{i1}$, $A_{i2}$, and $A_{i3}$ are the current gains of stages 1, 2, and 3, such that $i_{i2} = A_{i1} i_{i1}$, $i_{i3} = A_{i2} i_{i2}$, and $i_o = A_{i3} i_{i3}$, then Eq. (2.45) becomes $A_{is} = A_{i1} A_{i2} A_{i3}$. The current gains $A_{i1}$, $A_{i2}$, and $A_{i3}$ are related to the short-circuit current gains $A_{is1}$, $A_{is2}$, and $A_{is3}$ by

$$A_{i1} = \frac{R_{o1}}{R_{o1} + R_{i2}} A_{is1}$$

$$A_{i2} = \frac{R_{o2}}{R_{o2} + R_{i3}} A_{is2}$$

$$A_{i3} = \frac{R_{o3}}{R_{o3} + R_{L}} A_{is3}$$

If the output resistance of each stage is very high, tending to infinity, then

$$R_{o1} = R_{o2} = R_{o3} = \infty$$

and

$$i_{i2} = A_{is1} i_{i1} \quad i_{i3} = A_{is2} i_{i2} \quad i_o = A_{is3} i_{i3}$$
Finding the parameters of cascaded voltage amplifiers

The parameters of the cascaded voltage amplifiers in Fig. 2.11(a) are $R_s = 2 \, \text{k}\Omega$, $R_o = R_{o1} = R_{o2} = R_{o3} = 200 \, \Omega$, $R_i = R_{i1} = R_{i2} = R_{i3} = R_L = 1.5 \, \text{k}\Omega$, and $A_{v1} = A_{v2} = A_{v3} = 80$. Calculate (a) the overall open-circuit voltage gain $A_{vo} = v_o / v_i$, (b) the effective voltage gain $A_v = v_o / v_s$, (c) the overall current gain $A_i = i_o / i_s$, and (d) the power gain $A_p = P_L / P_i$.

**SOLUTION**

(a) Using Eq. (2.19), we can calculate the voltage gain of stage 1 and stage 2 as follows:

$$A_{v1} = A_{v2} = \frac{A_{v1}R_{i2}}{R_{i2} + R_{o1}} = \frac{80 \times 1.5 \, \text{k}}{1.5 \, \text{k} + 200} = 70.588 \, \text{V/V}$$

From Eq. (2.43), the overall open-circuit voltage gain of the cascaded amplifiers is

$$A_{vo} = \frac{v_o}{v_i} = A_{v1}A_{v2}A_{v3} = (70.588)^2 \times 80$$

$$= 398,616 \, \text{V/V} \quad \text{[or } 20 \times \log (398,616) = 112.01 \, \text{dB}]$$
2.5 Frequency Response of Amplifiers

So far, we have assumed that there are no reactive elements in an amplifier and that the gain of an amplifier remains constant at all frequencies. However, the gain of practical amplifiers is frequency dependent, and even the input and output impedances of amplifiers vary with the frequency. If $\omega$ is the frequency of the input signal in radians per second, the output sinusoid $V_o(\omega)$ can have a different amplitude and phase than the input sinusoid $V_i(\omega)$. The voltage gain $A_v(\omega) = V_o(\omega)/V_i(\omega)$ will have a magnitude and phase angle. If a sine-wave signal with a specific frequency is applied at the input of an amplifier, the output should be a sinusoid of the same frequency. The frequency response of an amplifier refers to the amplitude of the output sinusoid and its phase relative to the input sinusoid (see Appendix B).

An amplifier is operated at a DC $Q$-point and is subjected to two types of signals: AC signals and DC signals. Often several amplifiers are cascaded by coupling capacitors, as shown in Fig. 2.13, so that the AC signal from the source can flow from one stage to the next stage while the DC signal is blocked. As a result, the DC biasing voltages of the amplifiers do not affect the signal source, adjacent stages, or the load. Such cascaded amplifiers are called capacitive- (or AC-) coupled amplifiers. However, amplifiers in integrated circuits are connected directly, as shown in Fig. 2.14, because capacitors cannot be fabricated in integrated form; such amplifiers are called direct- (or DC-) coupled amplifiers.
Introduction to Amplifiers and Frequency Response

At low frequencies, coupling capacitors, which are on the order of $10^{-9}$ F, offer high reactance on the order of $1 \text{k} \Omega$ and attenuate the signal source. At high frequencies, these capacitors have reactance on the order of $1 \text{pF}$ and thus essentially short-circuit. Therefore, AC-coupled amplifiers will pass signals of high frequencies only.

There are no coupling capacitors in DC-coupled amplifiers. However, the presence of small capacitors on the order of $1 \text{pF}$ is due to the internal capacitances of the amplifying devices and also due to stray wiring capacitance between the signal-carrying conductors and the ground. The frequency response of an amplifier depends on the type of coupling. An amplifier can exhibit one of three frequency characteristics: low-pass, high-pass, or band-pass.

### 2.5.1 Low-Pass Characteristic

Consider the transconductance amplifier shown in Fig. 2.15(a). $C_2$, which is connected across the load $R_L$, could be the output capacitance of the amplifier or the stray capacitance between the output terminal and the ground. $C_2$ forms a parallel path to the signal flowing from the amplifier to the load $R_L$. The output voltage in Laplace’s domain is

$$V_o(s) = -G_{ms}V_i(s) \left( \frac{1}{sC_2} \right) = -G_{ms}R_L \frac{1}{1 + sC_2R_L} V_i(s) \quad (2.47)$$

Using the voltage divider rule, we get $V_i(s) = V_s R_1/(R_s + R_1)$, which, after substitution in Eq. (2.47), gives the voltage gain as

$$A_v(s) = \frac{V_o(s)}{V_i(s)} = -\frac{G_{ms}R_1R_i}{(R_s + R_1)(1 + sC_2R_L)} \quad (2.48)$$

![Capacitive-coupled amplifiers](image1)

**FIGURE 2.13** Capacitive-coupled amplifiers

At low frequencies, coupling capacitors, which are on the order of $10 \mu$F, offer high reactance on the order of $1 \text{k} \Omega$ and attenuate the signal source. At high frequencies, these capacitors have reactance on the order of $1 \text{pF}$ and thus essentially short-circuit. Therefore, AC-coupled amplifiers will pass signals of high frequencies only.

There are no coupling capacitors in DC-coupled amplifiers. However, the presence of small capacitors on the order of $1 \text{pF}$ is due to the internal capacitances of the amplifying devices and also due to stray wiring capacitance between the signal-carrying conductors and the ground. The frequency response of an amplifier depends on the type of coupling. An amplifier can exhibit one of three frequency characteristics: low-pass, high-pass, or band-pass.

### 2.5.1 Low-Pass Characteristic

Consider the transconductance amplifier shown in Fig. 2.15(a). $C_2$, which is connected across the load $R_L$, could be the output capacitance of the amplifier or the stray capacitance between the output terminal and the ground. $C_2$ forms a parallel path to the signal flowing from the amplifier to the load $R_L$. The output voltage in Laplace’s domain is

$$V_o(s) = -G_{ms}V_i(s) \left( \frac{1}{sC_2} \right) = -G_{ms}R_L \frac{1}{1 + sC_2R_L} V_i(s) \quad (2.47)$$

Using the voltage divider rule, we get $V_i(s) = V_s R_1/(R_s + R_1)$, which, after substitution in Eq. (2.47), gives the voltage gain as

$$A_v(s) = \frac{V_o(s)}{V_i(s)} = -\frac{G_{ms}R_1R_i}{(R_s + R_1)(1 + sC_2R_L)} \quad (2.48)$$

![Direct-coupled amplifiers](image2)

**FIGURE 2.14** Direct-coupled amplifiers
Equation (2.48) can be written in general form as

$$A_v(s) = \frac{A_{v(mid)}}{1 + s \tau_2} = \frac{A_{v(mid)}}{1 + s/\omega_H}$$

(2.49)

where

$$A_{v(mid)} = \frac{G_{m}R_L}{R_s + R_i}$$

(2.50)

$$\tau_2 = C_2R_L$$

(2.51)

$$\omega_H = \frac{1}{\tau_2} = \frac{1}{C_2R_L}$$

(2.52)

In the frequency domain, $s = j\omega$ and Eq. (2.49) becomes

$$A_v(j\omega) = \frac{A_{v(mid)}}{1 + j\omega/\omega_H}$$

(2.53)

Thus, the magnitude $|A_v(j\omega)|$ can be found from

$$|A_v(j\omega)| = \frac{A_{v(mid)}}{[1 + (\omega/\omega_H)^2]^{1/2}}$$

(2.54)

and the phase angle $\phi$ of $A_v(j\omega)$ is given by

$$\phi = -\tan^{-1}\left(\frac{\omega}{\omega_H}\right)$$

(2.55)
For $\omega << \omega_H$, let us assume that $A_v(\text{mid}) = 1$. That is,

$$
\left| A_v(j\omega) \right| = A_v(\text{mid}) = 1
$$

$$
20 \log_{10} \left| A_v(j\omega) \right| = 0
$$

$$
\phi = 0
$$

Therefore, at a low frequency, the magnitude plot of $A_v(j\omega)$ is approximately a straight horizontal line at 0 dB. For $\omega >> \omega_H$,

$$
\left| A_v(j\omega) \right| \approx \left( \frac{\omega_H}{\omega} \right)
$$

$$
20 \log_{10} \left| A_v(j\omega) \right| = 20 \log_{10} \left( \frac{\omega_H}{\omega} \right)
$$

$$
\phi \approx -\frac{\pi}{2} \quad (90 \text{ degrees})
$$

For $\omega = \omega_H$,

$$
\left| A_v(j\omega) \right| = \frac{1}{\sqrt{2}}
$$

$$
20 \log_{10} \left| A_v(j\omega) \right| = 20 \log_{10} \left( \frac{1}{\sqrt{2}} \right) = -3 \text{ dB}
$$

$$
\phi = -\frac{\pi}{4} \quad (45 \text{ degrees})
$$

Let us consider a high-frequency $\omega = \omega_1$ such that $\omega_1 >> \omega_H$. The magnitude is $20 \log_{10} (\omega_H/\omega_1)$ at $\omega = \omega_1$. At $\omega = 10\omega_1$, the magnitude is $20 \log_{10} (\omega_H/10\omega_1)$. The change in magnitude becomes

$$
20 \log_{10} \left( \frac{\omega_1}{10\omega_H} \right) - 20 \log_{10} \left( \frac{\omega_H}{\omega_1} \right) = 20 \log_{10} \left( \frac{1}{10} \right) = -20 \text{ dB}
$$

If the frequency is doubled so that $\omega = 2\omega_1$, the change in magnitude becomes

$$
20 \log_{10} \left( \frac{\omega_1}{2\omega_H} \right) - 20 \log_{10} \left( \frac{\omega_H}{\omega_1} \right) = 20 \log_{10} \left( \frac{1}{2} \right) = -6 \text{ dB}
$$

The frequency response is shown in Fig. 2.15(b). If the frequency is doubled, the increase on the frequency axis is called an octave increase. If the frequency is increased by a factor of 10, the increase is called a decade increase. For a decade increase in frequency, the magnitude changes by $-20$ dB and the magnitude plot is a straight line with a slope of $-20$ dB/decade (or $-6$ dB/octave). The magnitude curve is therefore defined by two straight-line asymptotes, which meet at the corner frequency $\omega_H$. The difference between the actual magnitude curve and the asymptotic curve is largest at the break frequency. The error can be found by substituting $\omega$ for $\omega_H$. That is, $\left| A_v(j\omega) \right| = 1/\sqrt{2}$ and $20 \log_{10} \left( 1/\sqrt{2} \right) = -3$ dB. This error is symmetrical with respect to the break (or corner) frequency, which is defined as the frequency at which the magnitude of the gain falls to 70.7% of the constant gain. The break frequency is also known as the 3-dB (or cutoff or half-power) frequency. The voltage gain will fall as the frequency increases beyond $\omega_H$. 

---

*Introduction to Amplifiers and Frequency Response*

65

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
For frequencies \( \omega \ll \omega_H \), the gain will be almost independent of frequency. An amplifier with this type of response is known as a low-pass amplifier. A \( v_{\text{mid}} \) is the pass-band or midband gain. The bandwidth (BW) of an amplifier is defined as the range of frequencies over which the gain remains within 3 dB (29.3%) of constant gain \( A_{v\text{(mid)}} \). That is, \( \text{BW} = \omega_H \). Amplifiers for video signals are generally DC coupled, and the frequencies vary from 0 (DC) to 4.5 MHz.

2.5.2 High-Pass Characteristic

Consider the transconductance amplifier shown in Fig. 2.16(a). \( C_1 \) is the isolating capacitor between the signal source and the amplifier. The output voltage in Laplace’s domain is

\[
V_o(s) = -G_{ms} R_L V_i(s)
\]

(2.56)

From the voltage divider rule, the voltage \( V_i(s) \) is related to \( V_s(s) \) by

\[
V_i(s) = \frac{R_i}{R_s + R_i + 1/sC_1} V_s(s) = \frac{sC_1 R_i}{1 + sC_1 (R_s + R_i)} V_s(s)
\]

(2.57)

Substituting \( V_i(s) \) from Eq. (2.57) into Eq. (2.56) gives the voltage gain:

\[
A_v(s) = \frac{V_o(s)}{V_s(s)} = -\frac{G_{ms} R_L R_i}{R_s + R_i} \times \frac{sC_1 (R_s + R_i)}{1 + sC_1 (R_s + R_i)}
\]

(2.58)

Equation (2.58) can be written in general form as

\[
A_v(s) = \frac{A_{v\text{(mid)}} s \tau_1}{1 + s \tau_1} = \frac{A_{v\text{(mid)}} s}{s + 1/\tau_1} = \frac{A_{v\text{(mid)}} s}{s + \omega_L}
\]

(2.59)
In the frequency domain, \( s = j\omega \) and Eq. (2.59) becomes

\[
A_v(j\omega) = \frac{A_v(\text{mid}) j\omega}{j\omega + \omega_L}
\] (2.63)

Thus, the magnitude \( |A_v(j\omega)| \) can be found from

\[
|A_v(j\omega)| = \frac{A_v(\text{mid}) \omega}{\sqrt{\omega^2 + \omega_L^2}}
\] (2.64)

and the phase angle \( \phi \) of \( A_v(j\omega) \) is given by

\[
\phi = 90^\circ - \tan^{-1}\left(\frac{\omega}{\omega_L}\right)
\] (2.65)

Let us assume that \( A_{v(\text{mid})} = 1 \). For \( \omega << \omega_L \),

\[
|A_v(j\omega)| = \frac{\omega}{\omega_L}
\]

\[
20 \log_{10} |A_v(j\omega)| = 20 \log_{10}\left(\frac{\omega}{\omega_L}\right)
\]

\[
\phi = \frac{\pi}{2} \quad (90 \text{ degrees})
\]

Therefore, for a decade increase in frequency, the magnitude changes by +20 dB. The magnitude plot of \( A_v(j\omega) \) is a straight line with a slope of +20 dB/decade (or +6 dB/octave). For \( \omega >> \omega_L \),

\[
|A_v(j\omega)| = A_{v(\text{mid})} = 1
\]

\[
20 \log_{10} |A_v(j\omega)| = 0
\]

\[
\phi = 0
\]

Therefore, at a high frequency, the magnitude plot is a straight horizontal line at 0 dB. At \( \omega = \omega_L \),

\[
|A_v(j\omega)| = \frac{1}{\sqrt{2}}
\]

\[
20 \log_{10} \left(\frac{1}{\sqrt{2}}\right) = -3 \text{ dB}
\]

\[
\phi = \frac{\pi}{4} \quad (45 \text{ degrees})
\]

The frequency response is shown in Fig. 2.16(b). This circuit passes only the high-frequency signal, and the amplitude is low at a low frequency. The voltage gain will vary with the frequency for \( \omega << \omega_L \). For \( \omega >> \omega_L \), the gain will be almost independent of frequency. This type of amplifier is known as a high-pass amplifier. \( \omega_L \) is known as the break (corner, cutoff, 3-dB, or half-power) frequency, and \( A_{v(\text{mid})} \) is the pass-band or midband gain. Note that for sufficiently high frequencies, the high-pass characteristic of practical amplifiers will tend to attenuate because of the internal capacitances of the amplifying devices.
2.5.3 Band-Pass Characteristic

A capacitive-coupled amplifier will have both coupling capacitors and device capacitors (or stray capacitors). Let us connect both $C_1$ and $C_2$, as shown in Fig. 2.17(a). The circuit will exhibit a band-pass characteristic. Substituting $V_i(s)$ from Eq. (2.57) into Eq. (2.47) gives the voltage gain as

$$A_v(s) = \frac{V_o(s)}{V_i(s)} = \frac{-G_m R_i R_1}{R_s + R_1} \times \frac{s C_1 (R_s + R_i)}{1 + s C_1 (R_s + R_i)} \times \frac{1}{1 + s C_2 R_L}$$

(2.66)

which can be written in general form as

$$A_v(s) = \frac{A_{v(mid)} s}{(s + \omega_L)(1 + s/\omega_H)}$$

(2.67)

In the frequency domain, $s = j\omega$ and Eq. (2.67) becomes

$$A_v(j\omega) = \frac{A_{v(mid)} j\omega}{(j\omega + \omega_L)(1 + j\omega/\omega_H)}$$

(2.68)

Thus, the magnitude $|A_v(j\omega)|$ can be found from

$$|A_v(j\omega)| = \frac{A_{v(mid)} \omega}{[\omega^2 + \omega_L^2]^{1/2} [1 + (\omega/\omega_H)^2]^{1/2}}$$

(2.69)

and the phase angle $\phi$ of $A_v(j\omega)$ is given by

$$\phi = 90^\circ - \tan^{-1}(\omega/\omega_L) - \tan^{-1}(\omega/\omega_H)$$

(2.70)

Thus, the voltage gain will remain almost constant if $\omega_L < \omega < \omega_H$. The frequency behavior is shown in Fig. 2.17(b). This is a band-pass circuit, and $A_{v(mid)}$ is the midfrequency (or pass-band) gain. The bandwidth (BW), which is the range of frequencies over which the gain remains within 3 dB (29.3%) of constant gain $A_{v(mid)}$, is thus the difference between the cutoff frequencies. That is, $\text{BW} = \omega_H - \omega_L$. Note that $A_{v(mid)}$ is not the DC gain because under DC conditions capacitor $C_1$ will be open-circuited and there will be no output voltage. Audio amplifiers are generally AC coupled because the frequency range of audio signals is 20 Hz to 15 kHz. The audio signal source and the loudspeakers are isolated by coupling capacitors.

If the bandwidth of a band-pass amplifier is shortened so that the gain peaks around a particular frequency (called the center frequency) and falls off on both sides of this frequency, as shown in Fig. 2.17(c), the amplifier is called a tuned amplifier. Such an amplifier is generally used in the front end of radio and TV receivers. The center frequency $f_C$ of a tuned amplifier can be adjusted to coincide with the frequency.
of a desired channel so that the signals of that particular channel can be received and signals of other channels are attenuated or filtered out.

### 2.5.4 Gain and Bandwidth Relation

Using $\omega = 2\pi f$, we can write the voltage gain of a low-pass amplifier as

$$A_v(j\omega) = \frac{A_{v(mid)}}{1 + \frac{j\omega}{f_H}} \quad (2.71)$$

where $f_H$ is the break (or 3-dB) frequency in hertz.

For $f >> f_H$, Eq. (2.71) is reduced to

$$A_v(j\omega) = \frac{A_{v(mid)}}{f_H} = \frac{A_{v(mid)}f_H}{f} \quad (2.72)$$

The magnitude of this gain becomes unity (or 0 dB) at frequency $f = f_{bw}$. That is,

$$f_{bw} = A_{v(mid)}f_H \quad (2.73)$$

where $f_{bw}$ is called the **unity-gain bandwidth**. Bandwidth (BW) is often quoted as the frequency range over which the voltage gain $|A(j\omega)|$ is unity. The unity-gain bandwidth of a band-pass amplifier becomes $A_{v(mid)}(f_H - f_L)$. It is important to note that according to Eq. (2.73), the gain–bandwidth product of an amplifier remains constant.

#### EXAMPLE 2.9

Determining coupling capacitors to satisfy frequency specifications  A voltage amplifier should have a midrange voltage gain of $A_{v(mid)} = -200$ in the frequency range of 1 kHz to 100 kHz. The source resistance is $R_s = 2 \Omega$, and the load resistance is $R_L = 10 \Omega$.

**(a)** Determine the specifications of the amplifier and the values for coupling capacitor $C_1$ and shunt capacitor $C_2$ shown in Fig. 2.17(a).

**(b)** Use PSpice/SPICE to verify your design by plotting the frequency response $|A_v(j\omega)|$ against frequency.

**SOLUTION**

**(a)** Let us choose a transconductance amplifier of $R_i = 1 \Omega$ and $R_o = \infty$. From Eq. (2.60), we can find the value of $G_{ms}$ that will give $A_{v(mid)} = -200$. That is,

$$G_{ms} = \frac{A_{v(mid)}(R_s + R_i)}{R_iR_o} = \frac{200 \times (2 \Omega + 1 \Omega)}{10 \Omega \times 1 \Omega} = -20.04 \text{ mA/V}$$

For $f_H = 100 \text{ kHz}$, Eq. (2.52) gives the required value of $C_2$ as

$$C_2 = \frac{1}{(R_L f_H)^2} = \frac{1}{(2\pi f_H R_L)^2} = \frac{1}{(2\pi \times 100 \text{ kHz} \times 10 \Omega)^2} = 159.15 \text{ pF}$$

For $f_L = 1 \text{ kHz}$, Eq. (2.62) gives the required value of $C_1$ as

$$C_1 = \frac{1}{[(R_s + R_i) f_L]^2} = \frac{1}{[2\pi (R_s + R_i) f_L]^2} = \frac{1}{[2\pi \times 1 \text{ kHz} \times (2 \Omega + 1 \text{ M}\Omega)]^2} = 158.84 \text{ pF}$$
FIGURE 2.19 PSpice plot of frequency response for Example 2.9

(b) The circuit for PSpice simulation is shown in Fig. 2.18. The PSpice plot of the frequency response is shown in Fig. 2.19, which gives $A_{v(mid)} = 198$ (expected value is 200), $f_L = 984$ Hz (expected value is 1 kHz), and $f_H = 102$ kHz (expected value is 100 kHz).

KEY POINTS OF SECTION 2.5

- The gain of practical amplifiers is frequency dependent. The frequency response of an amplifier refers to the amplitude and phase of the output sinusoid relative to the input sinusoid. The frequency response is an important specification of an amplifier.
- Video amplifiers operate in the frequency range from 0 (DC) to 4.5 MHz and use direct coupling. That is, there are no coupling capacitors. However, the presence of small capacitors is due to the internal capacitances of the amplifying devices and also to stray wiring capacitance. These capacitors form a parallel path with the AC signal and therefore pass signals of low frequencies only.
- Audio amplifiers, which operate in the frequency range from 20 Hz to 15 kHz, use coupling capacitors so that the AC signal can flow from one stage to the next stage and the DC signals are blocked. These capacitors form a series path with the AC signal and therefore pass signals of high frequencies only. The upper frequency is limited by the device and/or stray capacitances.
- Frequency response also known as a Bode plot, which is a plot of the magnitude and the phase against the frequency, can describe the frequency characteristic and stability of an amplifier.
- Depending on the frequency response, an amplifier falls into one of three categories: low-pass, high-pass, or band-pass.
- The coupling capacitances of an amplifier normally determine the low break frequencies, whereas internal capacitances determine the high break frequencies.
An impedance known as feedback impedance is often connected across the input and output sides of an amplifier. Miller’s theorem [1] simplifies the analysis of feedback amplifiers. The theorem states that if an impedance is connected between the input side and the output side of a voltage amplifier, this impedance can be replaced by two equivalent impedances—one connected across the input and the other connected across the output terminals. Figure 2.20 shows the relationship between the amplifier and its equivalent circuit. If we choose the appropriate values of impedances $Z_{im}$ and $Z_{om}$, the two circuits in Fig. 2.20[(a) and (b)] can be made identical. In Sec. 2.7.3 we will apply Miller’s theorem to find the frequency response of amplifiers.

If $A_{vo}$ is the open-circuit voltage gain of the amplifier, the output voltage $V_o$ is related to the input voltage $V_i$ by

$$V_o = A_{vo}V_i \quad (2.74)$$

The input current $I_i$ of the amplifier in Fig. 2.20(a) is given by

$$I_i = \frac{V_i - V_o}{Z_f} \quad (2.75)$$

Substituting $V_o$ from Eq. (2.74) into Eq. (2.75) yields

$$I_i = \frac{V_i - A_{vo}V_i}{Z_f} = V_i\left(\frac{1 - A_{vo}}{Z_f}\right) \quad (2.76)$$

The input impedance $Z_i$ of the circuit in Fig. 2.20(b) must be the same as that of Fig. 2.20(a), and it can be found from Eq. (2.76):

$$Z_{im} = \frac{V_i}{I_i} = \frac{Z_f}{1 - A_{vo}} \quad (2.77)$$

The output current $I_o$ of the circuit in Fig. 2.20(a) is given by

$$I_o = \frac{V_o - V_i}{Z_f} \quad (2.78)$$
Substituting $V_i$ from Eq. (2.74) into Eq. (2.78) yields

$$I_o = \frac{V_o - V_o/A_vo}{Z_f} = V_o \left( \frac{1 - 1/A_vo}{Z_f} \right)$$  \hspace{1cm} (2.79)

The output impedance $Z_{om}$ of the circuit in Fig. 2.20(b) must be the same as that of Fig. 2.20(a), and it can be found from Eq. (2.79):

$$Z_{om} = \frac{V_o}{I_o} = \frac{Z_f}{1 - 1/A_vo} = \frac{Z_f A_vo}{A_vo - 1}$$  \hspace{1cm} (2.80)

**NOTES**

1. Equations (2.77) and (2.80) are derived with the assumption that the voltage amplifier is an ideal one and that the open-circuit voltage gain $A_vo$ can be found without connecting the impedance $Z_f$. That is, the input impedance $R_i$ of the amplifier in Fig. 2.20(a) is very high, tending to infinity, and the output resistance $R_o$ is very small, tending to zero. They have no effect on the analysis. $Z_{im}$ and $Z_{om}$ are called the *Miller impedances*.

2. The Miller theorem is applicable provided the amplifier has no independent source. The open-circuit voltage gain $A_vo$ of the amplifier must be negative so that $(1 - A_vo)$ is a positive quantity. Otherwise $Z_{im}$ will have a negative value.

3. If a capacitor is connected between the input and output terminals of an amplifier with a negative voltage gain, this capacitor has a dominant effect and lowers the high break frequency significantly.

**KEY POINT OF SECTION 2.6**

- According to Miller’s theorem if an impedance is connected between the input side and the output side of a voltage amplifier, this impedance can be replaced by two equivalent impedances—one connected across the input and the other connected across the output terminal.

### 2.7 Frequency Response Methods

An amplifier generally receives a small AC signal from the input side, then amplifies the signal and delivers it to the output side. The amplifier requires DC supplies to operate the internal devices such as transistors. The internal DC voltages and DC currents within the amplifiers are subjected to variations. The amplifiers are often connected to the input signal source and the load resistor through coupling capacitors that effectively block low-frequency signals. The internal transistors [2, 3] have small capacitances that limit the maximum useful frequency of the amplifier.

A typical arrangement is shown in Fig. 2.21(a). Coupling capacitors $C_1$ and $C_2$, which have much higher values (typically on the order of 10 $\mu$F) than internal capacitances, are in series with the signal flow and set the low-frequency limit of the amplifier. Let us assume that the amplifier can be modeled by an equivalent circuit consisting of $R_i$, $R_o$, $C_1$, $C_o$, and $g_m$. This is shown in Fig. 2.21(b). A typical frequency
plot (magnitude versus frequency) is shown in Fig. 2.21(c). \( f_L \) is the dominant low cutoff frequency, \( f_H \) is the dominant high cutoff frequency, and \( A_{PB} \) is the pass-band voltage gain. Thus, the performance of amplifiers depends on the input signal frequency, and the design specifications usually quote the voltage gain at a specified bandwidth.

Since there are five capacitors in Fig. 2.21(b), the denominator of the transfer function \( A(s) \) will be a fifth-order polynomial in \( s \). Finding the exact cutoff frequencies requires the calculation of five polynomial roots. Because derivation of the voltage transfer function \( A(s) \) (similar to Eq. [2.48]) for the circuit in Fig. 2.21(b) is a tedious task, the analysis is normally carried out on a computer. However, the analysis can be simplified by assuming that \( f_L \) and \( f_H \) are separated by at least one decade so that \( f_L \) does not affect \( f_H \). Then the low and high cutoff frequencies can be found separately. Thus, we can use the following steps to determine the complete frequency response of an amplifier:

1. Find the small-signal AC equivalent circuit of the amplifier as shown in Fig. 2.21(b).
2. Find the low break frequency or frequencies due to the coupling capacitors.
3. Find the high break frequency or frequencies due to the internal capacitors.
4. Find the pass-band gain of the amplifier.

### 2.7.1 Low-Frequency Transfer Function Method

At low frequencies (usually less than \( f_L = 1.5 \text{ kHz} \)), the internal capacitors, which are typically on the range of 1 pF to 10 pF, have reactance on the order of 10 M\( \Omega \) and are essentially open-circuited. We will assume that the internal capacitances are small so that the capacitors are effectively open-circuited. Thus, the low-frequency behavior is determined mostly by the coupling capacitors. The equivalent circuit for
finding the low break frequencies is shown in Fig. 2.22. Using the voltage divider rule, we can relate $V_i(s)$ to $V_s(s)$:

$$V_i(s) = \frac{R_i V_s(s)}{R_s + R_i + 1/sC_1} = \frac{R_i}{R_s + R_i} \times \frac{s}{s + 1/[C_1(R_s + R_i)]} V_i(s) \quad (2.81)$$

The output voltage is given by

$$V_o(s) = R_l I_o(s) = -R_l \frac{R_o g_m V_i(s)}{R_o + R_L + 1/sC_2}$$

$$= -\frac{R_l R_o g_m}{R_o + R_L} \times \frac{s}{s + 1/[C_2(R_o + R_L)]} V_i(s) \quad (2.82)$$

Substituting $V_s(s)$ from Eq. (2.81) into Eq. (2.82) and simplifying, we get the voltage transfer function at low frequencies. That is,

$$A(s) = \frac{V_o(s)}{V_i(s)}$$

$$= -\frac{R_l R_i R_o g_m}{(R_s + R_i)(R_o + R_L)} \times \frac{s}{s + 1/[C_1(R_s + R_i)]} \times \frac{s}{s + 1/[C_2(R_o + R_L)]}$$

which gives the low break frequencies and high-pass gain as

$$f_{C1} = \frac{1}{2\pi C_1(R_s + R_i)} \quad (2.83)$$

$$f_{C2} = \frac{1}{2\pi C_2(R_o + R_L)} \quad (2.84)$$

$$A_{PB} = -\frac{R_l R_i R_o g_m}{(R_s + R_i)(R_o + R_L)} \quad (2.85)$$

We can notice from Eqs. (2.83) and (2.84) that the Thevenin’s equivalent resistances for $C_1$ and $C_2$ are $R_{C1} = (R_s + R_i)$ and $R_{C2} = (R_o + R_L)$, respectively. The corresponding time constants are $\tau_{C1} = C_1 R_{C1}$ and $\tau_{C2} = C_2 R_{C2}$. Either $f_{C1}$ or $f_{C2}$ will be the dominant low cutoff (or 3-dB) frequency $f_L$. For a voltage
amplifier, the input resistance \( R_i \) is normally much higher than the output resistance \( R_o \), so \( f_{C2} > f_{C1} \) and \( f_C = f_L \) (low cut-off frequency). The steps in setting the low cutoff (or 3-dB) frequency are as follows:

**Step 1.** Set the low 3-dB frequency \( f_L \) with the capacitor that has the lowest resistance.

**Step 2.** Keep the other frequencies sufficiently lower than \( f_L \) so that interactions are minimal. Separating the first break frequency \( f_{L1} \) from the second break frequency \( f_{L2} \) by a decade is generally adequate, as long as the other frequencies are kept lower than \( f_{L2} \) by means of the following relations:

\[
\begin{align*}
  f_{L1} &= f_L \\
  f_{L2} &= f_L / 10 \\
  f_{L3} &= f_L / 20 \\
  f_{L4} &= f_L / 20
\end{align*}
\]

That is, \( f_{L1} = f_L \) for Thevenin’s equivalent resistance \( R_{C1} \), \( f_{L2} = f_L / 10 \) for Thevenin’s equivalent resistance \( R_{C2} \), and \( f_{L3} = f_L / 20 \) for Thevenin’s equivalent resistance \( R_{C3} \), where \( R_{C1} < R_{C2} < R_{C3} \). Since we are interested only in keeping other frequencies far away from the cutoff frequency \( f_L \) and since a wider separation would require a higher capacitor value, it is not necessary to keep a separation of one decade between subsequent frequencies.

### 2.7.2 High-Frequency Transfer Function Method

At high frequencies greater than \( f_H > 15 \text{ kHz} \), the bypass and coupling capacitors, which are on the order of 10 μF, have reactances on the order of 1 Ω and are essentially short-circuited. Let us assume that the coupling capacitances are large so that the capacitors are effectively short-circuited. Thus, the high-frequency behavior is determined solely by the internal capacitors of the amplifier. The equivalent circuit for determining the high break frequencies is shown in Fig. 2.23. Capacitance \( C \), between the input and output terminals of the amplifier, can be replaced by Miller’s equivalent capacitances. Therefore, we can find the frequency response by s-domain analysis or by Miller’s capacitor method.

As we did for the low break frequencies, we will derive the transfer function for high break frequencies. Applying Kirchhoff’s current law at nodes 1 and 2, we get the following equations in Laplace’s domain of \( s \):

\[
\begin{align*}
  \frac{V_e - V_i}{R_s} &= \frac{V_i}{R_i} + V_i C_i s + (V_i - V_o) C_s \\
  g_m V_i + \frac{V_o}{R_o} + \frac{V_o}{R_L} + V_o C_o s + (V_o - V_i) C_s &= 0
\end{align*}
\]

\[ (2.86) \]

\[ (2.87) \]
Equations (2.86) and (2.87) can be solved to give the voltage transfer function

\[ V_o(s) = \frac{-(g_m - C_s)R_1R_2/R_s}{1 + s[R_1(C_i + C) + R_2(C_o + C) + g_mCR_1R_2] + s^2R_1R_2(C_iC_o + C_iC + C_oC)} \]  

(2.88)

where \( R_1 = (R_s \| R_i) \) and \( R_2 = (R_o \| R_L) \). The denominator of Eq. (2.88) has two poles. If \( p_1 \) and \( p_2 \) are the two poles, the denominator can be written as

\[ D(s) = \left(1 + \frac{s}{p_1}\right)\left(1 + \frac{s}{p_2}\right) = 1 + s\left(\frac{1}{p_1} + \frac{1}{p_2}\right) + \frac{s^2}{p_1p_2} \]  

(2.89)

If the poles are widely separated, which is generally the case, and \( p_1 \) is assumed to be the dominant pole, while \( p_2 \) is assumed large, then Eq. (2.89) can be approximated by

\[ D(s) = 1 + \frac{s}{p_1} + \frac{s^2}{p_1p_2} \]  

(2.90)

Equating the coefficients of \( s \) in Eq. (2.88) to those in Eq. (2.90) yields

\[ p_1 = \frac{1}{R_1(C_i + C) + R_2(C_o + C) + g_mCR_1R_2} \]  

(2.91)

Equating the coefficients of \( s^2 \) in Eq. (2.88) to those in Eq. (2.90) yields

\[ p_2 = \frac{R_1(C_i + C) + R_2(C_o + C) + g_mCR_1R_2}{R_1R_2(C_iC_o + C_iC + C_oC)} \]  

(2.92)

In practice, the value of \( C \) is higher than that of \( C_i \) and \( C_o \), and Eqs. (2.91) and (2.92) can be simplified further as follows:

\[ p_1 \approx \frac{1}{g_mCR_1R_2} \]  

(2.93)

\[ p_2 = \frac{g_mC}{C_iC_o + C_iC + C_oC} \]  

(2.94)

Here \( p_1 \) and \( p_2 \) correspond to the break frequencies \( \omega_1 \) and \( \omega_2 \) in frequency domain.

**NOTES**

1. The dominant pole \( p_1 \) decreases as \( C \) increases, whereas \( p_2 \) increases as \( C \) increases. Therefore, increasing \( C \) causes the poles to split apart, possibly making \( p_1 \) the dominant pole.
2. If \( C >> C_i \) and \( C >> C_o \), Eq. (2.94) can be approximated as

\[ p_2 \approx \frac{g_mC}{C(C_i + C_o)} = \frac{g_m}{C_i + C_o} \]  

(2.95)
3. If there is no feedback capacitance \( C = 0 \), Eq. (2.88) gives the following poles:

\[
\begin{align*}
p_1 &= \frac{1}{C_i R_1} \quad & \text{(2.96)} \\
p_2 &= \frac{1}{C_o R_2} \quad & \text{(2.97)}
\end{align*}
\]

### 2.7.3 Miller’s Capacitor Method

Assuming that the current through capacitor \( C \) in Fig. 2.23 is very small compared to the voltage-dependent current source \( g_m V_i \), the output voltage in Laplace’s domain is

\[
V_o(s) = -g_m V_i(s) (R_o \ || R_L)
\]

The current \( I_f(s) \) flowing through \( C \) (from the left side to the right side) is given by

\[
I_f(s) = sC[V_i(s) - V_o(s)] = sC[V_i(s) + g_m V_i(s) (R_o \ || R_L)]
\]

\[
= sC[1 + g_m (R_o \ || R_L)] V_i(s) = sC_m V_i(s)
\]

where \( C_m = C[1 + g_m (R_o \ || R_L)] \) \quad (2.98)

The current \(-I_f(s)\) flowing through \( C \) (from the right side to the left side) is given by

\[
-I_f(s) = sC[V_o(s) - V_i(s)] = sC\left[V_o(s) + \frac{V_o(s)}{g_m (R_o \ || R_L)}\right]
\]

\[
= sC\left[1 + \frac{1}{g_m (R_o \ || R_L)}\right] V_o(s) = sC_n V_o(s)
\]

where \( C_n = C\left[1 + \frac{1}{g_m (R_o \ || R_L)}\right] \) \quad (2.99)

Thus, capacitor \( C \), which is connected between the input and output terminals of a high-gain amplifier with 180° phase reversal, can be replaced by a shunt capacitor \( C_m \) on the input side and another capacitor \( C_n \) on the output side. This arrangement is shown in Fig. 2.24. The value of \( C \) is seen on the input side as a multiplying factor almost equal to the voltage gain \( g_m (R_o \ || R_L) \). This effect, known as **Miller’s effect**, is

![Miller's equivalent high cutoff circuit](image)
Finding the coupling capacitors to set the low cutoff frequency

(a) The amplifier in Fig. 2.21(a) has

\[ g_m = 50 \text{ mA/V}, \quad R_s = 2 \text{ k\Omega}, \quad R_i = 8 \text{ k\Omega}, \quad R_o = 15 \text{ k\Omega}, \quad R_L = 10 \text{ k\Omega}, \]

\[ C_i = 5 \text{ pF}, \quad \text{and} \quad C_o = 1 \text{ pF}. \]

Calculate the coupling capacitances \( C_1 \) and \( C_2 \) in order to set the low 3-dB frequency at \( f_L = 1.5 \text{ kHz} \), the pass-band gain \( A_{PB} \), and the feedback capacitance \( C \) so that the frequency of the dominant pole is \( f_H = 100 \text{ kHz} \).

(b) Use Miller’s method to find the high cutoff frequencies.

(c) Use PSpice/SPICE to plot the voltage gain against the frequency.

SOLUTION

We have

\[ R_s + R_i = 2 \text{ k\Omega} + 8 \text{ k\Omega} = 10 \text{ k\Omega} \]

\[ R_o + R_L = 15 \text{ k\Omega} + 10 \text{ k\Omega} = 25 \text{ k\Omega} \]

\[ R_i = R_i \parallel R_l = 2 \text{ k\Omega} \parallel 8 \text{ k\Omega} = 1.6 \text{ k\Omega} \]

and

\[ R_o = R_o \parallel R_L = 15 \text{ k\Omega} \parallel 10 \text{ k\Omega} = 6 \text{ k\Omega} \]

(a) \( f_L = 1.5 \text{ kHz} \) and \( f_H = 100 \text{ kHz} \). Since

\[ (R_s + R_i) = 10 \text{ k\Omega} < (R_o + R_L) = 25 \text{ k\Omega} \]
let us set \( f_{C1} \) equal to the low 3-dB frequency. That is, \( f_{C1} = f_L = 1.5 \text{ kHz} \). The capacitance can be found from Eq. (2.83):

\[
C_1 = \frac{1}{2\pi f_{C1}(R_s + R_i)} = \frac{1}{2\pi \times 1.5 \times (2 k + 8 k)} = 0.01 \text{ \( \mu \)F}
\]

Let \( f_{C2} = f_{C1}/10 = 1.5 k/10 = 150 \text{ Hz} \). We get \( C_2 \) from Eq. (2.84):

\[
C_2 = \frac{1}{2\pi f_{C2}(R_o + R_L)} = \frac{1}{2\pi \times 150 \times (15 k + 10 k)} = 0.04 \text{ \( \mu \)F}
\]

The pass-band gain is

\[
A_{PB} = \frac{R_s R_i g_m}{(R_s + R_i)(R_o + R_L)} = \frac{8 k \times 10 k \times 15 k \times 50 \text{ mA/V}}{(2 k + 8 k)(15 k + 10 k)} = -240
\]

From Eq. (2.93), we get the capacitance \( C \) for the dominant pole:

\[
C = \frac{1}{2\pi f_{H1} g_m R_1 R_2} = \frac{1}{2\pi \times 100 k \times 50 \text{ mA/V} \times 1.6 k \times 6 k} = 3.32 \text{ pF}
\]

From Eq. (2.91), we get

\[
f_{H1} = \frac{10^{12}}{2\pi [1.6 k \times (5 + 3.32) + 6 k \times (1 + 3.32) + 50 \text{ mA/V} \times 3.32 \times 1.6 k \times 6 k]}
\]

\[= 97.47 \text{ kHz}\]

From Eq. (2.92), we get

\[
f_{H2} = \frac{[1.6 k \times (5 + 3.32) + 6 k \times (1 + 3.32) + 50 \text{ mA/V} \times 3.32 \times 1.6 k \times 6 k] \times 10^{12}}{2\pi [1.6 k \times 6 k \times (5 \times 1 + 5 \times 3.32 + 1 \times 3.32)]}
\]

\[= 1.09 \text{ GHz}\]
2.7.4 Low-Frequency Short-Circuit Method

As we have done in Sec. 2.7.1, we can determine the low cutoff frequencies and the pass-band voltage gain from the low-frequency voltage transfer function $A(s)$. In many cases, the analysis becomes laborious and it is not a simple matter to find $A(s)$. In such cases, an approximate value of the low 3-dB break frequency $f_L$ can be found by the short-circuit method.

---

(b) From Eq. (2.98),

$$C_m = C[1 + \frac{1}{g_m(R_o || R_L)}] = (3.32 \text{ pF})(1 + 50 \text{ mA} / 6 \text{ k}) = 999.3 \text{ pF}$$

From Eq. (2.99),

$$C_n = C\left[1 + \frac{1}{g_m(R_o || R_L)}\right] = (3.32 \text{ pF})\left(1 + \frac{1}{300}\right) = 3.33 \text{ pF}$$

From Eqs. (2.100) and (2.101), we get

$$f_{H1} = \frac{1}{2\pi(C_1 + C_m)(R_s || R_i)} = \frac{1}{2\pi \times (5 \text{ pF} + 999.3 \text{ pF}) \times 1.6 \text{ k}} = 99.05 \text{ kHz}$$

$$f_{H2} = \frac{1}{2\pi(C_o + C_n)(R_o || R_L)} = \frac{1}{2\pi \times (1 \text{ pF} + 3.33 \text{ pF}) \times 6 \text{ k}} = 6.13 \text{ MHz}$$

Thus, Miller’s capacitor method gives $f_{H1} = 99.05 \text{ kHz}$, compared to 97.47 kHz calculated by $s$-domain analysis. However, $f_{H2} = 6.13 \text{ MHz}$, compared to 1.09 GHz. The error is due to the fact that Miller’s method does not take into account the effect of pole splitting.

(c) The circuit for PSpice simulation is shown in Fig. 2.25. Let us assume an input voltage of $v_s = 10 \text{ mV}$.

The results of the simulation are shown in Fig. 2.26, which gives $A_{mid} = 221.7$ (expected value is 240), $f_L = 1.376 \text{ kHz}$ (expected value is 1.5 kHz) at $|A(j\omega)| = 0.707 \times 221.7 = 156.7$, and $f_H = 107.23 \text{ kHz}$ (expected value is 100 kHz) at $|A(j\omega)| = 0.707 \times 221.7 = 156.7$.

---

2.7.4 Low-Frequency Short-Circuit Method

As we have done in Sec. 2.7.1, we can determine the low cutoff frequencies and the pass-band voltage gain from the low-frequency voltage transfer function $A(s)$. In many cases, the analysis becomes laborious and it is not a simple matter to find $A(s)$. In such cases, an approximate value of the low 3-dB break frequency $f_L$ can be found by the short-circuit method.
Let us assume that the voltage gain of an amplifier has two low break frequencies. Then, applying Eq. (2.59) for the high-pass characteristic with two break frequencies, we get

\[ A(j\omega) = \frac{A_{v\text{high}}}{(1 + \omega_{L1}/s)(1 + \omega_{L2}/s)} \]  

(2.104)

where \( A_{v\text{high}} = A_{PB} \) is the high-frequency gain and \( \omega_{L1} \) and \( \omega_{L2} \) are the two break frequencies. At the low 3-dB frequency, the denominator of Eq. (2.104) should be such that \( A_{PB} = 1/\sqrt{2} = 0.707. \) That is,

\[ \left| \left( 1 + \frac{\omega_{L1}}{j\omega} \right) \left( 1 + \frac{\omega_{L2}}{j\omega} \right) \right| = \sqrt{2} \]

or

\[ \left| 1 - j \frac{\omega_{L1} + \omega_{L2}}{\omega} - \frac{\omega_{L1}\omega_{L2}}{\omega^2} \right| = \sqrt{2} \]

If \( \omega > \sqrt{\omega_{L1}\omega_{L2}} \), the product term can be neglected. The imaginary term will be unity when

\[ \omega_L = \omega = \omega_{L1} + \omega_{L2} = \frac{1}{\tau_{C1}} + \frac{1}{\tau_{C2}} \]  

(2.105)

where \( \omega_L \) is the effective low 3-dB frequency and is the sum of the reciprocals of the time constants \( \tau_{C1} \) and \( \tau_{C2} \). For a circuit with multiple capacitors, the time constant \( \tau_{Ck} \) for the \( k \)th capacitor is found by considering one capacitor at a time while setting the other capacitors to \( \infty \) (or effectively short-circuiting them). This method assumes that only one capacitor contributes to the voltage gain. Thus, the low 3-dB frequency is determined from the effective time constant of all capacitors. That is,

\[ f_L = \frac{1}{2\pi} \sum_{k=1}^{n} \frac{1}{\tau_{Ck}} = \frac{1}{2\pi} \sum_{k=1}^{n} \frac{1}{C_k R_{Ck}} \]  

(2.106)

where \( \tau_{Ck} \) is the time constant due to the \( k \)th capacitor only and \( R_{Ck} \) is Thevenin’s equivalent resistance presented to \( C_k \). One cutoff frequency will push the next higher frequency toward the right and thereby influence the effective cutoff frequency of the amplifier. If one of the break frequencies is larger than the other frequencies by a factor of 5 to 10, \( f_L \) can be approximated by the highest frequency—say \( f_{C1} \). If \( f_L = f_{C1} \), the error introduced will usually be less than 10%. Otherwise, the error could be as high as 20%.

Let us apply this method to the circuit in Fig. 2.21(b). We will consider the effect of \( C_1 \) only; \( C_2 \) is short-circuited, as shown in Fig. 2.27(a). Thevenin’s equivalent resistance presented to \( C_1 \) is

\[ R_{C1} = R_S + R_i \]

Thus, the break frequency due to \( C_1 \) only is

\[ f_{C1} = \frac{1}{2\pi R_{C1} C_1} = \frac{1}{2\pi (R_S + R_i) C_1} \]  

(2.107)
The equivalent circuit, with $C_1$ considered to be short-circuited, is shown in Fig. 2.27(b). Thevenin’s equivalent resistance presented to $C_2$ is given by

$$R_{C_2} = R_o + R_L$$

The break frequency due to $C_2$ only is given by

$$f_{C_2} = \frac{1}{2\pi R_{C_2} C_2} = \frac{1}{2\pi (R_o + R_L) C_2}$$  \hspace{1cm} (2.108)$$

Therefore, the effective 3-dB frequency can be found from

$$f_L = f_{C_1} + f_{C_2}$$  \hspace{1cm} (2.109)$$

In general, one of the low break frequencies is set to the desired 3-dB frequency $f_L$ and the other frequencies are made much lower, normally separated by a decade. That is, if $f_L \approx f_{C_1}$, then $f_{C_2} = f_L/10$.

The steps in setting the low 3-dB frequency are as follows:

**Step 1.** Draw the equivalent circuit with all but one capacitor shorted.

**Step 2.** Find Thevenin’s equivalent resistance for each capacitor.

**Step 3.** Set the low 3-dB frequency $f_L$ with the capacitor that has the lowest resistance. This will give the smallest capacitor value.

**Step 4.** Keep the other frequencies sufficiently lower than $f_L$ so that interactions are minimal. That is, if $f_{C_1} = f_L$ for Thevenin’s equivalent resistance $R_{C_1}$, $f_{C_2} = f_L/10$ for Thevenin’s equivalent resistance $R_{C_2}$, and $f_{C_3} = f_L/20$ for Thevenin’s equivalent resistance $R_{C_3}$, where $R_{C_1} < R_{C_2} < R_{C_3}$.

### 2.7.5 High-Frequency Zero-Value Method

Let us assume that the voltage gain of an amplifier has two high break frequencies. Then, applying Eq. (2.49) for $s = j\omega$ and for the low-pass characteristic with two break frequencies, we get

$$A(j\omega) = \frac{A_{v(low)}}{(1 + j\omega/\omega_1)(1 + j\omega/\omega_2)}$$  \hspace{1cm} (2.110)$$
where $A_{v(low)} = A_{PB}$ is the low-frequency gain and $\omega_{H1}$ and $\omega_{H2}$ are the two high break frequencies. At the high 3-dB frequency, the denominator of Eq. (2.110) should be such that $A_{PB} = 1/\sqrt{2} = 0.707$. That is,

$$\left| \left( 1 + \frac{j\omega}{\omega_{H1}} \right) \left( 1 + \frac{j\omega}{\omega_{H2}} \right) \right| = \sqrt{2}$$

or

$$\left| 1 - \left( \frac{\omega}{\omega_{H1}} \right) \left( \frac{\omega}{\omega_{H2}} \right) + j\omega \left( \frac{1}{\omega_{H1}} + \frac{1}{\omega_{H2}} \right) \right| = \sqrt{2}$$

If $\omega < \sqrt{\omega_{H1}\omega_{H2}}$, the product term can be neglected. The imaginary term will be unity when

$$\frac{1}{\omega_{H}} = \frac{1}{\omega} = \frac{1}{\omega_{H1}} + \frac{1}{\omega_{H2}} = \tau_{C1} + \tau_{C2} \quad (2.111)$$

where $\omega_{H}$ is the effective high 3-dB frequency and is the reciprocal of the sum of the time constants $\tau_{C1}$ and $\tau_{C2}$. For a circuit with multiple capacitors, the time constant $\tau_{Cj}$ for the $j$th capacitor is found by considering one capacitor at a time while setting the other capacitors to zero (or effectively open-circuiting them). Thus, the high 3-dB frequency is determined from the effective time constant of all capacitors. That is,

$$f_{H} = \frac{1}{2\pi \sum_{j=1}^{n}\tau_{Cj}} = \frac{1}{2\pi \sum_{j=1}^{n}C_{j}R_{Cj}} \quad (2.112)$$

where $\tau_{Cj}$ is the time constant due to the $j$th capacitor only and $R_{Cj}$ is Thevenin’s equivalent resistance presented to $C_{j}$.

Let us apply this method to the circuit in Fig. 2.23. The equivalent circuit, with $C$ and $C_{o}$ open-circuitted, is shown in Fig. 2.28(a). The resistance seen by $C_{i}$ is given by

$$R_{Ci} = (R_{s} \parallel R_{i})$$

The equivalent circuit, with $C$ and $C_{i}$ open-circuitted, is shown in Fig. 2.28(b). The resistance faced by $C_{o}$ is given by

$$R_{Co} = (R_{o} \parallel R_{L})$$

**FIGURE 2.28** Small signal equivalent circuits for zero-value method
The equivalent circuit, with $C_i$ and $C_o$ open-circuited, is shown in Fig. 2.28(c). Let us replace $C$ with voltage source $v_x$, as shown in Fig. 2.28(d). Then, applying Kirchhoff’s voltage law (KVL), we get

$$v_x = v_i + (R_o \parallel R_L)(i_x + g_m v_i) = (R_s \parallel R_i)i_x + (R_o \parallel R_L)[i_x + g_m i_x (R_s \parallel R_L)]$$

$$= (R_o \parallel R_L) + (R_s \parallel R_i)[1 + g_m (R_o \parallel R_L)]i_x$$

which gives Thevenin’s equivalent resistance seen by $C$ as

$$R_{Cc} = \frac{v_x}{i_x} = (R_o \parallel R_L) + (R_s \parallel R_i)[1 + g_m (R_o \parallel R_L)]$$

$$= R_{L(\text{eff})} + R_{i(\text{eff})}[1 + g_m R_{L(\text{eff})}]$$

(2.113)

where $R_{i(\text{eff})} = (R_s \parallel R_i)$ and $R_{L(\text{eff})} = (R_o \parallel R_L)$. Thus, the high 3-dB frequency $f_H$ is given by

$$f_H = \frac{1}{2\pi (R_{Ci}C_i + R_{Co}C_o + R_{Cc}C)}$$

(2.114)

The steps in applying the zero-value method are as follows:

**Step 1.** Determine Thevenin’s resistance seen by each capacitor acting alone while the other capacitors are open-circuited.

**Step 2.** Calculate the time constant due to each capacitor.

**Step 3.** Add all the time constants to find the effective time constant:

$$\tau_H = \tau_{H1} + \tau_{H2} + \cdots + \tau_{Hi}$$

**Step 4.** Find the high 3-dB frequency from Eq. (2.112).

**Step 5.** To set the high 3-dB frequency to a desired value, add an extra capacitor $C_x$ in parallel with $C$ so that the effective shunt capacitance is $C_{\text{eff}} = C_x + C$.

### 2.7.6 Midband Voltage Gain

If the frequency is high enough that the coupling capacitors offer low impedances and behave almost as if they were short-circuited but low enough that the high-frequency capacitors of the transistor offer very high impedances, the voltage gain is the pass-band gain. The equivalent circuit for midband voltage gain, with coupling and bypass capacitors short-circuited and high-frequency capacitors open-circuited, is shown in Fig. 2.29. We can find the midband voltage gain as follows:

$$A_{PB} = \frac{v_o}{v_s} = -g_m (R_o \parallel R_L) \frac{R_i}{R_s + R_i}$$

(2.115)
EXAMPLE 2.11

Finding the 3-dB break frequencies by the short-circuit and zero-value methods  

The amplifier in Fig. 2.21(a) has \( g_m = 50 \text{ mA/V}, R_s = 2 \text{ k}\Omega, R_i = 8 \text{ k}\Omega, R_o = 15 \text{ k}\Omega, R_L = 10 \text{ k}\Omega, C_i = 5 \text{ pF}, \) and \( C_o = 1 \text{ pF} \). Use the following capacitor values: \( C_1 = 0.01 \mu\text{F}, C_2 = 0.04 \mu\text{F}, \) and \( C = 3.32 \mu\text{F} \).

(a) Use the short-circuit method to find the low 3-dB frequency \( f_L \).
(b) Use the zero-value method to find the high 3-dB frequency \( f_H \).
(c) Find the pass-band voltage gain \( A_{PB} \).

SOLUTION

(a) \( R_{C1} = R_s + R_i = 2 \text{ k} + 8 \text{ k} = 10 \text{ k}\Omega, R_{C2} = R_o + R_L = 15 \text{ k} + 10 \text{ k} = 25 \text{ k}\Omega \)

From Eq. (2.107), we get \( f_{C1} = 1/(2\pi C_1 R_{C1}) = 1/(2\pi \times 0.01 \mu\text{F} \times 10 \text{ k}\Omega) = 1.592 \text{ kHz} \).

From Eq. (2.108), we get \( f_{C2} = 1/(2\pi C_2 R_{C2}) = 1/(2\pi \times 0.04 \mu\text{F} \times 25 \text{ k}\Omega) = 159.2 \text{ kHz} \).

From Eq. (2.109), we get \( f_L = f_{C1} + f_{C2} = 1.592 \text{ k} + 159.2 = 1.751 \text{ kHz} \).

(b) \( R_{C1} = R_s \parallel R_i = 2 \text{ k} \parallel 8 \text{ k} = 1.6 \text{ k}\Omega \)

\( R_{C0} = R_o \parallel R_L = 15 \text{ k} \parallel 10 \text{ k} = 6 \text{ k}\Omega \)

Using Eq. (2.113),

\[
R_{Cc} = R_o \parallel R_L + (R_s \parallel R_i) [1 + g_m (R_o \parallel R_L)] = 6 \text{ k}\Omega + 1.6 \text{ k}\Omega + (1 + 50 \times 10^{-3} \times 6 \text{ k}\Omega) = 487.6 \text{ k}\Omega
\]

From Eq. (2.114), we get

\[
f_H = \frac{1}{2\pi (R_{C1} C_i + R_{C0} C_o + R_{Cc} C)}\]

\[
= \frac{1}{2\pi \times (1.6 \text{ k} \times 5 \times 10^{-12} + 6 \text{ k} \times 1 \times 10^{-12} + 487.6 \text{ k} \times 3.32 \times 10^{-12})}
\]

\( = 97.47 \text{ kHz} \)

(c) From Eq. (2.115), we get the pass-band voltage gain

\[
A_{PB} = -g_m (R_o \parallel R_L) \left[ \frac{R_l}{(R_o + R_l)} \right] = -50 \times 10^{-3} \times [6 \text{ k} \times 8 \text{ k}/(2 \text{ k} + 8 \text{ k})] = -240 \text{ V/V}
\]

**NOTE:** The short-circuit method gives the low 3-dB frequency \( f_L = 1.751 \text{ kHz} \) (designed for 1.5 kHz), and the zero-value method gives the high 3-dB frequency \( f_H = 97.47 \text{ kHz} \) (designed for 100 kHz). In Example 2.10, Miller’s capacitor method gives \( f_H = 99.05 \text{ kHz} \) compared to 97.47 kHz from \( s \)-domain analysis.
2.7.7 Multistage Amplifiers

Multistage amplifiers are often used to meet voltage gain, frequency range, input impedance, and/or output impedance requirements. In this section we will apply the short-circuit and zero-value methods to determine the cutoff frequencies of multistage amplifiers. Some equations will be similar to those used in the preceding sections because the equivalent circuits for the amplifiers are similar to those encountered previously.

When a capacitor $C_k$ is connected between the input and the output of an amplifier, it greatly influences the high 3-dB frequency. It is often necessary to find the time constant for $C_k$, and we will derive a generalized equation. Let us consider the circuit of Fig. 2.30(a). If the capacitor $C_k$ is replaced by a voltage source $v_x$, the equivalent circuit is shown in Fig. 2.30(b). Using KVL, we get

$$v_x = R_i i_x + R_L (i_x + g_m v_i) = R_i i_x + R_L (i_x + g_m R_i i_x)$$

$$= [R_L + R_i (1 + g_m R_L)] i_x$$

which gives Thevenin’s equivalent resistance faced by $C_k$ as

$$R_{eq} = \frac{v_x}{i_x} = R_L + R_i (1 + g_m R_L) \tag{2.116}$$

---

**KEY POINTS OF SECTION 2.7**

- The method of s-domain analysis can be used to determine the transfer function and the frequency characteristics of an amplifier. However, the analysis can be laborious, especially for a circuit with more than three capacitors.

- Miller’s capacitance method is a quick but approximate method for determining the high cutoff frequency.

- The short-circuit method gives the low 3-dB break frequency, and the zero-value method gives the high 3-dB break frequency. These are simple but effective methods for determining the break frequencies of amplifiers.

- In the short-circuit method, the time constant $\tau_{C_k}$ for the $k$th capacitor is determined by considering one capacitor at a time while setting other capacitors to $\infty$ (or effectively short-circuiting them). The effective low 3-dB frequency is the sum of the reciprocal of individual time constants.

- In the zero-value method, the time constant $\tau_{C_j}$ for the $j$th capacitor is determined by considering one capacitor at a time while setting the other capacitors to zero (or effectively open-circuiting them). The effective high 3-dB frequency is the reciprocal of the sum of the individual time constants.
2.8 PSpice/SPICE Amplifier Models

Amplifiers can be modeled in PSpice/SPICE as linear controlled sources [4]. However, these models will not exhibit the nonlinear characteristics expected in practical amplifiers. The PSpice/SPICE results must be interpreted in relation to the practical limits of a particular type of amplifier. For a current-controlled source, a dummy voltage source of 0 (say, \( V_x = 0 \)) is inserted to monitor the controlling current, which gives the output current or voltage. The controlling current is assumed to flow from the positive node of \( V_x \), through the voltage source \( V_x \), to the negative node of \( V_x \).

2.8.1 Voltage Amplifier

A voltage amplifier can be modeled as a voltage-controlled voltage source (VCVS). The symbol for a VCVS, as shown in Fig. 2.31(a), is \( E \). The linear form is

\[
E\text{name} \ N+ \ N- \ NC+ \ NC- \ (voltage \ gain) \ value
\]

\( N+ \) and \( N- \) are the positive and negative output nodes, respectively. \( NC+ \) and \( NC- \) are the positive and negative nodes, respectively, of the controlling voltage.

2.8.2 Current Amplifier

A current amplifier can be modeled as a current-controlled current source (CCCS). The symbol of a CCCS, as shown in Fig. 2.31(b), is \( F \). The linear form is

\[
F\text{name} \ N+ \ N- \ VX \ (current \ gain) \ value
\]

\( N+ \) and \( N- \) are the positive and negative nodes, respectively, of the output (current) source.

\[\text{FIGURE 2.31} \ Dependent sources\]

---

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
2.8.3 Transconductance Amplifier

A transconductance amplifier can be modeled as a voltage-controlled current source (VCCS). The symbol of a VCCS, as shown in Fig. 2.31(c), is $G$. The linear form is

\[ G_{N+ N-} NC+ NC- \text{ (transconductance value)} \]

$N+ \text{ and } N-$ are the positive and negative output nodes, respectively. $NC+ \text{ and } NC-$ are the positive and negative nodes, respectively, of the controlling voltage.

2.8.4 Transimpedance Amplifier

A transimpedance amplifier can be modeled as a current-controlled voltage source (CCVS). The symbol of a CCVS, as shown in Fig. 2.31(d), is $H$. The linear form is

\[ H_{N+ N-} VX \text{ (transimpedance value)} \]

$N+ \text{ and } N-$ are the positive and negative nodes, respectively, of the output (voltage) source.

2.9 Amplifier Design

So far we have regarded amplifiers as parts of a system. Several amplifiers may be cascaded to meet some design specifications. However, viewed from the input and output sides, cascaded amplifiers may be represented by a single equivalent amplifier. That is, an amplifier may consist of one or more internal amplifiers. At this stage of the course, amplifier design will be at the system level rather than at the level of the internal components of an amplifier itself, which we will cover in Chapters 7 and 8. This chapter has illustrated a number of design examples relating to each topic area. The circuit topology was given, and the design task was mainly to find the component values. Often a designer has to choose the circuit topology, which generally requires evaluating alternative solutions. The following sequence (or process) is recommended for the design of amplifiers at the system level:

**Step 1.** Study the design problem.

**Step 2.** Identify the design specifications: input resistance, output resistance, gain, and bandwidth requirements.

**Step 3.** Establish a design strategy, and find the functional block diagram solution. Identify the type and number of amplifiers to be used. Evaluate alternative methods of solving the design problem.

**Step 4.** Find the circuit-level solution through such means as circuit topologies and hand analysis using ideal amplifier models. Analysis and synthesis may be necessary to find the component values.

**Step 5.** Evaluate your design by using more realistic amplifier models, and modify your design values, if necessary.

**Step 6.** Carry out PSpice/SPICE verification using a complex circuit model, and get the worst-case results given your components and parameter variations. Modify your design, if needed.
Step 7. Get a cost estimate of the project, and have a plan for component layout so that the project requires the minimum fabrication time and is least expensive.

Step 8. Build a prototype unit in the lab, and take measurements to verify your design. Modify your design, if needed.

**EXAMPLE 2.12**

**Illustration of design steps** Two signals are coming from two different transducers: \( v_1 = 180 \text{ mV} \) to \( 200 \text{ mV} \) with \( R_{s1} = 2 \text{ k}\Omega \) and \( v_2 = 150 \text{ mV} \) to \( 170 \text{ mV} \) with \( R_{s2} = 2 \text{ k}\Omega \). Amplify the differential voltage so that the output voltage is \( v_o = 200(v_1 - v_2) \). The gain variation should be less than \( \pm 3\% \). The load resistance is \( R_L = 5 \text{ k}\Omega \). Determine the specifications of the amplifier.

**SOLUTION**

Step 1. Study the design problem. \( v_1 = 180 \text{ mV} \) to \( 200 \text{ mV} \) with \( R_{s1} = 2 \text{ k}\Omega \), and \( v_2 = 150 \text{ mV} \) to \( 170 \text{ mV} \) with \( R_{s2} = 2 \text{ k}\Omega \).

Step 2. Identify the design specifications. \( A_v = 200 \pm 3\% \), \( R_L = 5 \text{ k}\Omega \), and there is no bandwidth limit.

Step 3. Establish a design strategy, and find the functional block diagram solution. Since the input side will have two voltage signals whose difference is to be amplified, we need a voltage differential amplifier at the input stage. The output of this stage could be either voltage or current, which will be amplified by a gain stage, shown in Fig. 2.32(a).

Step 4. Find the circuit-level solution. We will use two identical transconductance amplifiers to give differential gain because it allows us to add (or subtract) two currents at a node. We will also use a transresistance amplifier at the output side to give the desired voltage gain and a low output resistance. This arrangement is shown in Fig. 2.32(b). Assuming ideal amplifiers of \( G_{ms1} = G_{ms2} = G_{ms} \), the output voltage is given by

\[
v_o = (G_{ms1}v_1 - G_{ms2}v_2)Z_{mo} = Z_{mo}G_{ms}(v_1 - v_2)
\]

which gives \( A_v = Z_{mo}G_{ms} \). Assuming \( G_{ms} = 20 \text{ mA}/\text{V} \), we get

\[
Z_{mo} = \frac{A_v}{G_{ms}} = \frac{200 \text{ V}}{20 \text{ mA}} = 10 \text{ k}\Omega
\]

Step 5. Evaluate your design. Let us take practical amplifiers with input and output resistances as shown in Fig. 2.32(c). Using Eq. (2.35), we can find the effective voltage gain \( A_v \) from

\[
A_v = \frac{R_{i1}}{R_{i1} + R_{s1}} \times \frac{R_{oe}}{R_{oe} + R_{i2}} \times \frac{R_L}{R_L + R_{o2}} Z_{mo} G_{ms} (R_{oe} = R_{o1} \parallel R_{o1}^{'})
\]

Since \( A_v \) will vary with variations in \( R_{i1}, R_{i1}', R_{o2}, Z_{mo}' \) and \( G_{ms} \), let us allow \( \pm 0.5\% \) variation for each of them so that the overall variation is limited to \( \pm 2.5\% \). Assume \( R_{s1}, R_{s2}, \) and \( R_L \) do not vary.

\[
\frac{R_{i1}}{R_{i1} + R_{s1}} = \frac{R_{i1}}{R_{i1} + 2 \text{ k}} = 0.995
\]
FIGURE 2.32 Amplifier design stages for Example 2.12

which gives \( R_{i1} = R'_{i1} \geq 398 \, \Omega \).

\[
\frac{R_{L}}{R_{L} + R_{o2}} = \frac{5 \, k}{5 \, k + R_{o2}} = 0.995
\]

which gives \( R_{o2} \leq 25.1 \, \Omega \).

Let us assume that \( R'_{o1} = 200 \, k \Omega \). Since \( R_{oe} = (R_{o1} \ || \ R'_{o1}) = 100 \, k \Omega \),

\[
\frac{R'_{o1}}{R'_{o1} + R_{i2}} = \frac{100 \, k}{100 \, k + R_{i2}} = 0.995
\]

which gives \( R_{i2} \leq 502 \, \Omega \).

FIGURE 2.33 Circuit for PSpice simulation for Example 2.12
Step 6. Use PSpice/SPICE verification. The results of PSpice simulation for the circuit shown in Fig. 2.33 are as follows:

<table>
<thead>
<tr>
<th>NODE</th>
<th>VOLTAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1)</td>
<td>.2000</td>
</tr>
<tr>
<td>(2)</td>
<td>.1990</td>
</tr>
<tr>
<td>(3)</td>
<td>.1500</td>
</tr>
<tr>
<td>(4)</td>
<td>.1493</td>
</tr>
<tr>
<td>(5)</td>
<td>.4970</td>
</tr>
<tr>
<td>(6)</td>
<td>.4970</td>
</tr>
<tr>
<td>(7)</td>
<td>9.9003</td>
</tr>
<tr>
<td>(8)</td>
<td>9.8508</td>
</tr>
</tbody>
</table>

The output voltage is \( v_o = 9.8796 \text{ V} \), and \( A_v = 9.8508 \text{ V} / (200 \text{ mV} - 150 \text{ mV}) = 197.02 \).

Step 7. Get a cost estimate.

Two identical transconductance amplifiers for the differential stage: \( G_{ms} = 20 \text{ mA/V} \pm 0.5\% \), \( R_{i1} \geq 398 \Omega \), and \( R_{o1} \geq 200 \text{ k}\Omega \). Estimated cost is $1.50.

One transresistance amplifier for the gain stage: \( Z_{mo} = 10 \text{ kV/A} \pm 0.5\% \), \( R_{i2} \leq 502 \Omega \), and \( R_{o2} \leq 25.1 \Omega \). Estimated cost is $1.

Two DC power supplies: \( V_{CC} = V_{EE} = 12 \text{ V} \).

Summary

Amplifiers are normally specified in terms of gain, input resistance, and output resistance. An amplifier can be classified as one of four types: a voltage amplifier, a current amplifier, a transconductance amplifier, or a transimpedance amplifier. The gain relationships of various amplifiers can be related to each other. In addition to amplifying signals, amplifiers can serve as building blocks for other applications, such as impedance matching, negative resistance simulation, inductance simulation, and capacitance multiplication. Cascaded amplifiers are often used to increase the overall gain.

Amplifiers use transistors as amplifying devices. Transistors have internal capacitances and also coupling capacitors for isolating the signal source and the load from DC signals. The gain of practical amplifiers varies with the frequency of the signal source, and amplifiers can be classified based on their frequency response as low-pass or band-pass.

Because it uses coupling, bypass, and transistor capacitors, an amplifier operates within a frequency range called a bandwidth. There are three types of frequency characteristics: low-pass, high-pass, and band-pass. An amplifier normally exhibits a band-pass characteristic. Analysis or design of an amplifier requires computer-aided methods because of the complexity of the circuits and the frequency-dependent parameters involving complex numbers.

In general, a capacitor that forms a series circuit with the input signal limits the low cutoff frequency, whereas a capacitor that forms a parallel circuit limits the upper cutoff frequency. Analysis of low break frequencies can be simplified by the short-circuit method, in which the time constant due to one capacitor is determined by assuming that the other capacitors are effectively short-circuited. This method can be extended to the analysis of multistage amplifiers. The dominant low cutoff (or 3-dB) frequency can be set to one of the low break frequencies. In that case, if one of the cutoff frequencies is less than the other frequencies by a factor of 5 to 10, the error introduced by this method is usually less than 10\%. Otherwise the error could be as high as 20\%.

At high frequencies, any capacitor that is connected between the input and output terminals dominates the frequency response as a result of Miller’s multiplication effect. Miller’s capacitor method, which can be applied to determine the approximate value of the high cutoff frequency, gives a value higher than the actual one. The zero-value method, which assumes that only one capacitor contributes to the circuit response and other capacitors have a value of zero, calculates the high 3-dB cutoff frequency from the effective time constant of all capacitors and gives a conservative estimate of the frequency.
References


Review Questions

1. What are the parameters of an amplifier?
2. What is the purpose of DC biasing of an amplifier?
3. What is the effect of rise time on the frequency response of an amplifier?
4. What is a slew rate?
5. What is the slew rate of a step input voltage?
6. What are the four types of amplifiers?
7. What is the circuit model of a voltage amplifier?
8. What is the open-circuit voltage gain of a voltage amplifier?
9. What is the effect of source resistance on the effective voltage gain of a voltage amplifier?
10. What is an ideal voltage amplifier?
11. What is the circuit model of a current amplifier?
12. What is the short-circuit current gain of a current amplifier?
13. What is the effect of source resistance on the effective current gain of a current amplifier?
14. What is an ideal current amplifier?
15. What is the circuit model of a transconductance amplifier?
16. What is the short-circuit transconductance of a transconductance amplifier?
17. What is the effect of source resistance on the overall voltage gain of a transconductance amplifier?
18. What is the open-circuit transimpedance of an amplifier?
19. What is the effect of source resistance on the effective current gain of a transimpedance amplifier?
20. What is an ideal transimpedance amplifier?
21. What is the effect on the overall gain of cascading amplifiers?
22. What is the principle of negative resistance simulation (see Prob. 2.18)?
23. What is a gyrator (see Prob. 2.21)?
24. What is the frequency response of an amplifier?
25. What is a low-pass amplifier?
26. What is a high-pass amplifier?
27. What is a band-pass amplifier?
28. Which capacitors contribute to the low cutoff frequency of amplifiers?
29. What is the short-circuit method?
30. What are the advantages and disadvantages of the short-circuit method?
31. Which capacitors contribute to the high cutoff frequency of amplifiers?
32. What is Miller’s capacitor method?
33. What are the advantages and disadvantages of Miller’s capacitor method?
34. What is the zero-value method?
35. What are the advantages and disadvantages of the zero-value method?
36. What are the steps involved in applying the zero-value method?

Problems

The symbol □ indicates that a problem is a design problem. The symbol ✶ indicates that you can check the solution to a problem using PSpice/SPICE or Electronics Workbench.

2.2 Amplifier Characteristics

2.1 The measured small-signal values of the linear amplifier shown in Fig. 2.3(a) are as follows: \( v_i = 50 \times 10^{-3} \) sin 1000\( \pi t \), \( i_i = 1 \times 10^{-6} \) sin 1000\( \pi t \), \( v_o = 6.5 \) sin 1000\( \pi t \), and \( R_L = 5 \) kΩ. The DC values are \( V_{CC} = V_{EE} = 15 \) V and \( I_{CC} = I_{EE} = 15 \) mA. Find (a) the values of amplifier parameters \( A_v, A_i, A_o, \) and \( R_i \); (b) the power delivered by the DC supplies \( P_{dc} \) and the power efficiency \( \eta \); and (c) the maximum value of the input voltage so that the amplifier operates within the saturation limits.

2.2 The measured values of the nonlinear amplifier in Fig. 2.4(a) are \( v_o = 5.3 \) V at \( v_i = 21 \) mV, \( v_o = 5.5 \) V at \( v_i = 24 \) mV, and \( v_o = 5.8 \) V at \( v_i = 27 \) mV. The DC supply voltage is \( V_{CC} = 12 \) V, and the saturation limits are \( 2 \leq v_o \leq 11 \) V.

a. Determine the small-signal voltage gain \( A_v \).
b. Determine the DC voltage gain \( A_{dc} \).
c. Determine the limits of input voltage \( v_i \).

2.3 Determine the power gain \( A_p \) of the amplifier for the measured values.

a. \( v_o = 2 \) V, \( v_i = 1 \) mV, \( R_i = 100 \) kΩ, and \( R_L = 10 \) kΩ.
b. \( i_o = 100 \) mA, \( i_i = 1 \) mA, \( R_i = 100 \) Ω, and \( R_L = 1 \) kΩ.

2.3 Amplifier Types

2.4 The voltage amplifier shown in Fig. 2.6(a) has an open-circuit voltage gain of \( A_{vo} = 150 \), an input resistance of \( R_i = 1.8 \) kΩ, and an output resistance of \( R_o = 50 \) Ω. It drives a load of \( R_L = 4.7 \) kΩ. The signal source voltage is \( v_s = 100 \) mV with a source resistance \( R_s = 200 \) Ω.

a. Calculate the effective voltage gain \( A_v = v_o / v_i \), the current gain \( A_i = i_o / i_i \), and the power gain \( A_p = P_L / P_i \).
b. Use PSpice/SPICE to check your results in part (a).

2.5 For the amplifier in Prob. 2.4, what should the load resistance \( R_L \) be for maximum power transfer to the load? Calculate the maximum output (or load) power \( P_{L(max)} \).

2.6 When a load resistance of \( R_L = 1.5 \) kΩ is connected to the output of a voltage amplifier, the output voltage drops by 15%. What is the output resistance \( R_o \) of the amplifier?

2.7 The voltage amplifier shown in Fig. 2.6(a) has an open-circuit voltage gain of \( A_{vo} = 200 \), an input resistance of \( R_i = 100 \) kΩ, and an output resistance of \( R_o = 20 \) Ω. The signal source voltage is \( v_s = 50 \) mV, the source resistance is \( R_s = 1.5 \) kΩ, and the load resistance is \( R_L = 22 \) Ω. Calculate (a) the output voltage \( v_o \), (b) the output power \( P_L \), (c) the effective voltage gain \( A_v = v_o / v_s \), (d) the current gain \( A_i = i_o / i_s \), and (e) the power gain \( A_p = P_L / P_i \).

2.8 An amplifier is required to amplify the output signal from a transducer that produces a voltage signal of \( v_s = 10 \) mV with an internal resistance of \( R_i = 2.5 \) kΩ. The load resistance is \( R_L = 2 \) kΩ to 10 kΩ. The desired output voltage is \( v_o = 5 \) V. The amplifier must not draw more than 1 μA from the transducer.
The variation in output voltage when the load is disconnected should be less than 0.5%. Determine the design specifications of the amplifier.

2.9 An amplifier is required to give a voltage gain of $A_v = 100 \pm 1.5\%$. The source resistance is $R_s = 500 \Omega$ to $5 \text{k} \Omega$, and the load resistance is $R_L = 5 \text{k} \Omega$ to $20 \text{k} \Omega$. Determine the design specifications of the amplifier.

2.10 A resistance $R$ is connected between the input and output terminals of a voltage amplifier, as shown in Fig. 2.6. The input voltage signal is $v_i = 20 \text{ mV}$ with an internal resistance of $R_s = 1.5 \text{k} \Omega$.

a. Derive an expression for the input resistance $R_x = v_i / i_s$.

b. Calculate $R_x$ and $i_s$ for $R_1 = 50 \text{k} \Omega$, $R_o = 75 \Omega$, $A_{vo} = 2$, and $R = 10 \text{k} \Omega$.

c. Design an amplifier circuit that will simulate a negative resistance so that the input current drawn from the source is $|i_s| \leq 2.5 \mu\text{A}$.

2.11 A capacitor C is connected between the input and output terminals of a voltage amplifier, as shown in Fig. P2.11(a). The peak input voltage is $V_{s(\text{peak})} = 20 \text{ mV}$ with an internal resistance of $R_s = 1.5 \text{k} \Omega$, and the signal frequency is $f_s = 100 \text{ Hz}$.

a. Derive an expression for the input impedance $Z_x = V_i / I_s$.

b. Assuming an ideal amplifier, as shown in Fig. P2.11(b), calculate $Z_x$ and $I_s$ for $C = 0.01 \mu\text{F}$ and $A_{vo} = -100$. That is, $R_1 = \infty$ and $R_o = 0$.

**FIGURE P2.11**

2.12 Design a voltage amplifier circuit that will simulate a negative resistance of $R = -5 \text{k} \Omega$ (see Prob. 2.10).

2.13 The current amplifier shown in Fig. 2.7(b) has a short-circuit current gain of $A_{is} = 200$, an input resistance of $R_1 = 150 \Omega$, and an output resistance of $R_o = 2.5 \text{k} \Omega$. The load resistance is $R_L = 100 \Omega$. The input source current is $i_s = 4 \text{ mA}$ with a source resistance of $R_s = 47 \text{k} \Omega$.

a. Calculate the current gain $A_i = i_o / i_s$, the voltage gain $A_v = v_o / v_s$, and the power gain $A_p = P_L / P_v$.

b. Use PSpice/SPICE to check your results in part (a).

2.14 The current amplifier shown in Fig. 2.7(b) has a short-circuit current gain of $A_{is} = 100$, an input resistance of $R_1 = 50 \Omega$, an output resistance of $R_o = 22 \text{k} \Omega$, and a load resistance of $R_L = 150 \Omega$. The input source current is $i_s = 50 \text{ mA}$ with a source resistance of $R_s = 100 \text{k} \Omega$. Calculate the output current $i_o$.

2.15 The current amplifier shown in Fig. 2.7(b) has a source current of $i_s = 5 \mu\text{A}$, a source resistance of $R_s = 100 \text{k} \Omega$, and an input resistance of $R_1 = 50 \Omega$. The short-circuit output current is $i_o = 100 \text{ mA}$ for $R_L = 0$, and the open-circuit output voltage is $v_o = 12 \text{ V}$ for $R_L = \infty$. The load resistance is $R_L = 2.7 \text{k} \Omega$. Calculate (a) the voltage gain $A_v = v_o / v_s$, (b) the current gain $A_i = i_o / i_s$, and (c) the power gain $A_p = P_L / P_v$. 
2.16 An amplifier is required to amplify the output signal from a transducer that produces a constant current of $i_s = 100 \mu A$ at an internal resistance varying from $R_s = 10 \, k\Omega$ to $100 \, k\Omega$. The desired output current is $i_o = 20 \, mA$ at a load resistance varying from $R_L = 20 \, \Omega$ to $500 \, \Omega$. The variation in output current should be kept within $\pm 3\%$. Determine the design specifications of the amplifier.

2.17 An amplifier is required to give a current gain of $A_i = 50 \pm 1.5\%$. The source resistance is $R_s = 100 \, k\Omega$, and the load resistance is $R_L = 100 \, \Omega$. Determine the design specifications of the amplifier.

2.18 A resistance $R$ is connected to a current amplifier, as shown in Fig. P2.18.

- a. Derive an expression for the input resistance $R_x = v_i / i_i$.
- b. Design an amplifier circuit that will simulate a negative resistance of $R_x = -10 \, k\Omega$.

**FIGURE P2.18**

![Image of a circuit diagram]

2.19 Suppose the resistance $R$ of the current amplifier in Fig. P2.18 is replaced by an impedance $Z$ consisting of $R$, $C$, and $-R$. This arrangement is shown in Fig. P2.19 for $A_{is} = 2$, converting the current source to a voltage source, $V_s$.

- a. Derive an expression for the input impedance $Z_x(s) = V_i(s) / I_i(s)$, where $s$ is Laplace’s operator. Note that $-R$ can be generated by another current amplifier such as the one shown in Fig. P2.18 with $A_{is} = 2$.
- b. Design an amplifier circuit that will simulate an inductance of $L_e = 10 \, \text{mH}$.
- c. Use PSpice/SPICE to calculate the input impedance $Z_x$ for frequencies from $1 \, \text{kHz}$ to $5 \, \text{kHz}$ with a linear increment of $1 \, \text{kHz}$. Use a PSpice/SPICE F-type dependent source (see Sec. 2.8).

**FIGURE P2.19**

![Image of a circuit diagram]
2.20 a. Design an amplifier circuit that will simulate an inductance of \( L = 50 \, \text{mH} \) (see Prob. 2.19).

b. Use PSpice/SPICE to verify your design.

2.21 The gyrator circuit shown in Fig. P2.19 has a capacitance of \( C = 100 \, \text{pF} \). Determine the value of resistance \( R \) that will give an effective inductance of \( L = 15 \, \text{mH} \).

2.22 Two ideal transconductance amplifiers are connected back to back, as shown in Fig. P2.22.

a. Find the relation between the input voltage and the input currents, and find the input impedance \( Z_i = V_i/I_i \).

b. If \( v_s = 1 \sin (2000\pi t) \), \( C = 0.1 \, \mu\text{F} \), and \( G_m1 = G_m2 = 3 \, \text{mA/V} \), use PSpice/SPICE to plot the transient response of the output voltage \( v_o(t) \) for a time interval from 0 to 1.5 ms with an increment of 15 \( \mu\text{s} \).

**FIGURE P2.22**

![Diagram of amplifier circuit](image)

2.23 A transconductance amplifier is needed to record the peak voltage of the circuit in Fig. 2.9. The recorder needs 5 mA for a reading of 1 cm, and it should read 20 cm ± 2% for a peak input voltage of 170 V. The input resistance of the recorder varies from \( R_L = 20 \, \Omega \) to 500 \( \Omega \). The frequency of the input voltage is \( f_s = 60 \, \text{kHz} \).

a. Determine the value of capacitance \( C \).

b. Determine the design specifications of the transconductance amplifier.

2.24 An amplifier is required to give a transconductance gain of \( Z_m = 20 \, \text{mA/V} \pm 2\% \). The source resistance is \( R_s = 1 \, \text{k}\Omega \), and the load resistance is \( R_L = 200 \, \Omega \). Determine the design specifications of the amplifier.

2.25 An amplifier is used to measure a DC voltage signal \( v_s = 0 \) to 10 V with a source resistance of \( R_s = 2 \, \text{k}\Omega \) to 5 \( \text{k}\Omega \). The output of the amplifier is a meter that gives a full-scale deflection at a current of \( i_o = 100 \, \text{mA} \) and whose resistance is \( R_m = 20 \, \Omega \) to 100 \( \Omega \). Determine the design specifications of the amplifier.

2.26 The transimpedance amplifier shown in Fig. 2.10(a) has a transimpedance of \( Z_{im} = 0.5 \, \text{kV/A} \), an input resistance of \( R_i = 1.5 \, \text{k}\Omega \), and an output resistance of \( R_o = 4.7 \, \text{k}\Omega \). The input source current is \( i_s = 50 \, \text{mA} \) with a source resistance of \( R_s = 10 \, \text{k}\Omega \). The load resistance is \( R_L = 4.7 \, \text{k}\Omega \). Calculate the current gain \( A_i = i_o/i_s \) and the voltage gain \( A_v = v_o/v_s \).

2.27 A transimpedance amplifier is used to record the short-circuit current of a transducer of unknown internal resistance; its output is a recorder that requires 10 V for a reading of 2 cm. The recorder should read 20 cm ± 2% for an input current of 100 mA. The input resistance of the recorder varies from \( R_L = 2 \, \text{k}\Omega \) to \( R_L = 10 \, \text{k}\Omega \). Determine the design specifications of the amplifier.

2.28 A transimpedance amplifier is used to measure a DC current signal \( i_s = 0 \) to 500 mA with a source resistance of \( R_s = 100 \, \text{k}\Omega \). The output of the amplifier is a meter that gives a full-scale deflection at a voltage of \( v_o = 5 \, \text{V} \) and whose resistance is \( R_m = 20 \, \text{k}\Omega \). Determine the design specifications of the amplifier.
2.29 The parameters of the voltage amplifier in Fig. 2.6(a) are $v_i = 100$ mV, $R_s = 2$ kΩ, $A_{v0} = 250$, $R_i = 50$ kΩ, $R_o = 1$ kΩ, and $R_L = 10$ kΩ. Calculate the values of the equivalent current, transconductance, and transimpedance amplifiers.

2.30 The parameters of the transconductance amplifier in Fig. 2.8(b) are $v_i = 100$ mV, $R_s = 2$ kΩ, $G_{ms} = 20$ mA/V, $R_i = 100$ kΩ, $R_o = 2$ kΩ, and $R_L = 200$ Ω. Calculate the values of the equivalent voltage, current, and transimpedance amplifiers.

2.31 The slew rate of a unity-gain amplifier is $SR = 0.5$ V/µs, and the rise time is 0.3 µs. What is the maximum value $V_{S(max)}$ of a step input voltage?

2.32 The slew rate of a unity-gain amplifier is $SR = 0.5$ V/µs. The input frequency is $f_s = 100$ kHz. Calculate the maximum voltage $V_{s(max)}$ of a sinusoidal input voltage.

2.33 The slew rate of a unity-gain amplifier is $SR = 0.5$ V/µs. The input is a sinusoidal peak voltage $V_m = 10$ V. Determine the maximum input frequency $f_{s(max)}$ that will avoid distortion.

2.4 Cascaded Amplifiers

2.34 The parameters of the cascaded voltage amplifiers in Fig. 2.11(a) are $R_s = 200$ kΩ, $R_{o1} = R_{o2} = R_{o3} = 100$ Ω, $R_{i1} = R_{i2} = R_{i3} = R_L = 2.5$ kΩ, and $A_{v01} = A_{v02} = A_{v03} = 50$.

a. Calculate the overall open-circuit voltage gain $A_{v0} = v_o/v_i$, the effective voltage gain $A_v = v_o/v_s$, the overall current gain $A_i = i_o/i_{i1}$, and the power gain $A_p = P_L/P_{i1}$.

b. Use PSpice/SPICE to check your results in part (a).

2.35 The parameters of the cascaded voltage amplifiers in Fig. 2.11(a) are $R_s = 200$ kΩ, $R_{o1} = R_{o2} = R_{o3} = 100$ Ω, $R_{o3} = 300$ Ω, $R_{i1} = R_{i2} = R_{i3} = R_L = 1.5$ kΩ, and $A_{v01} = A_{v02} = A_{v03} = 80$.

a. Calculate the overall voltage gain $A_{v0} = v_o/v_i$, the overall current gain $A_i = i_o/i_{i1}$, and the power gain $A_p = P_L/P_{i1}$.

b. Use PSpice/SPICE to check your results in part (a).

2.36 The parameters of the cascaded current amplifiers in Fig. 2.12(a) are $R_s = 20$ kΩ, $R_{o1} = R_{o2} = R_{o3} = 4.7$ kΩ, $R_{i1} = R_{i2} = R_{i3} = R_L = 100$ Ω, and $A_{is1} = A_{is2} = A_{is3} = 100$.

a. Calculate the effective current gain $A_i = i_o/i_{i}$, the overall voltage gain $A_v = v_o/v_s$, and the power gain $A_p = P_L/P_{i}$.

b. Use PSpice/SPICE to check your results in part (a).

2.37 One transconductance amplifier is cascaded with a transimpedance amplifier, as shown in Fig. P2.37. The parameters are $R_s = 5$ kΩ, $R_{i1} = 50$ kΩ, $R_{o1} = 200$ Ω, $Z_{mo} = 10$ kΩ, $R_{i2} = 1$ MΩ, $R_{o2} = 100$ kΩ, $R_L = 1$ kΩ, and $G_{ms} = 20$ mA/V.

a. Calculate the overall open-circuit voltage gain $A_{v0} = v_o/v_i$, the effective voltage gain $A_v = v_o/v_s$, the overall current gain $A_i = i_o/i_{i}$, and the power gain $A_p = P_L/P_{i}$.

b. Use PSpice/SPICE to check your results in part (a).

**FIGURE P2.37**
2.5 Frequency Response of Amplifiers

2.38 The voltage amplifier shown in Fig. 2.17(a) is required to have a midrange voltage gain of \( A_{\text{v(mid)}} = -50 \) in the frequency range of 10 kHz to 50 kHz. The source resistance is \( R_s = 1 \, \text{k}\Omega \), and the load resistance is \( R_L = 5 \, \text{k}\Omega \).

a. Determine the specifications of the amplifier and the values for coupling capacitor \( C_1 \) and shunt capacitor \( C_2 \).

b. Use PSpice/SPICE to verify your design by plotting the frequency response \( |A_v(j\omega)| \) against frequency.

2.39 The voltage gain of an amplifier is given by

\[
A_v(j\omega) = \frac{100(10 + j\omega)}{(100 + j\omega)(10^4 + j\omega)}
\]

Calculate (a) the cutoff frequencies \( f_L \) and \( f_H \), (b) the bandwidth \( \text{BW} = f_H - f_L \), and (c) the passband gain in decibels.

2.40 The voltage gain of an amplifier is given by

\[
A_v(j\omega) = \frac{200}{1 + j\omega/100}
\]

Calculate (a) the bandwidth \( \text{BW} \) frequency if \( |A_v(j\omega)| = 100 \) and (b) the bandwidth \( \text{BW} \) frequency if \( |A_v(j\omega)| = 50 \).

2.41 A low-pass transconductance amplifier is shown in Fig. P2.41. The circuit parameters are \( C = 0.1 \, \mu\text{F} \), \( R_s = 5 \, \text{k}\Omega \), \( G_{ms} = 20 \, \text{mA/V} \), \( R_i = 500 \, \text{k}\Omega \), and \( R_o = 50 \, \text{k}\Omega \). Calculate the unity-gain bandwidth \( f_{bw} = \frac{A_{v(mid)}f_H}{f_L} \) for (a) \( R_L = 1 \, \text{k}\Omega \) and (b) \( R_L = 10 \, \text{k}\Omega \).

![Figure P2.41](image)

2.6 Miller’s Theorem

2.42 A capacitor of \( C = 0.01 \, \mu\text{F} \) is connected across the input and output sides of an amplifier, as shown in Fig. P2.42. The amplifier parameters are \( A_{v_0} = -502, R_o = 50 \, \Omega \), and \( R_i = 100 \, \text{k}\Omega \). The source resistance is \( R_s = 2 \, \text{k}\Omega \), and the load resistance is \( R_L = 10 \, \text{k}\Omega \).

a. Use Miller’s theorem to find the break frequencies.

b. Express the frequency-dependent gain \( A_v(j\omega) = V_o(j\omega)/V_i(j\omega) \).
2.43 A capacitor of $C = 10 \text{nF}$ is connected across the input and output sides of an amplifier, as shown in Fig. P2.42. The amplifier parameters are $A_{vo} = -1000$, $R_o = 100 \Omega$, and $R_i = 200 \text{k} \Omega$. The source resistance is $R_s = 5 \text{k} \Omega$, and the load resistance is $R_L = 5 \text{k} \Omega$.
   a. Use Miller’s theorem to find the break frequencies.
   b. Express the frequency-dependent gain $A_v(j\omega) = V_o(j\omega)/V_s(j\omega)$.

2.44 A capacitor of $C = 0.1 \text{ \mu F}$ is connected across the input and output sides of an amplifier, as shown in Fig. P2.44(a). Determine the equivalent Miller capacitance $C_x$ seen by the source, as shown in Fig. P2.44(b), for (a) $A_{vo} = -200$ and (b) $A_{vo} = -1$.

2.45 A resistance $R_F$ is connected across the input and output sides of an amplifier, as shown in Fig. P2.45. The circuit parameters are $R_s = 1 \text{k} \Omega$, $A_{vo} = -2 \times 10^3$, $R_i = 2 \text{ M} \Omega$, $R_o = 75 \text{ \Omega}$, $R_F = 20 \text{ k} \Omega$, and $R_L = 5 \text{k} \Omega$.
   a. Use Miller’s theorem to find the effective voltage gain $A_v = v_o/v_v$.
   b. Use PSpice/SPICE to check your results in part (a).
2.7 Frequency Response Methods

2.46 The parameters of the circuit in Fig. P2.46 are $R_s = 500 \, \Omega$, $C_1 = 20 \, \mu F$, $R_i = 1 \, k \Omega$, $R_L = 10 \, k \Omega$, $C_2 = 10 \, pF$, and $g_m = 15 \, mA/V$. Use $s$-domain analysis to find the low 3-dB frequency $f_L$, the high 3-dB frequency $f_H$, and the midband gain $A_{v(mid)} = A_{PB}$.

FIGURE P2.46

2.47 The parameters of the circuit in Fig. P2.47 are $R_s = 1 \, k \Omega$, $C_1 = 10 \, \mu F$, $C_i = 20 \, pF$, $R_i = 25 \, k \Omega$, $R_L = 10 \, k \Omega$, $R_o = 10 \, k \Omega$, $C_o = 10 \, pF$, and $g_m = 15 \, mA/V$. Use $s$-domain analysis to find the low 3-dB frequency $f_L$, the high 3-dB frequency $f_H$, and the midband gain $A_{v(mid)} = A_{PB}$.

FIGURE P2.47

2.48 The parameters of the circuit in Fig. P2.48 are $R_s = 4 \, k \Omega$, $R_G = 20 \, k \Omega$, $R_L = 10 \, k \Omega$, $C_{gs} = 10 \, pF$, $C_{gd} = 20 \, pF$, and $g_m = 10 \, mA/V$. Use $s$-domain analysis to find the high 3-dB frequency $f_H$ and the low-pass gain $A_{v(low)} = A_{PB}$.

FIGURE P2.48

2.49 An amplifier circuit is shown in Fig. P2.49. Use the zero-value method to find the high 3-dB frequency $f_H$ and the low-pass gain $A_{v(low)} = A_{PB}$.

FIGURE P2.49
2.50 Repeat Prob. 2.49 for $R_{sr} = 0$.  

2.51 An amplifier circuit is shown in Fig. P2.51. Use the short-circuit and zero-value methods to find the low 3-dB frequency $f_L$, the high 3-dB frequency $f_H$, and the midband gain $A_{v(mid)} = A_{PB}$.  

![FIGURE P2.51](image)  

2.52 Repeat Prob. 2.51 for $R_E = 0$.  

2.53 An amplifier circuit is shown in Fig. P2.53. Use the short-circuit and zero-value methods to find the low 3-dB frequency $f_L$, the high 3-dB frequency $f_H$, and the midband gain $A_{v(mid)} = A_{PB}$.  

![FIGURE P2.53](image)  

2.54 An amplifier circuit is shown in Fig. P2.54. Use the short-circuit and zero-value methods to find the low 3-dB frequency $f_L$, the high 3-dB frequency $f_H$, and the midband gain $A_{v(mid)} = A_{PB}$.  

![FIGURE P2.54](image)
CHAPTER 3

INTRODUCTION TO OPERATIONAL AMPLIFIERS AND APPLICATIONS

Learning Outcomes

After completing this chapter, students should be able to do the following:

• Describe the external characteristics and circuit models of op-amps.
• Analyze op-amp circuits to derive their input and output relationships.
• Calculate the effect of a finite op-amp gain on the overall voltage gain of op-amp circuits.
• Determine the bandwidth of op-amp circuits.
• Design op-amp circuits to meet certain input and output specifications.
• List a few examples of op-amp applications in signal conditioning.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_o, A_f$</td>
<td>Op-amp open-loop voltage gain and overall (closed-loop) voltage gain of an op-amp circuit</td>
</tr>
<tr>
<td>$BW, A_{PB}$</td>
<td>Bandwidth and pass-band voltage gain</td>
</tr>
<tr>
<td>CMRR</td>
<td>Common-mode rejection ratio</td>
</tr>
<tr>
<td>$f_H, f_L$</td>
<td>High and low cutoff or break frequencies</td>
</tr>
</tbody>
</table>
### 3.1 Introduction

The operational amplifier (or op-amp) is a high-gain, direct-coupled amplifier consisting of multiple stages: an input stage to provide a high input resistance with a certain amount of voltage gain, a middle stage to provide a high voltage gain, and an output stage to provide a low output resistance. It operates with a differential voltage between two input terminals, and it is a complete, integrated-circuit, prepackaged amplifier. An op-amp, often referred to as a linear (or analog) integrated circuit (IC), is a popular and versatile integrated circuit. It serves as a building block for many electronic circuits. For most applications, knowledge of the terminal characteristics of op-amps is all you need to design op-amp circuits. However, for some applications requiring precision, internal knowledge of op-amps is necessary.

### 3.2 Characteristics of Ideal Op-Amps

The symbol for an op-amp is shown in Fig. 3.1. An op-amp has at least five terminals. Terminal 2 is called the “inverting input” because the output that results from the input at this terminal will be inverted. Terminal 3 is called the “noninverting input” because the output that results from the input at this terminal will have the same polarity as the input. Terminal 4 is for negative DC supply \(-V_{EE}\). Terminal 6 is the output terminal. Terminal 7 is for positive DC supply \(+V_{CC}\).

Instead of using two DC power supplies, we can generate \(V_{CC}\) and \(V_{EE}\) from a single power supply \(V_{DC}\), as shown in Fig. 3.2(a). The value of \(R\) should be high enough (usually \(R \geq 10 \, \text{k}\Omega\)) that it does not draw much current from the DC power supply \(V_{DC}\). Capacitors are used for decoupling (bypass) of the DC power supply, and the value of \(C\) is typically in the range of 0.01 \(\mu\text{F}\) to 10 \(\mu\text{F}\). Instead of two resistors, a potentiometer can be used to ensure that \(V_{CC} = V_{EE}\), as shown in Fig. 3.2(b). Diodes \(D_1\) and \(D_2\) (see Chapter 4)
prevent any reverse current flow; they are often used to protect the op-amp in case the positive and negative terminals of the supply voltage $V_{DC}$ are reversed accidentally. Also, two zener diodes (see Chapter 4) can be used to obtain symmetrical supply voltages, as shown in Fig. 3.2(c). The value of $R$ should be low enough to force the zener diodes to operate in the zener or avalanche mode (see Sec. 4.7). Note that these circuits will not work if the DC supply comes with a ground.

### 3.2.1 Op-Amp Circuit Model

The output voltage of an op-amp is directly proportional to the small-signal differential (or difference) input voltage. Thus, an op-amp can be modeled as a voltage-dependent voltage source; its equivalent circuit is shown in Fig. 3.3(a). The output voltage $v_O$ is given by

$$v_O = A_0v_d = A_0(v_p - v_n)$$

where $A_0 =$ small-signal open-loop voltage gain

$v_d =$ small-signal differential (or difference) input voltage

$v_n =$ small-signal voltage at the inverting terminal with respect to the ground

$v_p =$ small-signal voltage at the noninverting terminal with respect to the ground

![Equivalent circuit](image-url)
Input resistance $R_i$ is the equivalent resistance between the differential input terminals. The input resistance of an op-amp with a bipolar junction transistor (BJT) input stage is very high, with a typical value of 2 MΩ. Op-amps with a field-effect transistor (FET) input stage have much higher input resistances (i.e., $10^{12}$ Ω). Therefore, the input current drawn by the amplifier is very small (typically on the order of nanoamperes), tending to zero.

Output resistance $R_o$ is Thevenin’s equivalent resistance. It is usually in the range of 10 Ω to 100 Ω, with a typical value of 75 Ω. Its effective value is reduced, however, when external connections are made; then $R_o$ can be neglected for most applications.

Open-loop differential voltage gain $A_o$ is the differential voltage gain of the amplifier with no external components. It ranges from $10^4$ to $10^6$, with a typical value of $2 \times 10^5$. Since the value of $A_o$ is very large, $v_d$ becomes very small (typically on the order of microvolts), tending to zero. The transfer characteristic ($v_O$ versus $v_d$) is shown in Fig. 3.3(b). In reality, the output voltage cannot exceed the positive or negative saturation voltage $\pm V_{sat}$ of the op-amp, which is set by supply voltages $V_{CC}$ and $V_{EE}$, respectively. The saturation voltage is usually 1 V lower than the supply voltage $V_{CC}$ or $V_{EE}$. Thus, the output voltage will be directly proportional to the differential input voltage $v_d$ only until it reaches the saturation voltage; thereafter the output voltage remains constant. The gain of practical op-amps is also frequency dependent. Note that the model in Fig. 3.3(a) does not take into account the saturation effect and assumes that gain $A_o$ remains constant for all frequencies.

The analysis and design of circuits employing op-amps can be greatly simplified if the op-amps in the circuit are assumed to be ideal. Such an assumption allows you to approximate the behavior of the op-amp circuit and to obtain the approximate values of circuit components that will satisfy some design specifications. Although the characteristics of practical op-amps differ from the ideal characteristics, the errors introduced by deviations from the ideal conditions are acceptable in most applications. A complex op-amp model is used in applications requiring precise results. The circuit model of an ideal op-amp is shown in Fig. 3.4; its characteristics are as follows:

- The open-loop voltage gain is infinite: $A_o = \infty$.
- The input resistance is infinite: $R_i = \infty$.
- The amplifier draws no current: $i_i = 0$.
- The output resistance is negligible: $R_o = 0$.
- The gain $A_o$ remains constant and is not a function of frequency.
- The output voltage does not change with changes in power supplies. This condition is generally specified in terms of the power supply sensitivity (PSS): PSS = 0.
- An op-amp is a differential amplifier, and it should amplify the differential signal appearing between the two input terminals. Any signal that is common to two inputs (i.e., noise) should not be amplified and should not appear in the output. Thus, the differential gain (due to a differential signal) should tend to infinity, and the common-mode gain (due to a common signal) should tend to zero. The condition is generally specified in terms of the common-mode rejection ratio (CMRR): $CMRR = \infty$. This ratio is discussed in Sec. 3.2.3.
EXAMPLE 3.1

Finding the differential input voltage and input current of an op-amp  The op-amp of Fig. 3.3(a) has an open-loop gain of \( A_o = 2 \times 10^5 \). The input resistance is \( R_i = 0.6 \text{ M}\Omega \). The DC supply voltages are \( V_{CC} = 12 \text{ V} \) and \( -V_{EE} = -12 \text{ V} \). Assume that \( V_{sat} = 11 \text{ V} \).

(a) What value of \( v_d \) will saturate the amplifier?

(b) What are the values of the corresponding input current \( i_i \)?

SOLUTION

(a) \( v_d = \pm V_{sat}/A_o = \pm 11/(2 \times 10^5) = \pm 55 \text{ } \mu\text{V} \)

(b) \( i_i = -v_d/R_i = \mp 55 \text{ } \mu\text{V}/0.6 \text{ M}\Omega = \mp 0.1 \text{ nA} \)

EXAMPLE 3.2

Finding the maximum output voltage of an op-amp  The op-amp of Fig. 3.3(a) has \( A_o = 2 \times 10^5 \), \( R_i = 2 \text{ M}\Omega \), \( R_o = 75 \text{ } \Omega \), \( V_{CC} = 12 \text{ V} \), and \( -V_{EE} = -12 \text{ V} \). The maximum possible output voltage swing is \( \pm 11 \text{ V} \). If \( v_- = 100 \text{ } \mu\text{V} \) and \( v_+ = 25 \text{ } \mu\text{V} \), determine the output voltage \( v_O \).

SOLUTION

From Eq. (3.1),
\[
 v_O = A_o(v_+ - v_-) = 2 \times 10^5 \times (25 - 100) \times 10^{-6} = -15 \text{ V}
\]
Because of the saturation, the output voltage cannot exceed the maximum voltage limit of \( -11 \text{ V} \), and therefore \( v_O = -11 \text{ V} \).

3.2.2 Op-Amp Frequency Response

The differential voltage gain of an op-amp has the highest value at DC or low frequencies. The gain decreases with frequency. A typical frequency response is shown in Fig. 3.5. The gain falls uniformly with a slope of \(-20 \text{ dB/decade} \). This uniform slope is maintained by internal design in internally compensated op-amps. The voltage gain of an internally compensated op-amp at frequency \( f \) can usually be expressed as
\[
 A_o(j\omega) = \frac{A_o}{1 + j\omega/\omega_b} = A_o \frac{1}{1 + jf/f_b} \quad (3.2)
\]
where \( A_o \) is DC gain, typically \( 2 \times 10^5 \) and \( f_b \) is break (or 3-dB) frequency in hertz.
For \( f \gg f_b \) and \((f/f_b) \gg 1\), Eq. (3.2) is reduced to

\[
A_o(f\omega) = \frac{A_o}{if/f_b} = \frac{A_o f_b}{if}
\]

(3.3)

The magnitude of this gain becomes unity (or 0) at frequency \( f = f_{bw} \). That is,

\[
f_{bw} = A_o/f_b
\]

(3.4)

where \( f_{bw} \) is called the *unity-gain bandwidth*. The typical value of \( f_{bw} \) for the LF411 op-amp is 4 MHz. The 3-dB frequency can be related to time constant \( \tau \) or to rise time \( t_r \) by

\[
f = \frac{1}{2\pi\tau} = \frac{2.2}{2\pi t_r} = \frac{0.35}{t_r}
\]

(3.5)

Thus, the frequency response is inversely proportional to the rise time \( t_r \). The input signal frequency \( f_s \) should be less than the maximum op-amp frequency; otherwise the output voltage will be distorted. For example, if the rise time of an input signal is \( t_r = 0.1 \mu s \), its corresponding input frequency is \( f_s = 0.35/0.1 \mu s = 3.5 \) MHz, and the output voltage will be distorted in an op-amp of \( f_{bw} = 1 \) MHz.

### 3.2.3 Common-Mode Rejection Ratio

Because an op-amp is a differential amplifier, it should amplify the differential voltage between the input terminals. Any signal (i.e., noise) that appears simultaneously at both inputs should not be amplified. Let \( v_1 \) and \( v_2 \) be the input voltages at the noninverting and inverting terminals, respectively, as shown in Fig. 3.6(a). As shown in Fig. 3.6(b), these voltages can be resolved into two components: differential voltage \( v_d \) and common-mode voltage \( v_c \).

Let us define the differential voltage \( v_d \) as

\[
v_d = v_2 - v_1
\]

(3.6)

and the common-mode voltage \( v_c \) as

\[
v_c = \frac{v_1 + v_2}{2}
\]

(3.7)
Then, the two input voltages can be expressed as

\[ v_2 = v_c + \frac{v_d}{2} \]  
\[ v_1 = v_c - \frac{v_d}{2} \]  

(3.8)

(3.9)

Let \( A_1 \) be the voltage gain with an input at the inverting terminal and the noninverting terminal grounded. Let \( A_2 \) be the voltage gain with an input at the noninverting terminal and the inverting terminal grounded. We can obtain the output voltage of the op-amp by applying the superposition theorem. That is,

\[ v_O = A_1 v_1 + A_2 v_2 \]  

(3.10)

Substituting \( v_1 \) from Eq. (3.9) and \( v_2 \) from Eq. (3.8) into Eq. (3.10) yields

\[ v_O = A_1 \left( v_c - \frac{v_d}{2} \right) + A_2 \left( v_c + \frac{v_d}{2} \right) \]

\[ = \frac{(A_2 - A_1)}{2} v_d + (A_2 + A_1) v_c \]

\[ = A_d v_d + A_c v_c \]  

(3.11)

\[ = A_d \left( v_d + \frac{A_c}{A_d} v_c \right) \]  

(3.12)

where \( A_d = (A_2 - A_1)/2 \) is differential voltage gain and \( A_c = (A_2 + A_1) \) is common-mode voltage gain.

According to Eq. (3.12), the output voltage depends on the common-mode voltage \( v_c \) and the differential voltage \( v_d \). Since \( A_1 \) is negative and \( A_2 \) is positive, \( A_d > A_c \). If \( A_d \) can be made much greater than \( A_c \), \( v_O \approx A_d v_d \) and the output voltage will be almost independent of the common-mode signal \( v_c \). The ability of an op-amp to reject the common-mode signal is defined by a performance criterion called the common-mode rejection ratio (CMRR), which is defined as the magnitude of the ratio of the voltage gains. That is,

\[ \text{CMRR} = \left| \frac{A_d}{A_c} \right| \]  

(3.13)

\[ = 20 \log \left| \frac{A_d}{A_c} \right| \text{ (dB)} \]  

(3.14)

The value of CMRR should ideally be infinity; a typical value is 100 dB for the LF411 op-amp.
Finding the output voltages and gains of a practical op-amp  The input voltages of an op-amp are $v_2 = 1005 \, \mu V$ and $v_1 = 995 \, \mu V$. The op-amp parameters are CMRR = 100 dB and $A_d = 2 \times 10^5$. Determine (a) the differential voltage $v_d$, (b) the common-mode voltage $v_c$, (c) the magnitude of the common-mode gain $A_c$, and (d) the output voltage $v_o$.

SOLUTION

From Eq. (3.14),

$$20 \log (\text{CMRR}) = 100 \, \text{dB}$$

or $\log (\text{CMRR}) = \frac{100}{20} = 5$

which gives $\text{CMRR} = \left| \frac{A_d}{A_c} \right| = 10^5$.

(a) The differential voltage $v_d$ is

$$v_d = v_2 - v_1 = 1005 \, \mu V - 995 \, \mu V = 10 \, \mu V$$

(b) From Eq. (3.7), the common-mode voltage is

$$v_c = \frac{(v_1 + v_2)}{2} = \frac{(1005 \, \mu V + 995 \, \mu V)}{2} = 1000 \, \mu V$$

(c) From Eq. (3.13),

$$\left| \frac{A_d}{A_c} \right| = 10^5$$

or $\left| A_c \right| = \frac{\left| A_d \right|}{10^5} = \frac{2 \times 10^5}{10^5} = 2$

(d) From Eq. (3.11), the output voltage $v_o$ becomes

$$v_o = A_d v_d + A_c v_c$$

$$= 2 \times 10^5 \times 10 \, \mu V \pm 2 \times 1000 \, \mu V = 2 \pm 0.002 = 2.002 \, V \text{ or } 1.998 \, V$$

NOTE: In the absence of the common-mode signal, $v_c = 0$ and

$$v_o = A_d v_d = 2 \times 10^5 \times 10 \times 10^6 = 2 \, V$$

$v_c$ is 100 times $v_d$, but the CMRR introduces only a 0.1% error in the output voltage. Therefore, the effect of the common-mode signal can be neglected. An ideal op-amp will have CMRR = $\infty$ so that $v_o = A_d v_d$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
3.3 Op-Amp PSpice/SPICE Models

There are many types of op-amps, as we will see in Chapter 14. An op-amp can be simulated from its internal circuit arrangement. The internal structure of op-amps is very complex, however, and differs from one model to another. For example, the \( \mu A741 \) type of general-purpose op-amp consists of 24 transistors. It is too complex for the student version of the PSpice circuit simulation software to analyze; however, a macromodel, which is a simplified version of the op-amp and requires only two transistors, is quite accurate for many applications, and can be simulated as a subcircuit or a library file [1, 2]. Some manufacturers of op-amps supply macromodels of their products [3]. The student version of PSpice has a library called NOM.LIB, which contains models of three common types of op-amps: \( \mu A741 \), LM324, and LF411. The parameters of the three op-amps for the circuit model in Fig. 3.3(a) are as follows:

- The \( \mu A741 \) op-amp is a general-purpose op-amp with a BJT input stage. It is capable of producing output voltages of \( \pm 14 \text{ V} \) with DC power supply voltages of \( \pm 15 \text{ V} \). The parameters are \( R_i = 2 \text{ M}\Omega, R_o = 75 \text{ \Omega}, A_o = 2 \times 10^5 \), break frequency \( f_b = 10 \text{ Hz} \), and unity-gain bandwidth \( f_{bw} = 1 \text{ MHz} \).
- The LF411 op-amp is a general-purpose op-amp with an FET input stage. It is capable of producing output voltages of \( \pm 13.5 \text{ V} \) with DC power supply voltages of \( \pm 15 \text{ V} \). The parameters are \( R_i = 10^{12} \text{ \Omega}, R_o = 50 \text{ \Omega}, A_o = 2 \times 10^5 \), break frequency \( f_b = 20 \text{ Hz} \), and unity-gain bandwidth \( f_{bw} = 4 \text{ MHz} \).
- The LM324 op-amp has a BJT input stage and is used with a single DC power supply voltage. It can produce output voltages in the range from approximately 20 mV to 13.5 V with a DC supply voltage of \( \pm 15 \text{ V} \). The parameters are \( R_i = 2 \text{ M}\Omega, R_o = 50 \text{ \Omega}, A_o = 2 \times 10^5 \), break frequency \( f_b = 4 \text{ kHz} \), and unity-gain bandwidth \( f_{bw} = 1 \text{ MHz} \).

The professional version of PSpice supports library files for many devices. It is advisable to check the name of the current library file by listing the files of the PSpice programs.

If the PSpice/SPICE model of an op-amp is not available, it is possible to represent the op-amp by simple models that give reasonable results, especially for determining the approximate design values of op-amp circuits. PSpice/SPICE models can be classified into three types: DC linear models, AC linear models, and nonlinear macromodels. Taking the \( \mu A741 \) op-amp as an example, we will develop simple PSpice/SPICE models of these three types.

3.3.1 DC Linear Model

An op-amp may be modeled as a voltage-controlled voltage source, as shown in Fig. 3.7. Two zener diodes (see Sec. 4.7) are connected back to back in order to limit the output swing to the saturation voltages (say, between \(-14 \text{ V} \) and \(+14 \text{ V} \)). This simple model, which assumes that the voltage gain is independent of...
the frequency, is suitable only for DC or low-frequency applications. The list of the PSpice/SPICE subcircuit UA741_DC for Fig. 3.7 is shown here:

```
* Subcircuit definition for UA741_DC
.SUBCKT UA741_DC  1  2  3  4
* Subcircuit name Vi+ Vi- Vo+ Vo-
RI  1  2  2MEG ; Input resistance
RO  5  3  75 ; Output resistance
EA  5  4  1  2  2E+5 ; Voltage-controlled voltage source
D1  3  6  DMOD ; Zener diode with model DMOD
D2  4  6  DMOD ; Zener diode with model DMOD
.MODEL DMOD D (BV=14V) ; Ideal zener model with a zener voltage of 14 V
.ENDS UA741_DC ; End of subcircuit definition
```

In PSpice/SPICE, the name of a subcircuit must begin with X. For example, the calling statement for the amplifier A1, which uses the subcircuit UA741_DC, is as follows:

```
XA1 7 8 9 10 UA741_DC
* Vi+ Vi- Vo+ Vo- Subcircuit name
```

This subcircuit definition UA741_DC can be inserted into the circuit file. Alternatively, it can reside in a user-defined file, say USER.LIB in C drive, in which case the circuit file must contain the following statement:

```
C:USER.LIB ; Library file name must include the drive and directory location
```

### 3.3.2 AC Linear Model

The frequency response of internally frequency-compensated op-amps can be approximated by a single break frequency, as shown in Fig. 3.8(a). This characteristic can be modeled by the circuit of Fig. 3.8(b), which is a frequency-dependent model of an op-amp. The dependent sources have a common node, 4. Without this common node, PSpice/SPICE will give an error message because there will be no DC path from the nodes of the dependent current source to the ground. The common node could be either with the input stage or with the output stage. The time constant \( \tau = R_1C_1 \) gives the break frequency \( f_b \). If an op-amp has more than one break frequency, it can be represented by using as many capacitors as there are breaks. \( R_i \) and \( R_o \) are the input and output resistances, respectively. \( A_o \) is the open-circuit DC voltage gain. Two zener diodes are connected back to back in order to limit the output swing to the saturation voltages (say, between -14 V and +14 V).

The no-load output voltage can be expressed in Laplace’s domain as

\[
V_o(s) = A_o V_2(s) = \frac{A_o R_I I_1}{1 + R_I C_1 s} = \frac{A_o V_d}{1 + R_I C_1 s}
\]  

(3.15)
Substituting \( s = j\omega = j2\pi f \) into Eq. (3.15) gives

\[
A_o(j\omega) = \frac{A_o V_d}{1 + j2\pi f R_1 C_1} = \frac{A_o V_d}{1 + jf/f_b}
\]  

which gives the frequency-dependent open-loop voltage gain of an op-amp with a single break frequency as

\[
A_o(j\omega) = \frac{V_o}{V_d} = \frac{A_o}{1 + jf/f_b}
\]  

where \( f_b = 1/(2\pi R_1 C_1) \) is break frequency in hertz and \( A_o \) is large-signal (or DC) voltage gain of the op-amp.

For \( \mu A741 \) op-amps, \( f_b = 10 \text{ Hz}, A_o = 2 \times 10^5, R_i = 2 \text{ M}\Omega, \) and \( R_o = 75 \Omega. \) If we let \( R_1 = 10 \text{ k}\Omega \) (used as a typical value), \( C_1 = 1/(2\pi \times 10 \times 10 \times 10^3) = 1.5619 \mu F. \)

Note that we could also choose a different value of \( R_1. \) The list of the PSpice/SPICE subcircuit UA741_AC for Fig. 3.8(b) is shown here:

* Subcircuit definition for UA741_AC
  .SUBCKT UA741_AC 1 2 3 4
  .SUBCircuit name Vi+ Vi- Vo+ Vo-
  RI 1 2 2MEG ; Input resistance
  R0 6 3 75 ; Output resistance
  GB 4 5 1 2 0.1M ; Voltage-controlled current source
  R1 5 4 10K
  C1 5 4 1.5619UF
  EA 6 4 5 4 2E+5 ; Voltage-controlled voltage source
  D1 3 7 DMOD ; Zener diode with model DMOD
  D2 4 7 DMOD ; Zener diode with model DMOD
  .MODEL DMOD D (BV=14V) ; Ideal zener model with a zener voltage of 14 V
  .ENDS UA741_AC ; End of subcircuit definition

### 3.3.3 Nonlinear Macromodel

The subcircuit definitions of op-amp macromodels are described by a set of .MODEL statements. The macromodels are normally simulated at room temperature and contain nominal values. The effects
of temperature are not included. The library file NOM.LIB contains the subcircuit definition UA741, which can be called up by including the following general statements in the circuit file:

* Subcircuit call for UA741 (or LF411 or LM324) op-amp
* Connections: noninverting input
* | Inverting input
* | | Positive power supply
* | | | Negative power supply
* | | | | Output
* | | | | | Subcircuit name
XA1 1 2 4 5 6 UA741 (or LF411 or LM324) ; Subcircuit calling must begin with X
* Vi+ Vi- Vp+Vp- Vout ; Calling UA741 for amplifier A1
.LIB NOM.LIB ; Calling library file NOM.LIB

**NOTE** With PSpice/OrCAD schematics, there is no need for developing subcircuit definitions of an op-amp macromodel. PSpice or OrCAD automatically creates the macromodel from the op-amp schematic.

---

**KEY POINTS OF SECTION 3.3**

- An op-amp can be represented in PSpice/SPICE by one of three models: (a) a DC linear model, which is simple but suitable only for low frequencies (typically less than 20 Hz); (b) an AC linear model, which is simple and frequency dependent; (c) a nonlinear macromodel, which is more complex.
- The student version of PSpice limits the number of active devices and nodes, allowing only one macromodel in a circuit. Thus, the choice of a model depends on the complexity of the circuit; the preferred model is the macromodel, followed by the AC model and then the DC model.

---

### 3.4 Analysis of Ideal Op-Amp Circuits

In Eq. (3.1) there are three possible conditions for the output voltage $v_O$: (a) if $v_n = 0$, $v_O$ will be positive ($v_O = A_o v_p$); (b) if $v_p = 0$, $v_O$ will be negative ($v_O = -A_o v_n$); or (c) if both $v_p$ and $v_n$ are present, $v_O = A_o (v_p - v_n)$. Therefore, depending on the conditions of the input voltages, op-amp circuits can be classified into three basic configurations: noninverting amplifiers, inverting amplifiers, or differential (or difference) amplifiers (see Sec. 3.5.3).

#### 3.4.1 Noninverting Amplifiers

The configuration of a noninverting amplifier is shown in Fig. 3.9(a). The input voltage $v_S$ is connected to the noninverting terminal. The voltage $v_x$, which is proportional to the output voltage, is connected via $R_1$ and $R_F$ to the inverting terminal. Using Kirchhoff’s voltage law (KVL), we get

$$v_S = v_x + v_d$$

The differential voltage $v_d$, given by

$$v_d = v_S - v_x$$
is then amplified by the op-amp, whose output is then fed back to the inverting terminal. Thus, this is a feedback circuit; the block diagram is shown in Fig. 3.9(b). We will cover feedback in Chapter 10.

Let us assume an ideal op-amp. That is, $v_d = 0$, $i_s = 0$, and $A_o = \infty$. The voltage $v_x$ at the inverting terminal is

$$v_x = v_s - v_d \approx v_s$$

Using Kirchhoff’s current law (KCL) at the inverting terminal, we get

$$i_1 + i_f + i_i = 0$$

Since the current $i_i$ drawn by an ideal op-amp is zero, $i_i = -i_f$. That is,

$$\frac{v_x}{R_1} = -\frac{v_x - v_o}{R_f} \quad \text{or} \quad \frac{v_s}{R_1} = -\frac{v_s - v_o}{R_f}$$

which, after simplification, yields

$$v_o = \left( 1 + \frac{R_f}{R_1} \right) v_s$$

giving the closed-loop voltage gain $A_f$ as

$$A_f = \frac{v_o}{v_s} = 1 + \frac{R_f}{R_1} \quad (3.18)$$

Since the current drawn by the amplifier is zero, the effective input resistance of the amplifier is very high, tending to infinity:

$$R_{in} = \frac{v_s}{i_s} = \infty$$

**NOTE** $v_d$ and $i_s$ tend to be close to zero due to the large op-amp gain $A_o$, not zero. Otherwise, the circuit will not work as expected.
The effective output resistance is given by \( R_{out} = R_o \approx 0 \). If \( R_F = 0 \) or \( R_1 = \infty \), as shown in Fig. 3.10, Eq. (3.18) becomes

\[
A_f = 1
\]  

That is, the output voltage equals the input voltage: \( v_O = v_S \). The circuit of Fig. 3.10 is commonly referred to as a voltage follower because its output voltage follows the input voltage. It has the inherent characteristics of a high input impedance (or resistance, typically \( 10^{10} \Omega \)) and a low output impedance (or resistance, typically 50 m\( \Omega \)). The exact values can be found by applying the feedback analysis techniques discussed in Chapter 10. A voltage follower is commonly used as the buffer stage between a low impedance load and a source requiring a high impedance load.

**CMRR of a Noninverting Amplifier**

If CMRR_o is the CMRR of the op-amp, we can find out the CMRR of the noninverting amplifier from the following derivations. Let \( v_{id} \) and \( v_{icm} \) denote the difference and common-mode signals of the amplifier, respectively. Considering two input signals \( v_1 \) and \( v_2 \) to the noninverting amplifier as shown in Fig. 3.9(a), we have \( v_{id} = (v_2 - v_1) = v_2 - v_1 \) and \( v_{icm} = (v_2 + v_1)/2 = v_2/2 = v_S/2 \) since \( v_1 = 0 \).

The voltage \( v_p \) at the noninverting terminal is \( v_p = v_2 \), and the voltage \( v_n \) at the inverting terminal can be expressed in terms of \( v_O \) by

\[
v_n = \frac{R_1}{R_1 + R_F} v_O
\]

Similarly, let \( v_{do} \) and \( v_{cmo} \) denote the difference and common-mode signals at the op-amp input. Then, we have \( v_{do} = (v_p - v_n) \) and \( v_{cmo} = (v_p + v_n)/2 \). Thus, the output voltage \( v_o \) can be expressed in terms of the op-amp differential gain \( A_d \) and common-mode gain \( A_{cm} \) as

\[
v_o = A_d v_{do} + A_{cm} v_{cmo} = A_d (v_p - v_n) + \frac{A_{cm} (v_p + v_n)}{2}
\]

After substituting for \( v_p (v_S = 2v_{icm} = v_{id}) \) and \( v_n \) using the relationships in terms of \( v_{id} \) and \( v_{icm} \), we can find the following expression for \( v_O \):

\[
v_o = A_d (v_p - v_n) + \frac{A_{cm}}{2} (2v_{icm} + v_n) = A_d \left( v_{id} - v_o \frac{R_1}{R_1 + R_F} \right) + A_{cm} \left( v_{icm} + v_O \frac{R_1/2}{R_1 + R_F} \right)
\]  

(3.20)
This can be solved for \( v_O \) after we collect all the terms containing \( v_o \) in terms of \( v_{id} \) and \( v_{cm} \):

\[
v_o = \frac{A_d(1 + R_F/R_1)v_{id} + A_{cm}(1 + R_F/R_1)v_{icm}}{1 + R_F/R_1 + A_d - A_{cm}/2}
\]  

(3.21)

Since \( A_d \gg A_{cm} \) and \( A_d \gg R_F/R_1 \gg 1 \), the denominator of Eq. (3.21) is approximately equal to \( A_d \), and \( v_o \) in Eq. (3.21) can be approximated to

\[
v_o \approx \frac{A_d}{A_d} \left(1 + \frac{R_F}{R_1}\right) v_{id} + \frac{A_{cm}}{A_d} \left(1 + \frac{R_F}{R_1}\right) v_{icm} = \left(1 + \frac{R_F}{R_1}\right) v_{id} + \frac{1}{\text{CMRR}_o} \left(1 + \frac{R_F}{R_1}\right) v_{icm}
\]  

(3.22)

where \( \text{CMRR}_o = A_d/A_{cm} \) is the CMRR of the op-amp. From Eq. (3.22), we can find the differential voltage gain \( A_{d-amp} \) and the common-mode gain \( A_{cm-amp} \) of the noninverting amplifier:

\[
A_{d-amp} = 1 + \frac{R_F}{R_1}
\]  

(3.23)

\[
A_{cm-amp} = \frac{1}{\text{CMRR}_o} \left(1 + \frac{R_F}{R_1}\right)
\]  

(3.24)

Therefore, we can find the CMRR of the noninverting amplifier, which is the same as that of the op-amp:

\[
\text{CMRR}_{amp} = \frac{A_{d-amp}}{A_{cm-amp}} = \text{CMRR}_o
\]  

(3.25)

**NOTES**

1. The current \( i_S \) flowing into an op-amp and the differential voltage \( v_d \) are very small, tending to zero. Thus, the inverting terminal is at a ground potential with respect to the noninverting terminal, and it is said to be at the virtual short.
2. \( A_o \) is the open-loop voltage gain of the op-amp, whereas \( A_f \) is the closed-loop voltage gain of the op-amp circuit (or amplifier) and is dependent only on external components.
3. A noninverting amplifier can be designed to give a specified gain \( A_f \) simply by choosing the appropriate ratio \( R_F/R_1 \). A small value of \( R_1 \) will load the amplifier and cause it to draw appreciable current, and a large value of \( R_F \) will increase the noise generated in the resistor. As a guide, all resistances in op-amp circuits should be between 1 k\( \Omega \) and 10 M\( \Omega \).
4. Designing a noninverting voltage amplifier is very simple: Given gain \( A_f \), choose \( R_1 \) and then find \( R_F \).

**EXAMPLE 3.4**

**Designing a noninverting op-amp circuit** Design a noninverting amplifier as shown in Fig. 3.9(a) to provide a closed-loop voltage gain of \( A_f = 80 \). The input voltage is \( v_S = 200 \text{ mV} \) with a source resistance of \( R_s = 500 \Omega \). Find the value of output voltage \( v_O \). The DC supply voltages are \( V_{CC} = V_{EE} = 12 \text{ V} \).
SOLUTION

Choose a suitable value of $R_1$: Let $R_1 = 5 \, \text{k}\Omega$. Find the value of $R_F$ from Eq. (3.18). Since $A_f = 80 = 1 + R_F/R_1$,

\[
\frac{R_F}{R_1} = 79
\]

and $R_F = 79 \times 5 = 395 \, \text{k}\Omega$

Find the output voltage $v_O$ from Eq. (3.18):

\[
v_O = A_f v_S = 80 \times 200 \times 10^{-3} = 16 \, \text{V}
\]

which exceeds the maximum DC supply voltage $V_{CC} = 12 \, \text{V}$. Thus, the output voltage will be $v_O = V_{CC} = 12 \, \text{V}$.

NOTE: $R_s$ is in series with the op-amp input resistance $R_i$, which is very large in comparison to $R_s$. Therefore, $R_s$ will not affect the closed-loop gain $A_f$.

EXAMPLE 3.5

Finding the voltage gain of a noninverting op-amp circuit For the noninverting amplifier in Fig. 3.9(a), the input voltage is $v_S = 100 \, \text{mV}$ with a source resistance of $R_s = 500 \, \Omega$. The circuit parameters are $R_F = 395 \, \text{k}\Omega$, $R_1 = 5 \, \text{k}\Omega$, and $A_o = 2 \times 10^5$. Calculate (a) the closed-loop gain $A_f$, (b) the output voltage $v_O$, and (c) the errors in the output voltage $v_O$ and the gain $A_f$ if $A_o$ tends to infinity.

SOLUTION

Since the current drawn by the op-amp is zero, $i_1 = -i_f$. That is,

\[
\frac{v_x}{R_1} = -\frac{v_x - v_O}{R_F}
\]

which gives

\[
v_x = \frac{R_1}{R_1 + R_F} v_O \tag{3.26}
\]

The output voltage $v_O$ is

\[
v_O = A_o (v_S - v_x) = A_o v_d \tag{3.27}
\]

The input voltage at the noninverting terminal is the sum of $v_x$ and $v_d$. That is,

\[
v_S = v_x + v_d
\]

which, after substitution of $v_x$ from Eq. (3.26) and $v_d$ from Eq. (3.27), becomes

\[
v_S = \frac{R_1 v_O}{R_1 + R_F} + \frac{v_O}{A_o} = v_O \left( \frac{R_1}{R_1 + R_F} + \frac{1}{A_o} \right)
\]
Thus, the closed-loop voltage gain $A_f$ is given by

$$A_f = \frac{v_O}{v_S} = \frac{A_o(R_1 + R_F)}{A_o R_1 + R_1 + R_F} = \frac{1 + R_F/R_1}{1 + (1 + R_F/R_1)A_o} = \frac{1 + R_F/R_1}{1 + x} \tag{3.28}$$

where

$$x = \frac{1}{A_o} \left( 1 + \frac{R_F}{R_1} \right) \tag{3.29}$$

For a small value of $x$, which is usually the case, $(1 + x)^{-1} \approx 1 - x$, and Eq. (3.28) can be approximated by

$$A_f = \left( 1 + \frac{R_F}{R_1} \right) (1 - x) \tag{3.30}$$

Therefore, the error introduced for a finite value of gain $A_o$ is $x$.

(a) From Eq. (3.29),

$$x = \frac{1 + 395/5 \text{k}}{2 \times 10^5 \text{k}} = 40 \times 10^{-5} = 40 \times 10^{-3}\%$$

From Eq. (3.28),

$$A_f = \left( 1 + \frac{395}{1 + 40 \times 10^{-5}} \right) = 79.968$$

(b) The output voltage $v_O$ is

$$v_O = A_f v_S = 79.968 \times 100 \times 10^{-3} = 7.9968 \text{ V}$$

(c) From Eq. (3.30), the error in the output voltage $v_O$ is

$$\Delta v_O = -x \left( 1 + \frac{R_F}{R_1} \right) = -40 \times 10^{-5} \times 80 = -32 \text{ mV, or } -0.04\%$$

The error in the gain $A_f$ is

$$\Delta A_f = -x = -40 \times 10^{-5} = -0.04\%$$

**NOTE:** To minimize the dependence of the closed-loop gain $A_f$ on the open-loop gain $A_o$, the value of $x$ should be made very small. That is,

$$A_o >> \left( 1 + \frac{R_F}{R_1} \right) \tag{3.31}$$

This condition is often satisfied by making $A_o$ at least 10 times larger than $(1 + R_F/R_1)$. That is,

$$\left( 1 + \frac{R_F}{R_1} \right) \leq 0.1A_o \tag{3.32}$$
EXAMPLE 3.6

**Finding the parameters of a practical noninverting op-amp circuit**  The noninverting amplifier in Fig. 3.9(a) has \( R_1 = 10 \, \text{k} \Omega \) and \( R_F = 10 \, \text{k} \Omega \). The op-amp parameters are \( A_o = 2 \times 10^5 \), \( f_b = 10 \, \text{Hz} \), \( R_o = 75 \, \text{\Omega} \), and \( R_i = 2 \, \text{M} \Omega \). The frequency of the input signal is \( f_s = 10 \, \text{kHz} \). Determine (a) the unity-gain bandwidth \( f_{bw} \) of the op-amp, (b) the closed-loop voltage gain \( A_c \), and (c) the closed-loop break frequency \( f_c \) of the op-amp circuit.

**SOLUTION**

Using Eq. (3.28), we find the frequency-dependent voltage gain of the noninverting amplifier to be

\[
A_f(j\omega) = \frac{1 + R_F/R_1}{1 + (1 + R_F/R_1)/A_o(j\omega)}
\]

Substituting the frequency-dependent gain \( A(j\omega) \) from Eq. (3.2), we get

\[
A_f(j\omega) = \frac{1 + R_F/R_1}{1 + (1 + R_F/R_1)/A_o + jf(1 + R_F/R_1)/(A_o f_b)}
\]

Equation (3.33) becomes

\[
A_f(j\omega) = \frac{1 + R_F/R_1}{1 + j(1 + R_F/R_1)/f_{bw}}
\]

which gives the closed-loop break (or 3-dB) frequency as

\[
f_c = \frac{f_{bw}}{1 + R_F/R_1} = \frac{f_{bw} R_1}{R_1 + R_F} = \beta f_{bw} = \beta A_o f_b
\]

where \( \beta = R_1/(R_1 + R_F) \) is called the feedback ratio, or the feedback factor. This should not be confused with the current gain \( \beta_F \) of bipolar transistors in Chapters 1 and 8.

The closed-loop DC gain is \( (1 + R_F/R_1) \) (as expected), and this gain also falls off at a rate of \(-20 \, \text{dB} \) decade after a break frequency of \( f_c = \beta f_{bw} \).

For \( R_1 = 10 \, \text{k} \Omega \), \( R_F = 10 \, \text{k} \Omega \), \( A_o = 2 \times 10^5 \), \( f_b = 10 \, \text{Hz} \), and \( f = f_s = 10 \, \text{kHz} \),

\[
\frac{R_F}{R_1} = \frac{10 \, \text{k} \Omega}{10 \, \text{k} \Omega} = 1
\]

\[
\beta = \frac{R_1}{(R_1 + R_F)} = \frac{10 \, \text{k} \Omega}{(10 \, \text{k} \Omega + 10 \, \text{k} \Omega)} = 0.5
\]

which is small compared to \( A_o = 2 \times 10^5 \).

(a) From Eq. (3.4),

\[
f_{bw} = A_o f_b = 2 \times 10^5 \times 10 = 2 \, \text{MHz}
\]

(b) From Eq. (3.34), we get

\[
A_f(j\omega) = \frac{1 + 1}{1 + jf(1 + 1)/(f_{bw})} = \frac{2}{1 + j(10 \times 10^3) \times 2/(2 \times 10^5)} = \frac{2}{1 + j0.01} = 1.9999 \angle -0.57^\circ
\]

(c) From Eq. (3.35), we get

\[
f_c = \beta A_o f_b = 0.5 \times 2 \times 10^5 \times 10 = 1 \, \text{MHz}
\]
3.4.2 Inverting Amplifiers

Another common configuration is the inverting voltage amplifier, as shown in Fig. 3.11. $R_F$ is used to feed the output voltage back to the inverting terminal of the op-amp. Using Kirchhoff’s voltage law, we have

$$v_S = R_1 i_S - v_d \quad (3.36)$$

$$-v_d = R_F i_f + v_O \quad (3.37)$$

**NOTE** The circuit does not require the resistance $R_x$ for the normal operation as an inverting amplifier. However, $R_x$ establishes a nonzero voltage at the noninverting terminal, and any voltage signals generated due to the op-amp offset parameters will appear at both the inverting and noninverting terminals. By making $R_x = R_1 || R_F$, which is equal to Thévenin’s equivalent resistance looking at the inverting terminal, we minimize or eliminate the effect of the offset voltage on the difference voltage $v_d$.

Using Kirchhoff’s current law at the inverting terminal, we get

$$i_S = i_f + i_i \quad (3.38)$$

For an ideal op-amp, $v_d = 0$ and $i_i \approx 0$. That is, Eq. (3.36) becomes

$$v_S = R_1 i_S$$

which gives

$$i_S = \frac{v_S}{R_1}$$

Also, $R_F i_f + v_O = 0$, which gives the feedback current as

$$i_f = -\frac{v_O}{R_F}$$

For $i_i \approx 0$, Eq. (3.38) becomes

$$i_S = i_f \quad \text{or} \quad \frac{v_S}{R_1} = -\frac{v_O}{R_F}$$
Therefore, the output voltage is related to the input voltage by

\[ v_O = -\left(\frac{R_F}{R_1}\right)v_S \]  

which gives the closed-loop voltage gain of the op-amp circuit as

\[ A_f = \frac{v_O}{v_S} = -\frac{R_F}{R_1} \]

(3.40)

Since \( v_d \approx 0 \), the effective input resistance \( R_{in} \) of the amplifier is given by

\[ R_{in} = \frac{v_S}{i_S} = \frac{v_S}{(v_S + v_d)/R_1} \approx R_1 \]

The effective output resistance is given by \( R_{out} = R_o \approx 0 \).

**CMRR of an Inverting Amplifier**

Similar to the CMRR expression in Eq. (3.25) for a noninverting amplifier, we can derive the CMRR of an inverting amplifier. Considering two input signals \( v_1 \) and \( v_2 \), \( v_{id} = v_2 - v_1 = -v_1 \) and \( v_{icm} = (v_2 + v_1)/2 = v_1/2 \) since \( v_2 = 0 \).

The voltage \( v_p \) at the noninverting terminal is \( v_p = 0 \), and the voltage \( v_n \) at the inverting terminal can be expressed in terms of \( v_O \) by

\[ v_n = v_O + \frac{R_F}{R_1 + R_F} (v_1 - v_O) = v_O \frac{R_1}{R_1 + R_F} + v_1 \frac{R_F}{R_1 + R_F} \]

Similar to the noninverting op-amp, let \( v_{do} \) and \( v_{cmo} \) denote the difference and common-mode signals at the op-amp input. Then, we have \( v_{do} = (v_p - v_n) \) and \( v_{cmo} = (v_p + v_n)/2 \). Thus, the output voltage \( v_O \) can be expressed in terms of the op-amp differential gain \( A_d \) and common-mode gain \( A_{cm} \) as

\[ v_O = A_d v_{do} + A_{cm} v_{cmo} = A_d (v_p - v_n) + A_{cm} \frac{(v_p + v_n)}{2} \]

After substituting for \( v_p = 0 \) and \( v_n \) using the relationships in terms of \( v_{id} = -v_1 \) and \( v_{cm} = v_1/2 \), we can find the following expression for \( v_O \):

\[ v_O = A_d (0 - v_n) + A_{cm} \frac{2}{2} (0 + v_n) = \left( v_{id} \frac{R_F}{R_1 + R_F} - v_O \frac{R_1}{R_1 + R_F} \right) \]

\[ + A_{cm} \left( v_{icm} \frac{R_F}{R_1 + R_F} + v_O \frac{R_1/2}{R_1 + R_F} \right) \]

(3.41)
We can solve this for $v_o$ after collecting all the terms containing $v_o$ in terms of $v_{id}$ and $v_{cm}$:

$$v_o = \frac{A_d R_F/R_1 v_{id} + A_{cm} R_F/R_1 v_{icm}}{1 + R_F/R_1 + A_d - A_{cm}/2} \quad (3.42)$$

Since $A_d >> A_{cm}$ and $A_d >> R_F/R_1 >> 1$, the denominator of Eq. (3.42) is approximately equal to $A_d$, and $v_o$ in Eq. (3.42) can be approximated to

$$v_o \approx \frac{A_d R_F}{A_d R_1} v_{id} + \frac{A_{cm} R_F}{A_d R_1} v_{icm} = \frac{R_F}{R_1} v_{id} + \frac{1}{CMRR_o} \frac{R_F}{R_1} v_{icm} \quad (3.43)$$

where CMRR$_o$ = $A_d/A_{cm}$ is the CMRR of the op-amp. From Eq. (3.43), we can find the differential voltage gain $A_{d-amp}$ and common-mode gain $A_{cm-amp}$ of the noninverting amplifier:

$$A_{d-amp} = \frac{R_F}{R_1} \quad (3.44)$$

$$A_{cm-amp} = \frac{1}{CMRR_o} \frac{R_F}{R_1} \quad (3.45)$$

Therefore, we can find the CMRR of the noninverting amplifier, which is the same as that of the op-amp:

$$CMRR_{amp} = \frac{A_{d-amp}}{A_{cm-amp}} = CMRR_o \quad (3.46)$$

**NOTES**

1. The negative sign in Eq. (3.39) signifies that the output voltage is out of phase with respect to the input voltage by 180° (in the case of an AC input) or of opposite polarity (in the case of a DC input).
2. The current $i_i$ flowing into the op-amp is very small, tending to zero, and the voltage $v_{id}$ at the inverting terminal is also very small, tending to zero. Although the inverting terminal is not the ground point, this terminal is said to be a virtual short.
3. We can design an inverting amplifier to give a specified gain simply by choosing the appropriate ratio $R_F/R_1$. A small value of $R_1$ will load the input source, and a large value of $R_F$ will increase the noise generated in the resistor. As a guide, all resistances in op-amp circuits should be between 1 kΩ and 10 MΩ.
4. If $R_1 = R_F$, Eq. (3.40) gives $A_F = -1$ and $v_O = -V_S$. The circuit then behaves as a unity-gain inverter (or simply an inverter).
5. Designing an inverting voltage amplifier is straightforward: Given $R_{in}$ and gain $A_F$, find $R_1$ and then find $R_F$.

**EXAMPLE 3.7**

*Designing an inverting op-amp circuit to limit the input current*  
A transducer produces an input-signal voltage of $V_S = 100$ mV with an internal resistance of $R_s = 2$ kΩ. Design the inverting op-amp amplifier of Fig. 3.11 by determining the values of $R_1$, $R_F$, and $R_C$. The output voltage should be $v_O = -8$ V. The current drawn from the transducer should not be more than 10 μA. Assume an ideal op-amp and $V_{CC} = V_{EE} = 15$ V.
Finding the voltage gain of an inverting op-amp circuit

The parameters of the op-amp circuit in Fig. 3.11 are $R_F = 800 \, \text{k}\Omega$, $R_1 = 10 \, \text{k}\Omega$, and $A_o = 2 \times 10^5$. Calculate (a) the closed-loop gain $A_f = v_O/v_S$, (b) the output voltage $v_O$, and (c) the errors in the output voltage $v_O$ and the gain $A_f$ if $A_o$ tends to infinity. Assume that source resistance $R_s = 0$ and $V_S = 100 \, \text{mV}$.

SOLUTION

$$R_1 = 10 \, \text{k}\Omega, \quad R_F = 800 \, \text{k}\Omega, \quad R_F/R_1 = 80, \quad A_o = 2 \times 10^5, \quad \text{and} \quad V_S = 100 \, \text{mV}.$$ From Fig. 3.11, $v_O = A_o v_d$ or $v_d = v_O/A_o$. The input current $i_S$ through $R_1$ can be found from

$$i_S = \frac{v_S + v_d}{R_1} = \frac{v_S + v_O/A_o}{R_1} \quad (3.47)$$
From Fig. 3.11, the output voltage is given by

\[ v_O = -v_d - i_1R_F = -v_d - i_SR_F \] (since \( i_1 \approx i_S \))

\[ = -\frac{v_O}{A_o} - \frac{v_S + v_O/A_o}{R_1}R_F \]

which, after simplification, gives the closed-loop voltage gain \( A_f \) as

\[ A_f = \frac{v_O}{v_S} = -\frac{R_F/R_1}{1 + (1 + R_F/R_1)/A_o} = -\frac{R_F}{R_1(1 + x)} \] (3.48)

where \( x = \frac{1}{A_o} \left( 1 + \frac{R_F}{R_1} \right) \) (3.49)

For a small value of \( x \), which is usually the case, \( (1 + x)^{-1} \approx 1 - x \), and Eq. (3.48) can be approximated by

\[ A_f = -\frac{R_F}{R_1} (1 - x) \] (3.50)

Therefore, the error introduced for a finite value of gain \( A_o \) is \( x \).

(a) From Eq. (3.49),

\[ x = \frac{1 + 80}{2 \times 10^5} = 40.5 \times 10^{-5} = 40.5 \times 10^{-3} \%

From Eq. (3.48),

\[ A_f = \frac{-80}{1 + 40.5 \times 10^{-5}} = -79.9676 \]

(b) The output voltage \( v_O \) is

\[ v_O = A_f v_S = -79.9676 \times 100 \times 10^{-3} = -7.99676 \text{ V} \]

(c) From Eq. (3.50), the error in the output voltage \( v_O \) is

\[ \Delta v_O = \frac{xR_F}{R_1} = 40.5 \times 10^{-5} \times 80 = 32.4 \text{ mV}, \text{ or } 0.0405 \%

The error in the gain \( A_f \) is

\[ \Delta A_f = x = 40.5 \times 10^{-5} = 0.0405 \% \]
EXAMPLE 3.9

Finding the parameters of a practical inverting op-amp circuit

(a) An inverting amplifier has \( R_1 = 10 \, \text{k}\Omega \) and \( R_F = 800 \, \text{k}\Omega \). The op-amp parameters are \( A_o = 2 \times 10^5 \), \( f_b = 10 \, \text{Hz} \), \( R_o = 75 \, \text{\mu}\Omega \), and \( R_i = 2 \, \text{M}\Omega \). The frequency of the input signal is \( f_s = 10 \, \text{kHz} \). Determine the unity-gain bandwidth \( f_{bw} \), the closed-loop voltage gain \( A_f \), and the closed-loop break frequency \( f_c \) of the op-amp.

(b) Use PSpice/SPICE to plot the closed-loop frequency response of the voltage gain. Assume \( v_s = 0.1 \, \text{V (AC)} \), and use the linear AC model.

SOLUTION

(a) Using Eq. (3.48), we find the frequency-dependent voltage gain of the inverting amplifier to be

\[
A_f(j\omega) = \frac{-R_F/R_1}{1 + (1 + R_F/R_1)/A_o(j\omega)}
\]

Substituting the frequency-dependent gain \( A_o(j\omega) \) from Eq. (3.2), we get

\[
A_f(j\omega) = \frac{-R_F/R_1}{1 + (1 + R_F/R_1)/A_o + jf(1 + R_F/R_1)/(A_o f_b)} \tag{3.51}
\]

since \( \omega = 2\pi f \). If we assume that \((1 + R_F/R_1) << A_o\), which is generally the case, and substitute \( f_{bw} = A_o f_b \), Eq. (3.51) becomes

\[
A_f(j\omega) = \frac{-R_F/R_1}{1 + jf(1 + R_F/R_1)/f_{bw}} \tag{3.52}
\]

which gives the closed-loop break (or 3-dB) frequency as

\[
f_c = \frac{f_{bw}}{1 + R_F/R_1} = \frac{f_{bw} R_1}{R_1 + R_F} = \beta f_{bw} = \beta A_o f_b \tag{3.53}
\]

where \( \beta = R_1/(R_1 + R_F) \) is called the feedback ratio or the feedback factor. (It should not be confused with the current gain \( \beta_F \) of a bipolar transistor.) Notice from Eq. (3.52) that the DC gain is \(-R_F/R_1\) (as expected), and it falls off at a rate of \(-20 \, \text{dB/decade}\) after a break frequency of \( f_c = \beta f_{bw} \).

For \( R_1 = 10 \, \text{k}\Omega \), \( R_F = 800 \, \text{k}\Omega \), \( A_o = 2 \times 10^5 \), \( f_b = 10 \, \text{Hz} \), and \( f = f_s = 10 \, \text{kHz} \),

\[
\frac{R_F}{R_1} = \frac{800 \, \text{k}\Omega}{10 \, \text{k}\Omega} = 80
\]

\[
\beta = \frac{R_1}{(R_1 + R_F)} = \frac{10 \, \text{k}\Omega}{(10 \, \text{k}\Omega + 800 \, \text{k}\Omega)} = 12.346 \times 10^{-3}
\]

which is small compared to \( A_o = 2 \times 10^5 \).

From Eq. (3.4),

\[
f_{bw} = A_o f_b = 2 \times 10^5 \times 10 = 2 \, \text{MHz}
\]
Substituting the values in Eq. (3.52), we get the voltage gain at $f = f_s = 10$ kHz as

$$A_f(j\omega) = \frac{-80}{1 + j2\pi f(1 + 80)/2\pi f_{bw}} = \frac{-80}{1 + j(10 \times 10^3) \times 81/(2 \times 10^6)}$$

$$= \frac{-80}{1 + j0.405} = -74.15 \angle -22^\circ$$

Thus, the magnitude of the closed-loop voltage gain at $f_s = 10$ kHz is $-74.15$. If the input is a sinusoidal signal, the output voltage will be phase shifted by $(180^\circ - 22^\circ) = 158^\circ$.

From Eq. (3.53),

$$f_c = \beta f_{bw} = 12.346 \times 10^{-3} \times 2 \times 10^6 = 24.69$$

(b) The inverting amplifier for PSpice simulation is shown in Fig. 3.12(a).

The frequency response, which is shown in Fig. 3.13, gives the low-frequency gain $A_f(\text{dc}) = 79.95$ and $A_f = 61.67$ at $f_s = 10$ kHz. At $A_f = 56.48$ (estimated value is $0.707 \times 79.95 = 56.53$), $f_c = 12.198$ kHz. The calculated values are $f_c = 24.69$ kHz and $A_f = 74.15$ (at $f_s = 10$ kHz). However, this simulation was done using the nonlinear macromodel of UA741. If we run the simulation with the linear op-amp model shown in Fig. 3.12(b), we get the low-frequency gain $A_f(\text{dc}) = 79.93$ and $A_f = 74.07$ at $f_s = 10$ kHz. At $A_f = 56.45$ (estimated value is $0.707 \times 79.93 = 56.51$), $f_c = 24.5$ kHz. The calculated values are $f_c = 24.69$ kHz and $A_f = 74.15$ (at $f_s = 10$ kHz).
3.5 Op-Amp Applications

The applications of op-amps are endless, and there are numerous books about op-amps [4–6, 7–10]. Most of the applications are derived from the basic noninverting and inverting, configurations described in Sec. 3.4. In this section we discuss several applications of op-amps.

3.5.1 Integrators

If the resistance $R_F$ in the inverting amplifier of Fig. 3.11 is replaced by a capacitance $C_F$, the circuit will operate as an integrator. Such a circuit is shown in Fig. 3.14(a). $R_x$ is included to minimize the effect of op-amp imperfections (i.e., the input biasing current, which will be discussed in Sec. 14.3). The value of $R_x$ should be made equal to $R_1$. The impedance of $C_F$ in Laplace’s domain is $Z_F = 1/(sC_F)$. Applying Eq. (3.39) gives the output voltage in Laplace’s domain as

$$V_o(s) = -\left(\frac{Z_F}{Z_1}\right)V_s(s) = -\frac{1}{sR_1C_F}V_s(s)$$  \hspace{1cm} (3.54)

from which the output voltage in the time domain becomes

$$v_o(t) = -\frac{1}{R_1C_F}\int_0^tv_s\,dt - v_C(t = 0)$$  \hspace{1cm} (3.55)
where \( v_C(t = 0) = V_{co} \) represents the initial capacitor voltage. That is, the output voltage is given by the integral of the input voltage \( v_S \). Equation (3.55) can also be derived from a circuit analysis similar to that discussed in Sec. 3.4.2. That is,

\[
i_f = i_S = \frac{v_S + v_d}{R_1} = \frac{v_S}{R_1}
\]  

(3.56)
since the op-amp input current is zero. Therefore, the output voltage, which is the negative of the capacitor voltage, is given by

\[
v_O(t) = -v_C(t) = -\frac{1}{C_F} \int_0^t i_S \, dt - v_C(t = 0)
\]  

(3.57)

Substituting \( i_S = v_S/R_1 \) from Eq. (3.56) into Eq. (3.57), we can obtain Eq. (3.55). Time constant \( \tau_i = R_1C_F \) for Fig. 3.14(a) is known as the integration time constant. If the input is a constant current \( i_S = I_S \), then Eq. (3.57) gives

\[
v_O(t) = -\frac{I_S t}{C_F} - v_C(t = 0) = -\frac{Q}{C_F} - v_C(t = 0)
\]  

(3.58)

That is, the output voltage is the integral of the input current \( I_S \) and is proportional to the input charge \( Q \). Thus, the circuit in Fig. 3.14(a) can also be used as a current integrator, or charge amplifier. The plot of the output voltage for a pulse input is shown in Fig. 3.14(b). Due to the half-wave symmetry of the input voltage \( v_S \), the output voltage \( v_O \) will also be half-wave symmetrical. That is, the first zero crossing of the output voltage will be at \( t = T/4 \), where \( T \) is the period of the input voltage. The area \( +A \) under the input voltage during the interval \( (T/2 \) to \( T/4) \) will be equal and opposite to the area \( -B \) during the interval \( (3T/4 \) to \( T/2) \) such that the output waveform crosses the zero axis at every \( T/2 \) interval.

At lower frequencies, the impedance \( Z_F \) of \( C_F \) will increase, and less signal will be fed back to the inverting terminal of the op-amp. Thus, the output voltage will increase. At higher frequencies, the impedance \( Z_F \) will decrease, causing more signal to be fed back to the inverting terminal. Thus, the output voltage will decrease. Therefore, an integrator circuit behaves like a low-pass filter. The magnitude plot of the voltage gain \( V_o(j\omega)/V_s(j\omega) \) in Eq. (3.54) will have a low-pass characteristic with a zero break frequency, as shown in Fig. 3.14(c).

For the case in which the input signal is a constant DC voltage, Eq. (3.55) simplifies to

\[
v_O(t) = -\left( \frac{V_S}{R_1C_F} \right) t - V_{co}
\]  

(3.59)

\[\text{FIGURE 3.14} \quad \text{Integrator circuit}\]
Typical plots of some input signals and the resulting output signals are shown in Fig. 3.15.

In practice, as a result of its imperfections (e.g., drift, input offset current), an op-amp produces an output voltage even if the input signal is zero ($v_S = 0$), and the capacitor will be charged by the small but finite current through it. The capacitor prevents any DC signal from feeding back from the output terminal to the input side of the op-amp. As a result, the capacitor will be charged continuously, and the output voltage will build up until the op-amp saturates. A resistor with a large value of $R_F$ is normally connected in parallel with the capacitor of capacitance $C_F$, as shown in Fig. 3.16. $R_F$ provides the DC feedback and overcomes this saturation problem. Time constant $\tau_F (= R_F C_F)$ must be larger than the period $T_s (= 1/f_s)$ of the input signal. A ratio of 10 to 1 is generally adequate; that is, $\tau_F = 10 T_s$. For Fig. 3.16, the feedback impedance is

$$Z_F = R_F \parallel \left( \frac{1}{sC_F} \right) = \frac{R_F}{1 + sR_F C_F}$$

and Eq. (3.39) gives the output voltage in Laplace’s domain as

$$V_o(s) = -\frac{R_F}{1 + sR_F C_F} V_i(s)$$

(3.60)
For a step input voltage of $v_S = V_S$, $V_S(s) = V_S/s$, and Eq. (3.60) can be simplified to give the output voltage in the time domain as

$$v_O(t) = -V_S R_F \left( 1 - e^{-t/R_F C_F} \right)$$  \hspace{1cm} (3.61)$$

For $t \leq 0.1 R_F C_F$, Eq. (3.61) can be approximated by

$$v_O(t) = -V_S R_F \left( \frac{t}{R_F C_F} \right) = - \left( \frac{V_S}{R_1 C_F} \right) t$$  \hspace{1cm} (3.62)$$

which is the time integral of the input voltage. Therefore, the analysis and the input–output relation of the integrator in Fig. 3.14 can be applied to the one in Fig. 3.16, provided $\tau_F \geq 10T$.

**EXAMPLE 3.10**

**Designing an op-amp integrator**

(a) Design an integrator of the form shown in Fig. 3.16. The frequency of the input signal is $f_s = 500$ Hz. The voltage gain should be unity at a frequency of $f_1 = 1590$ Hz. That is, the unity-gain bandwidth is $f_{bw} = 1590$ Hz.

(b) The integrator in part (a) has $V_{CC} = 12$ V, $V_{EE} = -12$ V, and maximum voltage swing $\pm 10$ V. The initial capacitor voltage is $V_{co} = 0$. Draw the waveform of the output voltage for the input voltage shown in Fig. 3.17.

(c) Use PSpice/SPICE to plot the output voltage for the input voltage in part (b).

**SOLUTION**

(a) The steps in completing the design are as follows:

**Step 1.** Choose a suitable value of $C_F$: Let $C_F = 0.1 \mu$F.

**Step 2.** Calculate the time constant required to satisfy the unity-gain frequency requirement:

$$\tau_i = \frac{1}{2\pi f_1} = \frac{1}{2\pi \times 1590 \text{ Hz}} = 100 \mu \text{s}$$

**Step 3.** Calculate the value of $R_1$ from $\tau_i$:

$$R_1 = \frac{\tau_i}{C_F} = \frac{100 \mu \text{s}}{0.1 \mu \text{F}} = 1 \text{ k\Omega}$$

![FIGURE 3.17 Input voltage for Example 3.10](image)
The integrator for PSpice simulation is shown in Fig. 3.19. The plot of the output voltage \( V_o \) is shown in Fig. 3.20. Under the steady-state condition for the time interval of 10 ms to 14 ms after the initial transient. Note that the input signal voltage is delayed by 0.5 ms for PSpice simulation, and the PSpice model parameters of VS are

\[ V_{1} = -2V \quad V_{2} = 2V \quad TD = 0.5\text{ms} \quad TR = 1\text{ns} \quad TF = 1\text{ns} \quad PW = 1\text{ms} \quad PER = 2\text{ms} \]

The output voltage waveform is close to the expected values: \( V_{o(max)} = 10.103 \text{ V (expected 10 V)} \) and \( V_{o(min)} = -10.238 \text{ V. If we plot the output voltage, starting from 0, we can see the transient behavior, and it will take a number of cycles before the waveform reaches its steady-state condition.}

**NOTE:** While running the PSpice simulation, you must select Use Initial Condition in the setup; otherwise the output plot will differ from what is shown in Fig. 3.20.

**Step 4.** Choose time constant \( \tau_F = 10T_s = 10/f_s \):

\[
\tau_F = \frac{10}{500 \text{ Hz}} = 20 \text{ ms}
\]

**Step 5.** Calculate the value of \( R_F \) from \( \tau_F \):

\[
R_F = \frac{\tau_F}{C_F} = \frac{20 \text{ ms}}{0.1 \mu\text{F}} = 200 \text{ k}\Omega
\]

(b) \( V_{sat} = \pm 10 \text{ V, and } \tau_i = R_1C_F = 1 \times 10^3 \times 0.1 \times 10^{-6} = 0.1 \text{ ms. Since } \tau_F \gg \tau_i, \text{ the effect of } \tau_F \text{ can be neglected.}

Under the steady-state condition after the initial transient, the capacitor voltage at \( t = 0 \) will be \( V_{co} = -10 \text{ V (negative of the output voltage). If we assume an initial capacitor voltage of } V_{co} = 0, \text{ then we must start the integration at } T/4.

For \( 0 \leq t \leq 1 \text{ ms: From Eq. (3.55), the output voltage is given by}

\[
v_O = 10 - \frac{1}{R_1C_F} \int_0^t 2 \, dt = 10 - 2 \times 10,000t
\]

where \( t \) is in milliseconds. At \( t = 1 \text{ ms, } v_O = -10 \text{ V, which is more than the saturation voltage and thus is not possible. The time required for the output voltage to reach the saturation voltage of } -10 \text{ V is } t_1 = 10/(2 \times 10,000) = 0.5 \text{ ms. For } 0.5 \text{ ms} \leq t \leq 1 \text{ ms, the capacitor voltage is } V_{co} = 10 \text{ V.}

For \( 1 \text{ ms} \leq t \leq 2 \text{ ms: From Eq. (3.55), the output voltage is given by}

\[
v_O = -10 + \frac{1}{R_1C_F} \int_0^{t-1} 2 \, dt = -10 + 2 \times 10,000(t - 1)
\]

where \( t \) is in milliseconds. At \( t = 2 \text{ ms, } v_O = 10 \text{ V, and the capacitor voltage is } V_{co} = -10 \text{ V.}

For \( 2 \text{ ms} \leq t \leq 3 \text{ ms: From Eq. (3.55), the output voltage is given by}

\[
v_O = 10 - \frac{1}{R_1C_F} \int_0^{t-2} 2 \, dt = 10 - 2 \times 10,000(t - 2)
\]

where \( t \) is in milliseconds. At \( t = 3 \text{ milliseconds, } v_O = -10 \text{ V, and the capacitor voltage is } V_{co} = 10 \text{ V.}

For \( 3 \text{ ms} \leq t \leq 4 \text{ ms: From Eq. (3.55), the output voltage is given by}

\[
v_O = -10 + \frac{1}{R_1C_F} \int_0^{t-3} 2 \, dt = -10 + 2 \times 10,000(t - 3)
\]

where \( t \) is in milliseconds. At \( t = 4 \text{ ms, } v_O = 10 \text{ V, and the capacitor voltage is } V_{co} = -10 \text{ V.}

The waveforms for input and output voltages are shown in Fig. 3.18.

(c) The integrator for PSpice simulation is shown in Fig. 3.19. The plot of the output voltage \( v_O = V(CF; 2) \) is shown in Fig. 3.20. Under the steady-state condition for the time interval of 10 ms to 14 ms after the initial transient. Note that the input signal voltage is delayed by 0.5 ms for PSpice simulation, and the PSpice model parameters of VS are

\[ V_1 = -2V \quad V_2 = 2V \quad TD = 0.5\text{ms} \quad TR = 1\text{ns} \quad TF = 1\text{ns} \quad PW = 1\text{ms} \quad PER = 2\text{ms} \]

The output voltage waveform is close to the expected values: \( V_{o(max)} = 10.103 \text{ V (expected 10 V)} \) and \( V_{o(min)} = -10.238 \text{ V. If we plot the output voltage, starting from 0, we can see the transient behavior, and it will take a number of cycles before the waveform reaches its steady-state condition.
Introduction to Operational Amplifiers and Applications

**Figure 3.18** Waveforms for Example 3.10

**Figure 3.19** Integrator circuit for PSpice simulation

**Figure 3.20** PSpice plots for Example 3.10
Replacing the op-amp by its equivalent circuit gives the integrator shown in Fig. 3.21(a). Capacitor $C_i$ is the input capacitor of the op-amp, and it influences the high cutoff frequency. There will not be any low cutoff frequency, and the circuit will behave as a high-pass circuit. There will be two high break frequencies: $\omega_i$ for $C_i$ and $\omega_F$ for $C_F$. The low-frequency gain will be $-R_F/R_1$. Thus, the transfer function can be expressed as

$$A_f(s) = \frac{-R_F/R_1}{(1 + s/\omega_i)(1 + s/\omega_F)} \quad (3.63)$$

Since $C_F$ is connected between the input and output sides of the op-amp and the voltage gain is very high, $C_F$ will dominate the high cutoff frequency $f_H = f_F = 1/(2\pi C_F R_F)$. That is, $\omega_i > > \omega_F$. The typical frequency response is shown in Fig. 3.21(b).

**EXAMPLE 3.11**

Finding the 3-dB frequency of an integrator using Miller’s theorem \[ \text{The integrator of Fig. 3.14(a) has } C_F = 0.001 \mu F \text{ and } R_1 = 1 \text{ k}\Omega. \text{ The open-loop gain of the op-amp is } A_o = 2 \times 10^5. \text{ Use Miller’s theorem (discussed in Sec. 2.6) to find the 3-dB frequency of the integrator.} \]

**SOLUTION**

Miller’s theorem can be applied to replace the feedback capacitance $C_F$ by an equivalent input capacitance $C_x$ and an output capacitance $C_y$, as shown in Fig. 3.22. With the open-loop gain $A_o = A_{vo}$ and the capacitive impedance $Z_F = 1/(j2\pi f C_F)$, we can apply Eqs. (2.102) and (2.103) to find the Miller capacitances:

$$C_x = C_F(1 + A_o) = 0.001 \mu F \times (1 + 2 \times 10^5) = 200.001 \mu F$$
$$C_y = C_F(1 + 1/A_o) = C_F = 0.001 \mu F$$
The output voltage in Laplace’s domain is

\[ V_o(s) = A_0 V_d(s) \]

The transfer function between the input and output voltages is given by

\[ A(s) = \frac{V_o(s)}{V_d(s)} = \frac{A_0}{1 + R_1 C_x s} \]

Therefore, the 3-dB frequency is

\[ \omega_b = \frac{1}{(R_1 C_x)} = \frac{1}{(1 \times 10^{-3} \times 200.001 \times 10^{-6})} = 5 \text{ rad/s} \quad \text{or} \quad f_b = \omega_b/2\pi = 0.7958 \text{ Hz} \]

**EXAMPLE 3.12**

**Finding the frequency response of an op-amp integrator**  The op-amp integrator in Fig. 3.21(a) has \( R_1 = 1 \, \text{k}\Omega, R_F = 200 \, \text{k}\Omega, C_F = 0.1 \, \mu\text{F}, C_i = 1.5 \, \text{pF}, R_i = 2 \, \text{M}\Omega, R_o = 75 \, \Omega, \) and open-loop voltage gain \( A_0 = 2 \times 10^5. \)

(a) Calculate the low-frequency (or DC) voltage gain \( A_{PB} = v_o/v_s. \)

(b) Use the zero-value method to calculate the high 3-dB frequency \( f_H. \)

(c) Use PSpice/SPICE to plot the frequency response.

**SOLUTION**

(a) Assuming all capacitors are open-circuited, the low-frequency pass-band voltage gain can be found as follows:

\[ A_{PB} \approx -\frac{R_F}{R_1} = -\frac{200 \, \text{k}\Omega}{1 \, \text{k}\Omega} = -200 \]
(b) The high-frequency equivalent circuits of the op-amp integrator are shown in Fig. 3.23. If $C_F$ is open-circuited and the voltage-controlled voltage source is converted to a voltage-controlled current source, we get the circuit in Fig. 3.23(a). The transconductance $g_m$ is given by

$$g_m = \frac{A_o}{R_o}$$

$$= \frac{2 \times 10^5}{75} = 2.67 \text{kA/V}$$

Applying a test voltage $v_x$ and KVL, we get

$$v_x = R_Fi_x + (R_o \parallel R_L)(i_x - g_mv_x)$$

which gives the resistance $R_x$ as

$$R_x = \frac{v_x}{i_x} = \frac{R_F + R_o \parallel R_L}{1 + g_m(R_o \parallel R_L)}$$

$$= \frac{200 \text{kΩ} + 75 \Omega \parallel 10 \text{kΩ}}{1 + 2.67 \times 10^3 \text{A/V} \times (75\Omega \parallel 10 \text{kΩ})} = 1 \Omega$$

Thevenin’s equivalent resistance presented to $C_i$ is

$$R_{Ci} = R_1 \parallel R_x = 1 \text{kΩ} \parallel 2 \text{MΩ} \parallel 1 \Omega \approx 1 \Omega$$

and

$$f_{Ci} = \frac{\omega_i}{2\pi} = \frac{1}{[2\pi (C_iR_{Ci})]} = \frac{1}{[2\pi \times (1.5 \text{pF} \times 1)]} = 106.1 \times 10^9 \text{Hz}$$

FIGURE 3.23  High-frequency equivalent circuits for op-amp integrator
If we assume $C_i$ is open-circuited, the equivalent circuit is shown in Fig. 3.23(b). Applying a test voltage $v_y$ and using Eq. (2.116), we can find the equivalent resistance $R_y = v_y/i_y$. That is,

$$R_y = \frac{v_y}{i_y} = R_o \parallel R_L + (R_1 \parallel R_0)(1 + g_m(R_o \parallel R_L))$$

$$= 75 \Omega \parallel 10 \text{k}\Omega + (1 \text{k}\Omega \parallel 2 \text{M}\Omega)(1 + 2.667 \times 10^3 \text{A}/\text{V} \times (75 \Omega \parallel 10 \text{k}\Omega)) = 198.4 \text{M}\Omega$$

Thevenin’s equivalent resistance presented to $C_F$ is

$$R_{CF} = R_F \parallel R_y = 200 \text{k}\Omega \parallel 198.4 \text{M}\Omega = 199.8 \text{k}\Omega$$

Thus, using Eq. (2.112), high 3-dB frequency $f_H$ is

$$f_H = \frac{1}{2\pi (R_C C_i + R_{CF} C_F)} = \frac{1}{2\pi (1 \times 1.5 \text{pF} + 199.8 \text{k}\Omega \times 0.1 \mu\text{F})} = 7.97 \text{Hz}$$

which is dominated by $C_F$ as expected and can be approximated by

$$f_H = \frac{\omega_F}{2\pi} = \frac{1}{2\pi C_F R_{CF}} = \frac{1}{[2\pi \times (0.1 \mu\text{F} \times 199.8 \text{k}\Omega)]} = 7.97 \text{Hz}$$

There really was no need to find the value of $R_y$, which is usually very large for an op-amp circuit, and $R_{CF} \approx R_F$.

(e) Node numbers are assigned to the AC equivalent circuit of Fig. 3.21(a) for PSpice simulation. The PSpice plot of the frequency response is shown in Fig. 3.24, which gives the midfrequency gain as $|A_{\text{low}}| = 198.16$. The high 3-dB frequency is approximately $f_H = 7.96$ Hz. The expected values are $f_H = 7.97 \text{Hz}$ and $A_{\text{PB}} = -200$.

### FIGURE 3.24
Frequency response for Example 3.12

#### 3.5.2 Differentiators

If the resistance $R_1$ in the inverting amplifier of Fig. 3.11 is replaced by a capacitance $C_1$, as shown in Fig. 3.25(a), the circuit will operate as a differentiator. The value of $R_x$ should be made equal to $R_F$. The impedance of $C_1$ in Laplace’s transform is $Z_1 = 1/(sC_1)$. Using Eq. (3.39), we can find the output voltage in Laplace’s domain as

$$V_o(s) = -\left(\frac{R_F}{Z_1}\right)V_s(s) = -sR_FC_1V_s(s)$$

$$V_o(s) = -\left(\frac{R_F}{Z_1}\right)V_s(s) = -sR_FC_1V_s(s)$$

(3.67)
which gives the output voltage in the time domain as

\[ v_O = -R_F C_1 \frac{d v_S}{dt} \]  

This equation can also be derived from a circuit analysis similar to that discussed in Sec. 3.4.2. That is,

\[ i_S = i_f = C_1 \frac{d v_S}{dt} \]  
\[ v_O = -R_F i_f = -R_F i_S \]  

Substituting \( i_S \) from Eq. (3.69) into Eq. (3.70) gives Eq. (3.68). Time constant \( \tau_d = R_F C_1 \) in Fig. 3.25(a) is known as the differentiator time constant. The output voltage in response to a triangular wave is shown in Fig. 3.25.

A differentiator circuit is useful in producing sharp trigger pulses to drive other circuits. When the frequency is increased, the impedance \( Z_1 \) of \( C_1 \) decreases and the output voltage increases. Therefore, a differentiator circuit behaves like a high-pass network. The magnitude plot of the voltage gain \( V_o(j\omega)/V_s(j\omega) \) in Eq. (3.67) has a high-pass characteristic with an infinite break frequency, as shown in Fig. 3.25(c). Typical plots of some input signals and the resulting output signals are shown in Fig. 3.26.

**FIGURE 3.25** Differentiator circuit

**FIGURE 3.26** Typical input and output signals of a differentiator
If there is any sharp change in the input voltage \( v_S(t) \) due to noise or picked-up interference, there will be amplified spikes at the output, and the circuit will behave like a noise magnifier. Thus, this type of differentiating circuit is not often used. A modified circuit that is often utilized as a differentiator is shown in Fig. 3.27(a), in which a small resistance \( R_1 \) is connected in series with \( C_1 \) to limit the gain at high frequencies. However, this arrangement also limits the high-frequency range, as shown in the magnitude plot in Fig. 3.27(b).

The impedance \( Z_1 \) for \( R_1 \) and \( C_1 \) in Laplace’s domain is

\[
Z_1 = R_1 + \frac{1}{sC_1} = \frac{1 + sR_1C_1}{sC_1}
\]

Using Eq. (3.67), we have for the transfer function of the circuit in Fig. 3.27(a)

\[
A_f(s) = \frac{V_o(s)}{V_s(s)} = -\frac{R_F}{Z_1} = -\frac{R_FC_1s}{1 + sR_1C_1}
\]  

(3.71)

For \( s = j\omega \),

\[
A_f(j\omega) = -\frac{R_FC_1j\omega}{1 + j\omega R_1C_1}
\]  

(3.72)

The magnitude of Eq. (3.72) is given by

\[
|A_f(j\omega)| = \frac{R_FC_\omega}{\sqrt{1 + (\omega R_1C_1)^2}}
\]  

(3.73)

Therefore, the break frequency is \( \omega_b = 1/(R_1C_1) \). For frequencies greater than \( \omega_b \), \((\omega R_1C_1)^2 >> 1\), and Eq. (3.73) reduces to

\[
|A_f(j\omega)| = \frac{R_F}{R_1}
\]  

(3.74)
EXAMPLE 3.13

Designing an op-amp differentiator

(a) Design a differentiator of the form shown in Fig. 3.27(a) to satisfy the following specifications: gain-limiting break frequency $f_b = 1 \text{ kHz}$, and maximum closed-loop gain $A_{f_{(\text{max})}} = 10$. Determine the values of $R_1$, $R_F$, and $C_1$.

(b) Use PSpice/SPICE to plot the frequency response for part (a). Assume a sinusoidal input voltage of rms value $V_s = 0.1 \text{ V}$.

SOLUTION

$A_{f_{(\text{max})}} = 10$, and $f_b = 1 \text{ kHz}$.

(a) The steps in completing the design are as follows:

Step 1. Choose a suitable value for capacitance $C_1$: Let $C_1 = 0.1 \mu\text{F}$.

Step 2. Calculate the value of $R_1$ from the break frequency $f_b$:

$$f_b = \frac{1}{(2\pi R_1 C_1)}$$

$$1 \text{ kHz} = \frac{1}{(2\pi R_1 \times 0.1 \times 10^{-6})}$$

$$R_1 = 1592 \Omega$$

Step 3. Calculate the value of $R_F$ from Eq. (3.74):

$$A_{f_{(\text{max})}} = \frac{R_F}{R_1}$$

$$R_F = 1592 \times A_{f_{(\text{max})}} = 1592 \times 10 = 15.92 \text{ k}\Omega$$

(b) The differentiator circuit for PSpice simulation is shown in Fig. 3.28. The plot of the frequency response for the output voltage is shown in Fig. 3.29, which gives $A_{f_{(\text{max})}} = 9.995$ (expected value is $100 \times 0.1 = 10$).
Frequency Response of Op-Amp Differentiators

Replacing the op-amp by its equivalent circuit gives the differentiator shown in Fig. 3.30(a). The addition of capacitor $C_1$ to the integrator in Fig. 3.21(a) sets a low cutoff frequency $\omega_L$. Thus, the transfer function can be expressed as

$$A_f(s) = \frac{-\left(R_F/R_1\right)s}{(s + \omega_L)(1 + s/\omega_F)(1 + s/\omega_i)}$$  \hspace{1cm} (3.75)$$

$C_F$ will dominate the high cutoff frequency $f_H$. That is, $\omega_L < \omega_H \ll \omega_i$. The typical frequency response is shown in Fig. 3.30(b). The output will increase with frequency until $f = f_L = \frac{1}{2\pi C_1 R_1}$ and then remain constant between $f_L$ and $f_H$. The circuit can be made to operate effectively until $f_L$ only, after which we can let the gain fall by making $f_L = f_H$, as shown in Fig. 3.30(b) by the light-colored line.

FIGURE 3.30  Op-amp differentiator circuit

The break frequency $f_b$ (at $A_f = 9.995 \times 0.707 = 7.07$) is 983 Hz (expected value is 1 kHz). The upper frequency limit (i.e., 95 kHz) is due to the internal frequency behavior of the op-amp.
EXAMPLE 3.14

Designing a differentiator circuit to give a specified frequency response  Design a differentiator circuit as shown in Fig. 3.30(a) to give (a) $f_L = 1$ kHz and $f_H = 5$ kHz and (b) $f_L = f_H = 5$ kHz. The pass-band gain is $A_{PB} = -20$. The op-amp parameters are $C_1 = 1.5$ pF, $R_1 = 2$ MΩ, $R_0 = 75$ Ω, and open-loop voltage gain $A_o = 2 \times 10^5$.

SOLUTION

If we assume that $C_1$ is short-circuited and the other capacitors are open-circuited, the pass-band voltage gain is given by $A_{PB} \approx -R_F / R_1$. If we let $R_1 = 5$ kΩ,

$$R_F = \left| A_{PB} \right| R_1 = 20 \times 5 = 100 \text{ kΩ}$$

(a) We will first consider $f_L = 1$ kHz and $f_H = 5$ kHz. The low-frequency equivalent circuit is shown in Fig. 3.31. We can see from Eq. (3.65) that the effective resistance $R_x$ due to $R_F$ is very small because the voltage gain $A_o$ (i.e., $g_m A_o R_1$) is very large and the op-amp input voltage $v_d$ is very small. Thus, Thevenin’s equivalent resistance seen by $C_1$ becomes

$$R_{C1} = R_1 + (R_i \parallel R_x) \approx R_1$$

and Thevenin’s equivalent resistance seen by $C_F$ becomes

$$R_{CF} = R_F \parallel R_y \approx R_F$$

The low 3-dB frequency is given by

$$f_L = \frac{1}{2\pi R_{C1} C_1}$$

which gives

$$C_1 = \frac{1}{2\pi R_1 f_L} = \frac{1}{2\pi \times 5 \text{ kΩ} \times 1 \text{ kHz}} = 31.83 \text{ nF}$$

FIGURE 3.31  Low-frequency equivalent circuits for an op-amp differentiator
3.5.3 Differential Amplifiers

In the differential amplifier configuration, shown in Fig. 3.32, two input voltages \(v_a\) and \(v_b\) are applied—one to the noninverting terminal and another to the inverting terminal. Resistances \(R_a\) and \(R_x\) are used to step down the voltage applied to the noninverting terminal. Let us apply the superposition theorem to find the output voltage \(v_O\). That is, we will find the output voltage \(v_{oa}\), which is due to the input voltage \(v_a\) only, and then we will find the output voltage \(v_{ob}\), which is due to \(v_b\) only. The output voltage will be the sum of \(v_{oa}\) and \(v_{ob}\).

The voltage \(v_p\) can be related to the input voltage \(v_a\) by

\[
v_p = \frac{R_x}{R_x + R_a} v_a
\]

(3.78)

Applying Eqs. (3.18) and (3.78) gives the output voltage \(v_{oa}\), which is due to the input at the noninverting terminal, as

\[
v_{oa} = \left(1 + \frac{R_F}{R_1}\right)v_p = \left(1 + \frac{R_F}{R_1}\right)\left(\frac{R_x}{R_x + R_a}\right)v_a
\]

(3.79)

\[\text{FIGURE 3.32 Differential amplifier}\]
Applying Eq. (3.39) gives the output voltage $v_{ob}$, which is due to the input at the inverting terminal, as

$$v_{ob} = -\frac{R_F}{R_1} v_b$$  \hspace{1cm} (3.80)

Therefore, the resultant output voltage is given by

$$v_O = v_{ob} + v_{oa} = -\frac{R_F}{R_1} v_b + \left(1 + \frac{R_F}{R_1}\right) \left(\frac{R_x}{R_x + R_a}\right) v_a$$  \hspace{1cm} (3.81)

which, for $R_a = R_1$ and $R_F = R_x$, becomes

$$v_O = (v_a - v_b) \frac{R_F}{R_1}$$  \hspace{1cm} (3.82)

Thus, the circuit in Fig. 3.32 can operate as a differential voltage amplifier with a closed-loop voltage gain of $A_f = R_F / R_1$. For example, if $v_a = 3 \text{ V}$, $v_b = 5 \text{ V}$, $R_a = R_1 = 12 \text{ k}\Omega$, and $R_F = R_x = 24 \text{ k}\Omega$, then Eq. (3.82) gives

$$v_O = \frac{(3 - 5) \times 24 \text{ k}\Omega}{12 \text{ k}\Omega} = -4 \text{ V}$$

If all the resistances have the same values (i.e., $R_a = R_1 = R_F = R_x$), Eq. (3.82) is reduced to

$$v_O = v_a - v_b$$  \hspace{1cm} (3.83)

in which case the circuit will operate as a difference amplifier. For example, if $v_a = 3 \text{ V}$, $v_b = 5 \text{ V}$, and $R_a = R_F = R_1 = R_x = 20 \text{ k}\Omega$, then Eq. (3.83) gives

$$v_O = v_a - v_b = 3 - 5 = -2 \text{ V}$$

**CMRR of a Differential Amplifier**

Similar to the CMRR expression in Eq. (3.46) for an inverting amplifier, we can derive the CMRR of a differential amplifier. Considering two input signals $v_b$ and $v_a$, we have $v_{id} = v_a - v_b$ and $v_{icm} = (v_a + v_b)/2$. The voltage $v_p$ at the noninverting terminal is

$$v_p = \frac{R_x}{R_a + R_x} v_a = \frac{R_F}{R_1 + R_F} v_a \hspace{1cm} \text{for} \hspace{0.5cm} R_a = R_1 \hspace{0.5cm} \text{and} \hspace{0.5cm} R_x = R_F$$

and the voltage $v_n$ at the inverting terminal can be expressed in terms of $v_O$ by

$$v_n = v_O + \frac{R_F}{R_1 + R_F} (v_b - v_O) = v_O \frac{R_1}{R_1 + R_F} + v_b \frac{R_F}{R_1 + R_F}$$

Similarly, let $v_{do}$ and $v_{cmo}$ denote the differential and common-mode signals at the op-amp input. Then, we have $v_{do} = (v_p - v_n)$ and $v_{cmo} = (v_p + v_n)/2$. Thus, the output voltage $v_O$ can be expressed in terms of the op-amp differential gain $A_d$ and common-mode gain $A_{cm}$ as

$$v_O = A_d v_{do} + A_{cm} v_{cmo} = A_d (v_p - v_n) + \frac{A_{cm} (v_p + v_n)}{2}$$

After substituting for $v_p$ and $v_n$ using the relationships in terms of $v_{id}$ and $v_{icm}$, we can find the following expression for $v_O$:

$$v_O = A_d (v_p - v_n) + \frac{A_{cm}}{2} (v_p + v_n) = \frac{A_d}{R_1 + R_F} (R_F v_{id} - v_O R_1) + \frac{A_{cm}}{R_1 + R_F} \left( R_F v_{icm} + \frac{v_O R_1}{2} \right)$$
This can be solved for \( v_O \) after we collect all the terms containing \( v_O \) in terms of \( v_{id} \) and \( v_{cm} \):

\[
v_O = \frac{A_d R_F / R_1 v_d + A_{cm} R_F / R_1 v_{icm}}{1 + R_F / R_1 + A_d - A_{cm}/2}
\]  

(3.84)

Since \( A_d \gg A_{cm} \) and \( A_d \gg R_F R_1 \gg 1 \), the denominator of Eq. (3.84) is approximately equal to \( A_d \), and \( v_O \) in Eq. (3.84) can be approximated to

\[
v_O \approx \frac{A_d R_F}{A_d R_1} v_{id} + \frac{A_{cm} R_F}{A_c R_1} v_{icm} = \frac{R_F}{R_1} v_{id} + \frac{1}{\text{CMRR}_o} \frac{R_F}{R_1} v_{icm}
\]  

(3.85)

where \( \text{CMRR}_o = A_d/\text{Acm} \) is the CMRR of the op-amp. From Eq. (3.85), we can find the differential voltage gain \( A_{d-amp} \) and the common-mode gain \( A_{cm-amp} \) of the difference amplifier:

\[
A_{d-amp} = \frac{R_F}{R_1}
\]

\[
A_{cm-amp} = \frac{1}{\text{CMRR}_o} \frac{R_F}{R_1}
\]  

(3.86)

Therefore, we can find the CMRR of the difference amplifier, which is the same as that of the op-amp:

\[
\text{CMRR}_{amp} = \frac{A_{d-amp}}{A_{cm-amp}} = \text{CMRR}_o
\]  

(3.87)

Since the common-mode signal can be orders of magnitude higher than the differential signal, the common-mode component of Eq. (3.85) can be significant. Therefore, the basic differential amplifier shown in Fig. 3.32 suffers from two disadvantages: a low input resistance and an insufficient common-mode rejection, because \( R_F/R_1 \) has in general a high value.

### 3.5.4 Instrumentation Amplifiers

An instrumentation amplifier is a dedicated differential amplifier with an extremely high input impedance. Its gain can be precisely set by a single resistance. It has a high common-mode rejection capability (i.e., it can reject a signal that is common to both terminals but amplify a differential signal), and this feature is useful for receiving small signals buried in large common-mode offsets or noise. Therefore, instrumentation amplifiers are commonly used as signal conditioners of low-level (often DC) signals in large amounts of noise. The circuit diagram of an instrumentation amplifier is shown in Fig. 3.33. The amplifier consists of two stages. The first stage is the differential stage. Each input signal \( (v_{S1} \text{ or } v_{S2}) \) is applied directly to the noninverting terminal of its op-amp in order to provide the very high input impedance. The second stage is a difference amplifier, which gives a low output impedance and can also allow voltage gain.

The voltage drop between the input terminals of an op-amp is very small, tending to zero: \( v_{d1} = v_{d2} = 0 \). Thus, the voltage drop across the middle resistor \( R_g \) of the potential divider is

\[
v_{rg} = v_{S1} - v_{S2}
\]

which gives the current \( i_{rg} \) through \( R_g \) as

\[
i_{rg} = \frac{v_{rg}}{R_g} = \frac{v_{S1} - v_{S2}}{R_g}
\]
This current flows through all three of the resistors because the currents flowing into the input terminals of the op-amps are practically zero. Therefore, the output voltage of the differential stage becomes

\[ v_{od} = i_g (R_g + 2R) = \frac{v_{S1} - v_{S2}}{R_g} (R_g + 2R) = (v_{S1} - v_{S2}) \left( 1 + \frac{2R}{R_g} \right) \]

Using Eq. (3.82), we can calculate the output voltage \( v_O \) as

\[ v_O = -v_{od} \frac{R_F}{R_1} = - (v_{S1} - v_{S2}) \left( 1 + \frac{2R}{R_g} \right) \left( \frac{R_F}{R_1} \right) \]  
\[ (3.88) \]

which is the output of the instrumentation amplifier. This gain is normally varied by \( R_g \). If the gain variation is not desired, then \( R_g \) can be removed and the differential amplifier can be made with two unity-gain voltage followers. This arrangement is shown in Fig. 3.34 by making \( R_g = \infty \).

**CMRR of an Instrumentation Amplifier**

We can find the overall differential gain of an instrumentation amplifier from Eq. (3.88) as

\[ A_{d-amp} = \left( 1 + \frac{2R}{R_g} \right) \frac{R_F}{R_1} \]  
\[ (3.89) \]

It can easily be shown that the common-mode voltage gain of the first stage is unity. Applying, therefore, the common-mode gain of the second stage from Eq. (3.86), we can find the common-mode gain of the amplifier:

\[ A_{cm-amp} = \frac{1}{CMRR_O} \frac{R_F}{R_1} \]  
\[ (3.90) \]
Thus, the ratio of Eqs. (3.89) and (3.90) gives the CMRR of the instrumentation amplifier:

\[ \text{CMRR}_{\text{amp}} = \left(1 + \frac{2R}{R_g}\right)\text{CMRR}_o \]  

Therefore, the CMRR of the instrumentation amplifier is greater than that of the op-amps by a factor of \(1 + \frac{2R}{R_g}\), which can be large. For example, if we make \(R_F = R_1\), there is a large multiplying factor of \(1 + \frac{2R}{R_g}\) for the differential voltage gain, but not for the common-mode gain.

### 3.5.5 Noninverting Summing Amplifiers

The basic noninverting amplifier in Fig. 3.9 can be operated as a summing amplifier. A noninverting summing amplifier with three inputs is shown in Fig. 3.35. Summing amplifiers are commonly employed in analog computing. By the superposition theorem, the voltage \(v_p\) at the noninverting terminal is

\[ v_p = \frac{R_b\|R_c}{R_a + R_b\|R_c}v_a + \frac{R_a\|R_c}{R_b + R_a\|R_c}v_b + \frac{R_a\|R_b}{R_c + R_a\|R_b}v_c \]

\[ = \frac{R_A}{R_a}v_a + \frac{R_A}{R_b}v_b + \frac{R_A}{R_c}v_c \]  

where \(R_A = (R_a \| R_b \| R_c)\)

Applying Eq. (3.18) for the noninverting amplifier and Eq. (3.92) gives the output voltage:

\[ v_o = \left(1 + \frac{R_F}{R_B}\right)v_p = \left(1 + \frac{R_F}{R_B}\right)\left(\frac{R_A}{R_a}v_a + \frac{R_A}{R_b}v_b + \frac{R_A}{R_c}v_c\right) \]

For \(R_a = R_b = R_c = R\), Eq. (3.93) gives \(R_A = R^3\), and Eq. (3.94) becomes

\[ v_o = \left(1 + \frac{R_F}{R_B}\right)\left(\frac{v_a + v_b + v_c}{3}\right) \]  

\[ \text{FIGURE 3.34 Instrumentation amplifier with fixed gain} \]
Thus, the output voltage is equal to the average of all the input voltages times the closed-loop gain 
\((1 + R_F/R_B)\) of the circuit. If the circuit is operated as a unity follower with 
\(R_F = 0\) and \(R_B = \infty\), the output voltage will equal the average of all the input voltages. That is,

\[
V_O = \frac{V_a + V_b + V_c}{3} \quad (3.96)
\]

If the closed-loop gain \((1 + R_F/R_B)\) is made equal to the number of inputs, the output voltage becomes equal to the sum of all the input voltages. That is, for three inputs, \(n = 3\), and \((1 + R_F/R_B) = n = 3\). Then, Eq. (3.95) becomes

\[
V_O = V_a + V_b + V_c \quad (3.97)
\]

### 3.5.6 Inverting Summing Amplifiers

The basic inverting amplifier in Fig. 3.11 can be operated as an inverting summing amplifier. An inverting summing amplifier with three inputs is shown in Fig. 3.36. Depending on the values of the feedback resistance \(R_F\) and the input resistances \(R_1, R_2,\) and \(R_3\), the circuit can be operated as a summing amplifier, a scaling amplifier, or an averaging amplifier. Since the output voltage is inverted, another inverter may be required, depending on the desired polarity of the output voltage.

The value of \(R_x\) should equal the parallel combination of \(R_1, R_2, R_3,\) and \(R_F\). That is,

\[
R_x = (R_1 \parallel R_2 \parallel R_3 \parallel R_F) \quad (3.98)
\]

For an ideal op-amp, \(v_d = 0\). Using Ohm’s law, we get

\[
i_1 = \frac{v_1}{R_1}, \quad i_2 = \frac{v_2}{R_2}, \quad i_3 = \frac{v_3}{R_3}, \quad i_f = -\frac{V_O}{R_F}
\]

Since the current flowing into the op-amp is zero \((i_i = 0)\),

\[
i_1 + i_2 + i_3 = i_f
\]

or

\[
\frac{v_1}{R_1} + \frac{v_2}{R_2} + \frac{v_3}{R_3} = -\frac{V_O}{R_F} \quad (3.99)
\]
which gives the output voltage as

\[
v_O = -\left(\frac{R_F}{R_1}v_1 + \frac{R_F}{R_2}v_2 + \frac{R_F}{R_3}v_3\right)
\]  
\[ (3.100) \]

Thus, \( v_O \) is a weighted sum of the input voltages, and this circuit is also called a *weighted*, or *scaling*, *summer*. If \( R_1 = R_2 = R_3 = R = R_F \), Eq. (3.100) is reduced to

\[
v_O = -(v_1 + v_2 + v_3)
\]  
\[ (3.101) \]

and the circuit becomes a summing amplifier. If \( R_1 = R_2 = R_3 = nR_F \), where \( n \) is the number of input signals, the circuit operates as an averaging amplifier. For three inputs, \( n = 3 \), and Eq. (3.100) becomes

\[
v_O = -\frac{v_1 + v_2 + v_3}{3}
\]  
\[ (3.102) \]

### 3.5.7 Addition–Subtraction Amplifiers

The functions of noninverting and inverting summing amplifiers can be implemented by only one op-amp, as shown in Fig. 3.37, in order to give output voltage of the form

\[
v_O = A_1v_a + A_2v_b + A_3v_c - B_1v_1 - B_2v_2 - B_3v_3
\]

where \( A_1, A_2, A_3, B_1, B_2, \) and \( B_3 \) are the gain constants. The resistances \( R_x \) and \( R_y \) are included to make the configuration more general. Applying Eqs. (3.94) and (3.100) gives an expression for the resultant output voltage:

\[
v_O = \left(1 + \frac{R_F}{R_B}\right) \left(\frac{R_A}{R_a}v_a + \frac{R_A}{R_b}v_b + \frac{R_A}{R_c}v_c\right) - \left(\frac{R_F}{R_1}v_1 + \frac{R_F}{R_2}v_2 + \frac{R_F}{R_3}v_3\right)
\]  
\[ (3.103) \]

where \( R_A = (R_a || R_b || R_c || R_x) \)  
\[ (3.104) \]

\( R_B = (R_1 || R_2 || R_3 || R_y) \)  
\[ (3.105) \]

To minimize the effects of offset biasing currents on the output of op-amps (discussed further in Sec. 14.3), Thevenin’s equivalent resistance looking from the noninverting terminal is normally made equal to that looking from the inverting terminal. That is,

\[
(R_B || R_F) = R_A
\]  
\[ (3.106) \]
or \[ \frac{R_B R_F}{R_B + R_F} = R_A \] (3.106)

Using this condition, we can simplify the term \((1 + R_F R_B)R_A\):

\[
\left(1 + \frac{R_F}{R_B}\right)R_A = \left(1 + \frac{R_F}{R_B}\right)R_B R_F = R_F
\]

Substituting this relation into Eq. (3.103) yields

\[
v_O = \left(\frac{R_F}{R_a}v_a + \frac{R_F}{R_b}v_b + \frac{R_F}{R_c}v_c\right) - \left(\frac{R_F}{R_1}v_1 + \frac{R_F}{R_2}v_2 + \frac{R_F}{R_3}v_3\right)
\]

which has the general form

\[
v_O = A_1v_a + A_2v_b + A_3v_c - B_1v_1 - B_2v_2 - B_3v_3
\]

Equation (3.107) is valid only if the condition of Eq. (3.106) is satisfied. For known values of gain constants \(A_s\) and \(B_s\), the resistance values can be determined. Difficulty arises, however, in determining values of \(R_x\) and \(R_y\) that meet the criteria of Eq. (3.106). A technique proposed by W. P. Vrbancis [11] can be applied to determine the values of \(R_x\) and \(R_y\). If details and proof of this technique are omitted, the design procedures can be simplified to the following steps:

**Step 1.** Add all the positive coefficients: \(A = A_1 + A_2 + A_3\).

**Step 2.** Add all the negative coefficients: \(B = B_1 + B_2 + B_3\).

**Step 3.** Define a parameter \(C = A - B - 1\).

**Step 4.** Depending on the value of \(C\), determine the values of \(R_x\) and \(R_y\):

a. If \(C > 0\), \(R_x = \infty\) and \(R_y = R_F C\).

b. If \(C < 0\), \(R_x = -R_F C\) and \(R_y = \infty\).

c. If \(C = 0\), \(R_x = \infty\) and \(R_y = \infty\).
Step 5. Choose a suitable value of $R_F$, and find the values of the other components. $R_F$ is normally chosen to meet one of the following constraints:

a. If the equivalent resistance $R_A$ is to be set to a particular value, $R_F$ can be found from the relation $R_F = MR_A$, where $M$ is the largest value of $A$, or $(B + 1)$.

b. If the minimum value of any resistances is to be limited to $R_{\text{min}}$, $R_F$ can be found from the relation $R_F = NR_{\text{min}}$, where $N$ is the largest value of $A_1, A_2, A_3, B_1, B_2, B_3, \text{ or } C$.

(If it is not necessary to meet any of these conditions, we can complete the design by choosing a suitable value of $R_F$.)

Step 6. If the value of any resistor is too high or too low, we can multiply all the resistances by a constant without affecting the output voltage or the condition of Eq. (3.106).

**EXAMPLE 3.15**

**Designing a summing op-amp circuit for a certain resistance $R_A$** Design an inverting and a noninverting summing amplifier of the configuration shown in Fig. 3.37 to give an output voltage of the form

$$v_O = 4v_a + 6v_b + 3v_c - 7v_1 - v_2 - 5v_3$$

The equivalent resistance $R_A$ is to be set to 15 kΩ.

**SOLUTION**

The coefficients are $A_1 = 4, A_2 = 6, A_3 = 3, B_1 = 7, B_2 = 1, \text{ and } B_3 = 5$. Let us follow the design steps just described.

Step 1. $A = 4 + 6 + 3 = 13$.

Step 2. $B = 7 + 1 + 5 = 13$.


Step 4. Since $C < 0$, $R_x = -R_F, \quad C = R_F$ and $R_y = \infty$.

Step 5. The design can be completed by choosing a value of $R_F$. For the given value of $R_A = 15 \text{kΩ}, R_F = MR_A$. In this case, $M = B + 1 = 13 + 1 = 14$. Thus, the values are as follows:

$$R_F = R_y = MR_A = 14 \times 15 = 210 \\text{kΩ}$$

$$R_a = \frac{R_F}{A_1} = \frac{210 \text{kΩ}}{4} = 52.5 \text{kΩ}$$

$$R_b = \frac{R_F}{A_2} = \frac{210 \text{kΩ}}{6} = 35 \text{kΩ}$$

$$R_c = \frac{R_F}{A_3} = \frac{210 \text{kΩ}}{3} = 70 \text{kΩ}$$

$$R_x = \frac{R_F}{C} = 210 \text{kΩ}$$

$$R_1 = \frac{R_F}{B_1} = \frac{210 \text{kΩ}}{7} = 30 \text{kΩ}$$
\[ R_2 = \frac{R_F}{B_2} = \frac{210 \text{ k}\Omega}{1} = 210 \text{ k}\Omega \]

\[ R_3 = \frac{R_F}{B_3} = \frac{210 \text{ k}\Omega}{5} = 42 \text{ k}\Omega \]

\[ R_y = \infty \]

Check:

From Eq. (3.104),

\[ R_A = (52.5 \text{ k}\Omega || 35 \text{ k}\Omega || 70 \text{ k}\Omega || 210 \text{ k}\Omega) = 15 \text{ k}\Omega \]

From Eq. (3.105),

\[ R_B = (30 \text{ k}\Omega || 210 \text{ k}\Omega || 42 \text{ k}\Omega) = 16.15 \text{ k}\Omega \]

From Eq. (3.106),

\[ R_B || R_F = (16.15 \text{ k}\Omega || 210 \text{ k}\Omega) = 15 \text{ k}\Omega \]

Thus, the condition of \( R_A = (R_B || R_F) \) is satisfied.

**EXAMPLE 3.16**

**Designing a summing op-amp circuit for a minimum resistance \( R_{\text{min}} \)** Design an inverting and a noninverting summing amplifier of the configuration shown in Fig. 3.37 to give an output voltage of the form

\[ v_O = 8v_a + 6v_b + 3v_c - 7v_1 - v_2 - 5v_3 \]

The minimum value of any resistance is to be set to \( R_{\text{min}} = 15 \text{ k}\Omega \).

**SOLUTION**

The coefficients are \( A_1 = 8, \ A_2 = 6, \ A_3 = 3, \ B_1 = 7, \ B_2 = 1, \) and \( B_3 = 5 \). Let us follow the design steps described earlier.

**Step 1.** \( A = 8 + 6 + 3 = 17. \)

**Step 2.** \( B = 7 + 1 + 5 = 13. \)

**Step 3.** \( C = A - B - 1 = 17 - 13 - 1 = 3. \)

**Step 4.** Since \( C > 0, R_x = \infty \) and \( R_y = \frac{R_F}{C} \). 

**Step 5.** The design can be completed by choosing a value of \( R_F \). For the given value of \( R_{\text{min}} = 15 \text{ k}\Omega, \) \( R_F = NR_{\text{min}}, \) where \( N \) is the largest value of \( A_1, A_2, A_3, B_1, B_2, B_3, \) or \( C \). In this case, \( N = 8 \). Thus, the values are as follows:

\[ R_F = NR_{\text{min}} = 8 \times 15 \text{ k}\Omega = 120 \text{ k}\Omega \]

\[ R_A = \frac{R_F}{A_1} = \frac{120 \text{ k}\Omega}{8} = 15 \text{ k}\Omega \]

\[ R_B = \frac{R_F}{A_2} = \frac{120 \text{ k}\Omega}{6} = 20 \text{ k}\Omega \]
3.5.8 Optocoupler Drivers

Optocouplers, also known as optical isolators, are generally used to transfer electrical signals from one part of a system to another without direct electrical connection. They find many applications in instrumentation for electrical power engineering, where direct electrical connections between low-level signals and high-current power lines must be avoided, and in medical electronics, where direct connections between patients and electrical power systems must be avoided.

An optocoupler consists of a light-emitting diode (LED), which emits light when forward current is applied, and a photodiode, which converts light to electrical current proportional to the incident light. The light power produced by an LED is directly proportional to the current through the diode. However, the output power is a nonlinear function of the diode voltage. Therefore, an optocoupler is supplied by a current source.

An optocoupler drive circuit is shown in Fig. 3.38. This circuit is a modification of the inverting op-amp shown in Fig. 3.11. Since the current flowing through the op-amp is very small, tending to zero, \( i_S = i_f \). Thus, the voltage across \( R_2 \) is

\[
v_O = -R_F i_f = -R_F i_S
\]

The load current \( i_O \) is given by

\[
i_O = i_f - i_1 = i_S - \frac{v_O}{R_2} = i_S + \frac{R_F i_S}{R_2} = \left( 1 + \frac{R_F}{R_2} \right) i_S
\]  (3.108)
Therefore, the circuit operates as a current amplifier. The LED acting as the load does not determine the load current $i_O$. Only the multiplier $R_F/R_2$ determines the load current. Substituting $i_S = v_S/R_1$ gives the output current as a function of the input voltage. That is,

$$i_O = \left(1 + \frac{R_F}{R_2}\right)\left(\frac{1}{R_1}\right)v_S$$

(3.109)

The circuit then operates as a transconductance amplifier (or voltage–current converter).

### 3.5.9 Photodetectors

A photodiode produces a current that is a linear function of the light intensity; this current is normally measured as incident optical power density $D_P$. The ratio of the output current to the incident optical power density is called the current responsivity. This current can be measured by an inverting op-amp of the type shown in Fig. 3.11, which is a current–voltage converter. The output voltage depends on the input current. From Eq. (3.37) for $v_d = 0$, we get

$$v_O = -R_F i_f = -R_F i_S$$

A simple light-sensing circuit consisting of a photodiode and an inverting op-amp is shown in Fig. 3.39. The anode terminal of the diode can be connected to either the ground or a negative voltage. However, a reverse-biasing voltage will reduce the diode junction capacitance, which in turn decreases the frequency (or transient) response time of the circuit.
3.5.10 Voltage–Current Converters

If the input signal is a voltage source and it is transmitted to a remote load, the load current will depend on the series resistance between the input signal and the load. Even a small drop across the series resistance could significantly change the percentage error of the load voltage. Any changes in the load resistance due to wear and tear or temperature will contribute to the error. The simplest type of voltage–current converter, shown in Fig. 3.40(a), is a modification of the basic noninverting amplifier shown in Fig. 3.9(a). The current through the resistor $R_1$ is given by

$$i_O = i_1 = \frac{v_S - v_d}{R_1} = \frac{v_S}{R_1}$$

Thus, the output current $i_O$ through the load resistance $R$ depends only on $v_S$ and $R_1$, not on $R$. For a fixed value of $R_1$, $i_O$ is directly proportional to $v_S$. Note that none of the load terminals in Fig. 3.40(a) is connected to the ground. That is, the load is floating. The advantage of this arrangement is that no common-mode signal (i.e., noise) will appear across the load.

Op-amps are primarily voltage amplifiers; their current-carrying capability is very limited. Many applications (such as indicators and actuators) require regulated variable current, which is beyond the op-amp’s capability. The circuit shown in Fig. 3.40(b) can provide the load current $i_L$ proportional to the input voltage $v_S$. The output of the op-amp forces the base current through transistor $Q_1$, resulting in a proportional collector current through $Q_1$, the load $R_L$, and $R_1$. The load current $i_L$ can be controlled by varying either the input voltage or the value of $R_1$. The value of the base resistance $R$ must be sufficiently large to protect the base–emitter junction of $Q_1$ and to limit the output current of the op-amp. Also, the DC supply voltage $V_{CC} \geq R_L i_L$ ($= \approx R_L i_L = v_S R_L / R_1$). The load resistance $R_L$ is floating. Thus, the circuit cannot be used with a grounded load.

**EXAMPLE 3.17**

**Designing an op-amp photodetector circuit** Design a photodetector circuit of the form shown in Fig. 3.39 to give an output voltage of $v_O = -200$ mV at an incident power density of $D_P = 500$ nW cm$^2$. The current responsivity of the photodiode is $D_i = 1$ A W, and the active area is $a = 40$ mm$^2$.

**SOLUTION**

The power produced by the photodiode is

$$P = D_P a = (500 \text{ nW cm}^2) \times 40 \text{ mm}^2 = 200 \text{ nW}$$

Therefore, the current produced by the diode is

$$i_S = P D_i = 1 \text{ A W} \times 200 \text{ nW} = 200 \text{ nA}$$

The output voltage is $v_O = -R_F i_S$, which, for $i_S = 200$ nA and $v_O = -200$ mV, gives

$$R_F = -\frac{v_O}{i_S} = \frac{200 \text{ mV}}{200 \text{ nA}} = 1 \text{ M}\Omega$$

**Designing an op-amp photodetector circuit** Design a photodetector circuit of the form shown in Fig. 3.39 to give an output voltage of $v_O = -200$ mV at an incident power density of $D_P = 500$ nW cm$^2$. The current responsivity of the photodiode is $D_i = 1$ A W, and the active area is $a = 40$ mm$^2$.

**SOLUTION**

The power produced by the photodiode is

$$P = D_P a = (500 \text{ nW cm}^2) \times 40 \text{ mm}^2 = 200 \text{ nW}$$

Therefore, the current produced by the diode is

$$i_S = P D_i = 1 \text{ A W} \times 200 \text{ nW} = 200 \text{ nA}$$

The output voltage is $v_O = -R_F i_S$, which, for $i_S = 200$ nA and $v_O = -200$ mV, gives

$$R_F = -\frac{v_O}{i_S} = \frac{200 \text{ mV}}{200 \text{ nA}} = 1 \text{ M}\Omega$$

**3.5.10 Voltage–Current Converters**

If the input signal is a voltage source and it is transmitted to a remote load, the load current will depend on the series resistance between the input signal and the load. Even a small drop across the series resistance could significantly change the percentage error of the load voltage. Any changes in the load resistance due to wear and tear or temperature will contribute to the error. The simplest type of voltage–current converter, shown in Fig. 3.40(a), is a modification of the basic noninverting amplifier shown in Fig. 3.9(a). The current through the resistor $R_1$ is given by

$$i_O = i_1 = \frac{v_S - v_d}{R_1} = \frac{v_S}{R_1}$$

Thus, the output current $i_O$ through the load resistance $R$ depends only on $v_S$ and $R_1$, not on $R$. For a fixed value of $R_1$, $i_O$ is directly proportional to $v_S$. Note that none of the load terminals in Fig. 3.40(a) is connected to the ground. That is, the load is floating. The advantage of this arrangement is that no common-mode signal (i.e., noise) will appear across the load.

Op-amps are primarily voltage amplifiers; their current-carrying capability is very limited. Many applications (such as indicators and actuators) require regulated variable current, which is beyond the op-amp’s capability. The circuit shown in Fig. 3.40(b) can provide the load current $i_L$ proportional to the input voltage $v_S$. The output of the op-amp forces the base current through transistor $Q_1$, resulting in a proportional collector current through $Q_1$, the load $R_L$, and $R_1$. The load current $i_L$ can be controlled by varying either the input voltage or the value of $R_1$. The value of the base resistance $R$ must be sufficiently large to protect the base–emitter junction of $Q_1$ and to limit the output current of the op-amp. Also, the DC supply voltage $V_{CC} \geq R_L i_L$ ($= \approx R_L i_L = v_S R_L / R_1$). The load resistance $R_L$ is floating. Thus, the circuit cannot be used with a grounded load.
3.5.11 DC Voltmeters

The voltage–current converter in Fig. 3.40(a), which consists of a noninverting amplifier, can be used as a DC voltmeter, as shown in Fig. 3.41. Since all signals are DC quantities, we will use uppercase symbols. A moving coil meter with an internal resistance of $R_m$ is connected in the feedback path. For an ideal op-amp, $v_d = 0$; the meter current is given by

$$I_M = I_1 = \frac{V_S}{R_1} - v_d$$

which gives the relation between the input voltage and the meter current as

$$V_S = R_1 I_M$$

Thus, the input voltage $V_S$ can be measured from the deflection of the meter, which is proportional to $I_M$. If the full-scale deflection current of the moving coil is $I_{M(max)} = 100 \mu A$ and $R_1 = 2 \, \text{M}\Omega$, the full-scale reading will be $V_{S(max)} = R_1 I_{M(max)} = 2 \, \text{M}\Omega \times 100 \, \mu A = 200 \, \text{V}$. 
3.5.12 DC Millivoltmeters

The inverting amplifier in Fig. 3.11 can be operated as a DC millivoltmeter, as shown in Fig. 3.42. This circuit is similar to the optocoupler drive in Fig. 3.38, except that the LED is shorted, and we expect similar equations. As before, we will use uppercase symbols for DC quantities. For an ideal amplifier, \( v_d = 0 \) and \( I_i = 0 \). The current through \( R_1 \), which is the same as that through \( R_F \), is

\[
I_S = I_F = \frac{V_S}{R_1} \tag{3.113}
\]

Applying Kirchhoff’s voltage law around the loop formed by op-amp inputs \( R_F \) and \( R_2 \) yields

\[-v_d = R_F I_F + R_2 (I_F - I_M) \quad \text{or} \quad 0 = R_F I_F + R_2 (I_F - I_M)\]

from which we can find the meter current \( I_M \):

\[
I_M = \frac{R_F + R_2}{R_2} I_F = \left(1 + \frac{R_F}{R_2}\right) I_F = \left(1 + \frac{R_F}{R_2}\right) \frac{V_S}{R_1} \tag{3.114}
\]

This equation is the same as Eq. (3.109) for the optocoupler in Fig. 3.38. If \( R_F \gg R_2 \), which is usually the case, Eq. (3.114) can be approximated by

\[
I_M \approx \frac{R_F}{R_1} \left(\frac{1}{R_2}\right) V_S \tag{3.115}
\]

from which we can find the input voltage \( V_S \) in terms of the meter current \( I_M \):

\[
V_S = \frac{R_1 R_2}{R_F} I_M \tag{3.116}
\]

\[
= R_2 I_M \quad \text{for} \quad R_1 = R_F
\]

If \( R_1 = R_F = 150 \, \text{k}\Omega, \, R_2 = 1 \, \text{k}\Omega \), and the full-scale deflection current of the moving coil is \( I_{M(\text{max})} = 100 \, \mu\text{A} \), the full-scale reading will be \( V_{S(\text{max})} = R_2 I_{M(\text{max})} = 1 \, \text{k}\Omega \times 100 \, \mu\text{A} = 100 \, \text{mV} \).
3.5.13 Negative Impedance Converters

Some applications (e.g., oscillators, which we will study in Chapter 13) require the characteristic of negative resistance (or impedance) to compensate for any undesirable resistance (or impedance). The op-amp circuit shown in Fig. 3.43 can be employed to obtain this characteristic. Since the circuit has an impedance $Z$, all voltages and currents will have a magnitude and a phase angle. All quantities are expressed in rms values, and we will use uppercase symbols.

Since $v_d = 0$,

$$V_s = V_x + V_d = V_x$$

Applying Eq. (3.18) for the noninverting amplifier, we get the rms output voltage:

$$V_o = \left(1 + \frac{R_F}{R_1}\right)V_s$$

Since the current drawn by the op-amp is zero, the current $I$ flowing through the impedance $Z$ is the same as the input current $I_s$. That is,

$$I = I_s = \frac{V_s - V_o}{Z} = \frac{1}{Z}V_s - \frac{R_F}{R_1}V_s = -\frac{R_F}{ZR_1}V_s$$

which gives the input impedance $Z_{in}$ as

$$Z_{in} = \frac{V_s}{I_s} = -Z\left(\frac{R_1}{R_F}\right)$$

(3.118)

If $Z$ is replaced by a resistance $R$, then $Z = R$. The circuit will behave as a negative resistance, and Eq. (3.118) becomes

$$Z_{in} = R_{in} = -R\left(\frac{R_1}{R_F}\right)$$

(3.119)

Thus, the ratio $R_1 \ R_F$ acts as a multiplying factor for $R$. If $R_1 = R_F = R$, Eq. (3.119) becomes

$$R_{in} = -R$$

(3.120)

For example, if $R_1 = R_F = R = 10 \text{ k\Omega}$, the circuit in Fig. 3.43 will behave as a resistance of $R_{in} = -10 \text{ k\Omega}$. 
3.5.14 Constant Current Sources

It is often necessary to generate a constant current source from a voltage source. The circuit of Fig. 3.43 can be modified to convert a voltage source to a current source, as shown in Fig. 3.44(a). One side of the load \( Z_L \) is connected to the ground. If \( R_1 = R_F \) and \( Z = R \), the input resistance becomes \( R_{in} = -R \). The circuit inside the shaded area can be replaced by \( R \); the equivalent circuit is shown in Fig. 3.44(b). The voltage source \( V_s \) can be replaced by its Norton equivalent, as shown in Fig. 3.44(c). Since the parallel combination of \( R \) and \( -R \) is infinite, or an open circuit, Fig. 3.44(c) can be reduced to Fig. 3.44(d). The current flowing into load impedance \( Z_L \) is simply

\[
I_L = I_s = \frac{V_s}{R} \tag{3.121}
\]

Thus, the load current \( I_L \) is directly proportional to the input voltage \( V_s \) and is independent of the load impedance \( Z_L \). To simplify the design, we can choose \( R_1 = R_F = R \).

3.5.15 Noninverting Integrators

The integrators in Figs. 3.14(a) and 3.16(a) invert the polarity of the input signal and thus require an additional unity-gain inverter to get a signal of the same polarity. The circuit of Fig. 3.44(a) can operate as a noninverting integrator if the impedance \( Z_L \) is replaced by a capacitor, as shown in Fig. 3.45(a). That is,

\[
R_1 = R_F = R \quad \text{and} \quad Z_L = X_C = \frac{1}{(j\omega C)}
\]

Since \( I_i \approx 0 \), the voltage at the inverting terminal is given by

\[
V_x = \frac{R_1}{R_1 + R_F} V_o = \frac{R}{R + R} V_o = \frac{V_o}{2} \tag{3.122}
\]

The voltage across the capacitor is given by

\[
V_c = I_L Z_L \tag{3.123}
\]
For an ideal op-amp, \( v_d = 0 \). Thus,

\[
V_c = V_x + V_d = V_x
\]

which, after substitution of \( V_x \) from Eq. (3.122) and \( V_c \) from Eq. (3.123), gives

\[
I_L Z_L = \frac{V_o}{2}
\]

or

\[
V_o = 2V_c = 2I_L Z_L \tag{3.124}
\]

Substituting \( I_L \) from Eq. (3.121) into Eq. (3.124), we get

\[
V_o = \frac{2Z_L V_o}{R} = \frac{2V_s}{j\omega CR} \tag{3.125}
\]

which, if converted into the time domain, gives the output voltage as

\[
v_O(t) = \frac{2}{CR} \int v_S(t) \, dt + 2V_{co} \tag{3.126}
\]

where \( V_{co} \) is the initial capacitor voltage at the beginning of integration. The charging of the capacitor can be represented by an equivalent circuit, as shown in Fig. 3.45(b). Thus, the capacitor voltage \( v_C \) can be found directly from Fig. 3.45(b) as follows:

\[
v_C(t) = \frac{1}{C} \int i_S(t) \, dt + V_{co} = \frac{1}{RC} \int v_S \, dt + V_{co} \tag{3.127}
\]

Thus, \( v_O(t) = 2v_x(t) = 2v_C(t) \).

**NOTE** Since one terminal of the capacitor \( C \) is grounded, the capacitor can be charged easily to a desired initial condition at the beginning of integration.

![Figure 3.45 Noninverting integrator](image-url)
### 3.5.16 Inductance Simulators

An op-amp circuit can be used to simulate the characteristic of an inductor. Such an op-amp circuit is shown in Fig. 3.46(a). It consists of two op-amps. The part of the circuit within the shaded area is identical to the negative impedance converter of Fig. 3.43; we can apply Eq. (3.118) to replace it with an equivalent impedance, provided we substitute $Z = R_3, R_1 = R_4, $ and $R_F \equiv Z_C = 1 \ (j\omega C)$. Thus, the equivalent impedance is given by

$$Z_L = \frac{V_1}{I_1} = -R_3 \left( \frac{R_4}{Z_C} \right)$$

If the circuit within the shaded area is replaced by $Z_L$, the resultant circuit also becomes a negative impedance converter, as shown in Fig. 3.46(b). Applying Eq. (3.118) gives the input impedance of the circuit:

$$Z_{in} = \frac{V_s}{I_s} = -R_1 \left( \frac{Z_L}{R_2} \right)$$

Substituting $Z_L$ from Eq. (3.128) into Eq. (3.129) yields

$$Z_{in} = \frac{V_s}{I_s} = -R_1 \left( \frac{1}{R_2} \right) \left( -R_3 \right) \left( \frac{R_4}{Z_C} \right)$$

$$= j\omega C \frac{R_1 R_2 R_4}{R_2} = j\omega L_e$$

where $L_e$ is the effective inductance given by

$$L_e = \frac{R_1 R_2 R_4}{R_2} C$$

Therefore, by choosing the values of $R_1, R_2, R_3, R_4$, and $C$, we can simulate the desired value of inductance $L_e$. 

---

**FIGURE 3.46** Inductance simulator
NOTES

1. From this theoretical development it might appear that inductance simulators could be used in many applications as a replacement for bulky physical inductors. Because of the physical limitations of op-amps, however, inductance simulators suffer from many drawbacks and do not find many practical applications.
2. The op-amp nonlinearities begin limiting the behavior of the inductance simulator at appallingly low frequencies (even less than 20 Hz), and the inductor does not reduce the current at high frequencies as expected.
3. Inductors are commonly used in electrical power applications for storing magnetic energy. A simulated inductor cannot be used to store energy in a magnetic field, so it cannot be used in electrical power circuits (i.e., as a power filter).

EXAMPLE 3.18

Designing an op-amp inductance simulator

Determine the values required for the components in Fig. 3.46(a) in order to simulate an inductor of \( L = 1 \) mH.

SOLUTION

Let \( R_3 = R_4 = 100 \) k\( \Omega \) and \( C = 10 \) pF. From Eq. (3.131), we get

\[
\frac{R_2}{R_1} = \frac{R_3 R_4 C}{L_e} = \frac{100 \times 10^3 \times 100 \times 10^3 \times 10 \times 10^{-12}}{(1 \times 10^{-3})} = 100
\]

If \( R_1 = 5 \) k\( \Omega \), then \( R_2 = 100 \times 5 = 500 \) k\( \Omega \).

\[ \textbf{NOTE:} \text{ To use Eq. (3.131), the designer needs to know the values of five quantities to find the value of } L_e. \text{ The designer has to assume four values, and there is no unique solution to this design problem.} \]

3.5.17 AC-Coupled Bootstrapped Voltage Followers

To minimize the effect of DC input biasing current on the output voltage of op-amps, a resistance \( R_x \) may be connected to the noninverting terminal, as shown in Fig. 3.47(a). This reduces the effective input impedance of the voltage follower to \( R_x \). However, the input impedance can be increased by the circuit, as shown in Fig. 3.47(b); an AC equivalent circuit is shown in Fig. 3.47(c) for higher frequencies at which the capacitors appear as short circuits. The op-amp is operated as a unity follower, which can be represented by an amplifier of approximately unity gain: \( A_v \approx 1 \). \( R_F \) appears to be connected from the input terminal to the output terminal of the amplifier, and its effect on the input impedance is the same as the Miller impedance \( Z_{in} \) connected from the input terminal to the ground. The equivalent circuit is shown in Fig. 3.47(d). From Eq. (2.77), \( Z_{in} \) is given by

\[
Z_{in} = \frac{V_s}{I_s} = \frac{R_F}{1 - A_v} \tag{3.132}
\]
which, for $A_v = 1$, yields $Z_{in} = \infty$. Since the amplifier gain is unity, the output voltage equals the input voltage and there is no voltage drop across $R_F$. Therefore, no current flows through $R_F$, and the input impedance is very high—ideally, infinity. Notice that the voltage at the end of $R_F$ in Fig. 3.47(c) is “pulled up” to the value of the input voltage, thereby offering infinite input impedance. Because of this “bootstrap” characteristic, the circuit is known as a bootstrapped amplifier.

**KEY POINT OF SECTION 3.5**

- The three basic op-amp configurations—inverting, noninverting, and differential—can be applied to perform various signal-processing functions such as integrators, differentiators, inductance simulators, meters, limiters, detectors, comparators, and precision rectifiers.
3.6 Op-Amp Circuit Design

So far, we have designed numerous op-amp circuits. Once the circuit configuration was known, the task was to find the component values. Since the output is dependent mostly on external components, we often must choose some components before a final solution can be found. Generally, in a practical design problem, the circuit diagrams are not known. A designer must decide on the type of configuration, and alternative solutions are possible. In addition, like any other design problem, designing an op-amp circuit requires weighing alternative solutions and comparing complexity and costs. The design sequence can be summarized as follows:

**Step 1.** Study the problem.

**Step 2.** Create a block diagram of the solution.

**Step 3.** Find a hand-analysis circuit-level solution.

**Step 4.** Use PSpice/SPICE for verification.

**Step 5.** Construct the circuit in the lab and take measurements.

### EXAMPLE 3.19

**Designing a proportional controller** A control system requires a proportional controller that will produce \( v_O = 5 \text{ V} \) if the error signal \( v_e = 0 \), \( v_O = 0 \) if \( v_e \leq -0.1 \text{ V} \), and \( v_O = 10 \text{ V} \) if \( v_e \geq 0.1 \text{ V} \). These requirements are graphed in Fig. 3.48. Design a circuit that will implement this control strategy.

**SOLUTION**

**Step 1.** Study the problem. The output voltage is related to the error voltage by

\[
 v_O = 50v_e + 5 = 50(V_{\text{ref}} - v_S) + 5 = 50V_{\text{ref}} - 50v_S + 5
\]

**Step 2.** Create a block diagram of the solution. The problem requires a summing amplifier, as shown in Fig. 3.49(a). Since the signal \( v_S \) is expected to be positive, we also need an inverter.

**Step 3.** Devise a hand-analysis circuit-level solution. The inverting summing amplifier and the circuit implementation are shown in Fig. 3.49(b). Let \( R_1 = R_2 = 10 \text{ k}\Omega \), \( R_F = 50R_1 = 500 \text{ k}\Omega \), and \( R_3 = R_F = 500 \text{ k}\Omega \). Choose \( V_{\text{CC}} = 12 \text{ V} \). Since the maximum output voltage is 10 V, there is no need for a voltage-limiting circuit.

**Step 4.** Use PSpice/SPICE for verification. You are encouraged to plot \( v_O \) against \( v_e \) for \( v_S = 4.6 \text{ V} \) to 5.4 V in increments of 0.01. Invoke DC sweep with the following statement: \( v_{\text{ref}} \) is set to \(-5 \text{ V}\).

\[ .\text{DC VS } 4.6 \text{ 5.4 0.01} \]
An op-amp is a high-gain differential amplifier that can perform various functions in electronic circuits. Op-amps are normally used with a feedback circuit, and the output voltage becomes almost independent of the op-amp parameters. The basic configurations of op-amp amplifiers can be used in many applications such as integrators, differentiators, inductance simulators, meters, limiters, detectors, comparators, and precision rectifiers.

The analysis of an op-amp circuit can be simplified by assuming ideal characteristics. An ideal op-amp has a very high voltage gain, a very high input resistance, a very low output resistance, and a negligible input current. The characteristics of practical op-amps differ from the ideal characteristics, but analyses based on the ideal conditions are valid for many applications and provide the starting point for practical circuit design. Although the DC model of op-amps can be used to analyze complex op-amp circuits, it does not take into account the frequency dependence and op-amp nonlinearities. If the op-amp is operated at frequencies higher than the op-amp break frequency, the effect of frequency dependence should be evaluated.

The op-amp macromodel gives better accuracy. However, the student version of PSpice allows simulation of an amplifier with only one op-amp. If the limit is reached, then the use of the AC model is recommended. The DC model should be the last choice unless the input signal is DC.
References


Review Questions

1. What are the characteristics of an ideal op-amp?
2. What is the minimum number of terminals in an op-amp?
3. What is the typical open-loop voltage gain of an op-amp?
4. What is the typical input resistance of an op-amp?
5. What are the saturation voltages of an op-amp?
6. What is the purpose of supply voltages in an op-amp?
7. What is the PSS of an op-amp?
8. What is the CMRR of an op-amp?
9. What is the typical value of the output resistance of an op-amp?
10. Ideally, what should be the differential voltage gain of an op-amp?
11. Ideally, what should be the common-mode voltage gain of an op-amp?
12. What is the unity-gain bandwidth of an op-amp?
13. What is the effect of rise time on the frequency response of an op-amp?
14. What is the difference between a closed-loop gain and an open-loop gain?
15. What is the virtual ground of an op-amp?
16. What is the integration time constant?
17. What is the frequency response of an integrator?
18. What is the differentiator gain constant?
19. What are the problems of a differentiator?
20. What is the frequency response of a differentiator?
21. What is a voltage follower?
22. What are the advantages of a voltage follower?
23. What is the significance of negative resistance?
24. What is a weighted summing amplifier?

Problems

The symbol \( \boxed{\text{D}} \) indicates that a problem is a design problem. The symbol \( \boxed{\text{P}} \) indicates that you can check the solution to a problem using PSpice/SPICE.

3.2 Characteristics of Ideal Op-Amps

3.1 The op-amp in Fig. 3.3(a) has an open-loop gain of \( A_o = 2 \times 10^5 \). The input resistance is \( R_i = 2 \, \text{M}\Omega \). The DC supply voltages are \( V_{CC} = 15 \, \text{V} \) and \( V_{EE} = -15 \, \text{V} \).
   a. What value of \( v_d \) will saturate the amplifier?
   b. What is the value of op-amp input current \( i_i \)?

3.2 The op-amp shown in Fig. P3.2 is used as a noninverting amplifier. The values are \( A_o = 10^5 \), \( V_{CC} = 12 \, \text{V} \), and \( V_{EE} = -12 \, \text{V} \). If \( v_S = 50 \, \mu\text{V} \), determine the output voltage \( v_O \).

**FIGURE P3.2**

3.3 The op-amp shown in Fig. P3.3 is used as an inverting amplifier. The op-amp parameters are \( A_o = 10^5 \), \( V_{CC} = 12 \, \text{V} \), and \( V_{EE} = -12 \, \text{V} \). If \( v_S = 10 \, \mu\text{V} \), determine the output voltage \( v_O \).

**FIGURE P3.3**

3.4 The op-amp in Fig. 3.3(a) has the following specifications: \( A_o = 2 \times 10^5 \), \( R_i = 2 \, \text{M}\Omega \), \( R_o = 75 \, \Omega \), \( V_{CC} = 15 \, \text{V} \), \( V_{EE} = -15 \, \text{V} \), and maximum output voltage swing = ±14 V. If \( v_+ = 0 \) and \( v_- = 2 \sin 377t \), plot the instantaneous output voltage \( v_O \).

3.5 The op-amp in Fig. 3.3(a) has the following specifications: \( A_o = 2 \times 10^5 \), \( R_i = 2 \, \text{M}\Omega \), \( R_o = 75 \, \Omega \), \( V_{CC} = 15 \, \text{V} \), \( V_{EE} = -15 \, \text{V} \), and maximum output voltage swing = ±14 V. If \( v_+ = 75 \, \mu\text{V} \) and \( v_- = -25 \, \mu\text{V} \), determine the output voltage \( v_O \).
3.6 The input voltages of an op-amp are $v_1 = 100 \mu V$ and $v_2 = 60 \mu V$. The op-amp parameters are $\text{CMRR} = 90 \text{ dB}$ and $A_d = A_o = 2 \times 10^5$. Determine (a) the differential voltage $v_d$, (b) the common-mode voltage $v_c$, (c) the magnitude of the common-mode gain $A_c$, and (d) the output voltage $v_O$.

3.3 **Op-Amp PSpice/SPICE Models**

3.7 Develop PSpice/SPICE subcircuits for the DC model (in Fig. 3.7) and the AC model (in Fig. 3.8) for the LF411 op-amp. The parameters are $R_1 = 10^{12} \Omega$, $R_o = 50 \Omega$, $A_o = 2 \times 10^5$, break frequency $f_b = 20 \text{ Hz}$, and unity-gain bandwidth $f_{bw} = 4 \text{ MHz}$. Assume DC power supply voltages of $\pm 15 \text{ V}$.

3.8 Develop PSpice/SPICE subcircuits for the DC model (in Fig. 3.7) and the AC model (in Fig. 3.8) for the LM324 op-amp. The parameters are $R_1 = 2 \text{ M}\Omega$, $R_2 = 50 \Omega$, $A_o = 2 \times 10^5$, break frequency $f_b = 4 \text{ kHz}$, and unity-gain bandwidth $f_{bw} = 1 \text{ MHz}$. Assume a DC supply voltage of $+15 \text{ V}$.

3.4 **Analysis of Ideal Op-Amp Circuits**

3.9 Design a noninverting amplifier as shown in Fig. 3.9(a) to provide a closed-loop voltage gain of $A_f = 100$. The input voltage is $v_S = 100 \text{ mV}$ with a source resistance of $R_s = 1 \text{ k}\Omega$. Find the value of output voltage $v_O$. The DC supply voltages are given by $V_{CC} = V_{EE} = 15 \text{ V}$. Assume an ideal op-amp.

3.10 With the design values in Prob. 3.9, find the output voltage $v_O$, the input resistance $R_{in} = v_S / i_S$, and the output resistance $R_{out}$ under the following conditions:

   a. $A_o = 25 \times 10^5$, $R_1 = 10^{12} \Omega$, and $R_o = 50 \Omega$.
   b. $A_o = 5 \times 10^5$, $R_1 = 10^{12} \Omega$, and $R_o = 50 \Omega$.
   c. Use PSpice/SPICE to verify your results in parts (a) and (b).

3.11 Design a noninverting amplifier as shown in Fig. 3.9(a) by determining the values of $R_F$ and $R_1$. The closed-loop gain should be $A_f = 10$. The input voltage to the amplifier is $v_S = 500 \text{ mV}$, and it has a source resistance of $200 \Omega$. What is the value of output voltage $v_O$?

3.12 The input voltage to the noninverting amplifier in Fig. 3.9(a) is shown in Fig. P3.12. The source resistance $R_s$ is negligible, $R_F = 20 \text{ k}\Omega$, $R_1 = 5 \text{ k}\Omega$, $V_{CC} = 15 \text{ V}$, and $-V_{EE} = -15 \text{ V}$. Plot the output voltage $v_O$ if $R_F = 20 \text{ k}\Omega$ and $R_1 = 5 \text{ k}\Omega$.

**FIGURE P3.12**

3.13 The noninverting op-amp amplifier in Fig. 3.9(a) has an open-loop gain of $A_o = 5 \times 10^3$, $R_1 = 10 \text{ k}\Omega$, and $R_F = 30 \text{ k}\Omega$. Calculate (a) the closed-loop voltage gain $A_f$, (b) the output voltage $v_O$, and (c) the error in output voltage if $A_o$ is assumed to be infinite.

3.14 The input voltage to the noninverting amplifier in Fig. 3.9(a) is $v_S = 10 \sin(2000\pi t)$. The source resistance $R_s$ is negligible. If $R_F = 20 \text{ k}\Omega$, $R_1 = 5 \text{ k}\Omega$, $V_{CC} = 15 \text{ V}$, and $-V_{EE} = -15 \text{ V}$, plot the output voltage $v_O$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
3.15 A voltage follower is shown in Fig. P3.15. The op-amp parameters are $A_o = 5 \times 10^5$, $R_o = 75 \, \Omega$, and $R_i = 2 \, \text{M}\Omega$. The input voltage is $v_S = 5 \, \text{V}$, and $R_s = 10 \, \text{k}\Omega$. Find the output voltage $v_O$, the input resistance $R_{in} = v_S/i_S$, and the output resistance $R_{out}$.

**FIGURE P3.15**

![Voltage Follower Diagram](image)

3.16 a. A noninverting amplifier has $R_1 = 15 \, \text{k}\Omega$ and $R_F = 50 \, \text{k}\Omega$. The op-amp parameters are $A_o = 2 \times 10^3$, $f_b = 10 \, \text{Hz}$, $R_o = 75 \, \Omega$, and $R_i = 2 \, \text{M}\Omega$. The frequency of the input signal is $f_s = 100 \, \text{kHz}$. Determine the unity-gain bandwidth $f_{bw}$, the closed-loop voltage gain $A_f$, and the closed-loop break frequency $f_c$ of the op-amp circuit.

b. Use PSpice/SPICE to plot the closed-loop frequency response of the voltage gain. Assume $v_s = 0.1 \, \text{V}$ (AC), and use the linear AC model.

3.17 Repeat Prob. 3.16 for $R_1 = R_F = 15 \, \text{k}\Omega$.

3.18 Two noninverting op-amps are cascaded as shown in Fig. P3.18. The unity-gain bandwidth of the op-amps is $f_a = 1 \, \text{MHz}$, and the slew rate is $SR = 6 \, \text{V}/\mu\text{s}$.

a. If $R_1 = 20 \, \text{k}\Omega$, $R_2 = 200 \, \text{k}\Omega$, $R_3 = 180 \, \text{k}\Omega$, $R_4 = 50 \, \text{k}\Omega$, $R_5 = 500 \, \text{k}\Omega$, and $R_6 = 45 \, \text{k}\Omega$, determine the voltage gain $A_f = v_O/v_1$.

b. Determine the maximum frequency $f_{max}$ of the input signal $v_1$ if the amplitude of the output voltage is limited to $10 \, \text{V}$.

c. Use PSpice to verify your results.

**FIGURE P3.18**

![Cascaded Noninverting Amplifiers Diagram](image)

3.19 A transducer produces a voltage signal of $v_S = 50 \, \text{mV}$ and has an internal resistance of $R_s = 5 \, \text{k}\Omega$. Design the inverting op-amp amplifier of Fig. 3.11 by determining the values of $R_1$, $R_F$, and $R_x$. The output voltage should be $v_O = -5 \, \text{V}$. The current drawn from the transducer should not be more than $20 \, \mu\text{A}$. Assume an ideal op-amp and $V_{CC} = V_{EE} = 12 \, \text{V}$.
3.20 With the design values in Prob. 3.19, find the value of output voltage $v_O$, the input resistance $R_{in} = v_S \cdot i_S$, and the output resistance $R_{out}$ under the following conditions:

a. $A_o = 25 \times 10^3$, $R_1 = 10^{12}$ $\Omega$, and $R_o = 50$ $\Omega$.

b. $A_o = 5 \times 10^3$, $R_1 = 10^{12}$ $\Omega$, and $R_o = 50$ $\Omega$.

c. Use PSpice/SPICE to verify your results in parts (a) and (b).

3.21 The inverting amplifier in Fig. 3.11 has $R_1 = 5$ $k\Omega$, $R_F = \infty$, $R_x = 5$ $k\Omega$, $V_{CC} = 15$ $V$, $V_{EE} = -15$ $V$, and maximum output voltage swing $= \pm 14$ $V$. If $v_S = 200$ $mV$, determine the output voltage $v_O$.

3.22 The inverting amplifier in Fig. 3.11 has $R_1 = 10$ $k\Omega$, $R_F = 50$ $k\Omega$, and $R_x = 8.33$ $k\Omega$. The op-amp has an open-loop voltage gain of $A_o = 2 \times 10^5$. The input voltage is $v_S = 100$ $mV$. Calculate (a) the closed-loop gain $A_f$, (b) the output voltage $v_O$, and (c) the error in output voltage if the open-loop gain $A_o$ is assumed to be infinite.

3.23 a. An inverting amplifier has $R_1 = 15$ $k\Omega$ and $R_F = 50$ $k\Omega$. The op-amp parameters are $A_o = 2 \times 10^5$, $f_b = 10$ $Hz$, $R_o = 75$ $\Omega$, and $R_1 = 2$ $M\Omega$. The frequency of the input signal is $f_s = 100$ $kHz$. Determine the unity-gain bandwidth $f_{bw}$, the closed-loop voltage gain $A_f$, and the closed-loop break frequency $f_c$ of the op-amp circuit.

b. Use PSpice/SPICE to plot the closed-loop frequency response of the voltage gain. Assume $v_s = 0.1$ $V$ (AC), and use the linear AC model.

3.24 Repeat Prob. 3.23 for $R_1 = R_F = 15$ $k\Omega$.

3.25 The inverting amplifier shown in Fig. P3.25 has $R_1 = 50$ $k\Omega$, and $R_2 = R_3 = 20$ $k\Omega$. The unity-gain bandwidth of the op-amps is $f_u = 1$ $MHz$, and the slew rate is $SR = 6$ $V/\mu s$.

a. Determine the value of $R_4$ that will give a voltage gain of $A_f = v_O/v_1 = -10$ $V/V$.

b. Determine the maximum frequency $f_{max}$ of the input signal $v_1$ if the amplitude of the output voltage is limited to $10$ $V$.

c. Use PSpice to verify your results.

3.26 Two inverting op-amps are cascaded as shown in Fig. P3.26. The unity-gain bandwidth of the op-amps is $f_u = 1$ $MHz$, and the slew rate is $SR = 6$ $V/\mu s$.

a. If $R_1 = 20$ $k\Omega$, $R_2 = 100$ $k\Omega$, $R_3 = 150$ $k\Omega$, $R_4 = 20$ $k\Omega$, and $R_5 = 160$ $k\Omega$, determine the voltage gain $A_f = v_O/v_1$.

b. Use PSpice to verify your results.
3.27 Two transducers produce voltage signals of \(v_b = 200 \text{ mV}\) and \(v_a = 220 \text{ mV}\). Design a differential amplifier as shown in Fig. 3.32 to produce an output voltage \(|v_o| = 5 \text{ V}|. Assume an ideal op-amp and \(V_{CC} = V_{EE} = 12 \text{ V}.\)

3.28 With the design values in Prob. 3.27, find the value of output voltage \(v_o\) under the following conditions of the op-amp:

a. \(A_o = 25 \times 10^3\) and \(R_i = 10^{12} \Omega\).

b. \(A_o = 5 \times 10^5\) and \(R_i = 10^{12} \Omega\).

c. Use PSpice/SPICE to verify your results in parts (a) and (b).

3.29 a. Design a differential amplifier as shown in Fig. 3.32 to give a differential voltage gain of \(|A_f| = 200\). The input voltages are \(v_b = 70 \text{ mV}\) and \(v_a = 50 \text{ mV}\). Assume an ideal op-amp and \(V_{CC} = V_{EE} = 12 \text{ V}.\)

b. Calculate the error in output voltage if the open-loop gain is \(A_o = 5 \times 10^5\).

3.30 The values of the differential amplifier in Fig. 3.32 are \(A_o = 5 \times 10^5\), \(R_1 = 5 \text{ k}\Omega\), \(R_F = 50 \text{ k}\Omega\), \(R_6 = 2 \text{ k}\Omega\), and \(R_2 = 20 \text{ k}\Omega\). The input voltages are \(v_b = 5 \text{ mV}\) and \(v_a = -15 \text{ mV}\). Find the output voltage \(v_O\).

3.31 The differential amplifier shown in Fig. P3.31 has \(R_1 = 100 \text{ k}\Omega\), \(R_2 = 150 \text{ k}\Omega\), \(R_3 = 50 \text{ k}\Omega\), and \(R_4 = R_5 = R_6 = R_7 = 50 \text{ k}\Omega\). The unity-gain bandwidth of the op-amps is \(f_u = 1 \text{ MHz}\), and the slew rate is \(SR = 6 \text{ V}/\mu\text{s}\).

a. Determine the output voltage in terms of \(v_1\) and \(v_2\).

b. Use PSpice to verify your results.

3.32 The differential amplifier shown in Fig. P3.32 has \(R_1 = 40 \text{ k}\Omega\), \(R_2 = 80 \text{ k}\Omega\), \(R_3 = 50 \text{ k}\Omega\), \(R_4 = 100 \text{ k}\Omega\), and \(R_F = 500 \text{ k}\Omega\). The unity-gain bandwidth of the op-amps is \(f_u = 1 \text{ MHz}\), and the slew rate is \(SR = 6 \text{ V}/\mu\text{s}\).

a. Determine the voltage gain \(A_f = v_O/v_i\).

b. Use PSpice to verify your results.
Two noninverting op-amps can be cascaded to produce a differential voltage as shown in Fig. P3.33 such that \( v_O = a_1 v_1 - b_1 v_2 \). The unity-gain bandwidth of the op-amps is \( f_u = 1 \text{ MHz} \), and the slew rate is \( \text{SR} = 6 \text{ V}/\mu\text{s} \).

a. If \( R_1 = R_3 = 10 \text{ k}\Omega, R_2 = R_4 = 500 \text{ k}\Omega, \) and \( R_3 = R_6 = 9.8 \text{ k}\Omega \), determine the differential voltage gains \( a_1 \) and \( a_2 \).

b. Using the values in part (a), determine the maximum frequency \( f_{\text{max}} \) of the input signal \( v_1 \) if the amplitude of the output voltage due to \( v_1 \) is limited to 10 V.

c. Use PSpice to verify your results.

Two noninverting op-amps can be cascaded to produce a differential voltage as shown in Fig. P3.34 such that \( v_O = a_1 v_1 - b_1 v_2 \). The unity-gain bandwidth of the op-amps is \( f_u = 1 \text{ MHz} \), and the slew rate is \( \text{SR} = 6 \text{ V}/\mu\text{s} \).

a. If \( R_1 = R_3 = 10 \text{ k}\Omega, R_2 = R_4 = 500 \text{ k}\Omega, \) and \( R_3 = R_6 = 9.8 \text{ k}\Omega \), determine the differential voltage gains \( a_1 \) and \( a_2 \).

b. Using the values in part (a), determine the maximum frequency \( f_{\text{max}} \) of the input signal \( v_1 \) if the amplitude of the output voltage due to \( v_1 \) is limited to 10 V.

c. Use PSpice to verify your results.
3.35 The amplifier shown in Fig. P3.35 can produce a differential voltage such that $v_O = a_1v_1 - a_2v_2$. The unity-gain bandwidth of the op-amps is $f_u = 1 \text{ MHz}$, and the slew rate is $SR = 6 \text{ V}/\mu\text{s}$.

a. If $R_1 = R_3 = 10 \text{ k}\Omega$ and $R_2 = R_4 = 1 \text{ M}\Omega$, determine the differential voltage gains $a_1$ and $a_2$.
b. Using the values in part (a), determine the maximum frequency $f_{\text{max}}$ of the input signal $v_1$ if the amplitude of the output voltage due to $v_1$ is limited to 10 V.
c. Use PSpice to verify your results.

**FIGURE P3.35**

3.36 The amplifier shown in Fig. P3.36 can produce a differential voltage such that $v_O = a_1v_1 - a_2v_2$. The unity-gain bandwidth of the op-amps is $f_u = 1 \text{ MHz}$, and the slew rate is $SR = 6 \text{ V}/\mu\text{s}$. Derive an expression of the output voltage if $R_2 = R_3$, $R_1 = R_4$, and $R_F = R_5$.

**FIGURE P3.36**

3.5 Op-Amp Applications

3.37 The integrator in Fig. 3.16 has $V_{CC} = 15 \text{ V}$, $-V_{EE} = -15 \text{ V}$, maximum voltage swing $= \pm 14 \text{ V}$, $C_F = 0.01 \mu\text{F}$, $R_1 = 1 \text{ k}\Omega$, and $R_F = 1 \text{ M}\Omega$. The initial capacitor voltage is $V_{co} = 0$. Draw the waveform for the output voltage if the input voltage is described by

$$v_S = \begin{cases} 1 \text{ V} & \text{for } 0 \leq t < 1 \text{ ms} \\ -1 \text{ V} & \text{for } 1 \leq t < 2 \text{ ms} \\ 1 \text{ V} & \text{for } 2 \leq t < 3 \text{ ms} \\ -1 \text{ V} & \text{for } 3 \leq t < 4 \text{ ms} \end{cases}$$
3.38 The integrator in Fig. 3.16 has $V_{CC} = 15 \, \text{V}$, $-V_{EE} = -15 \, \text{V}$, maximum voltage swing $= \pm 14 \, \text{V}$, $C_F = 0.1 \, \mu\text{F}$, $R_1 = 10 \, \text{k}\Omega$, and $R_F = 1 \, \text{M}\Omega$. The initial capacitor voltage is $V_{c0} = 0$. Draw the waveform of the output voltage for the input voltage shown in Fig. P3.38.

![FIGURE P3.38](image)

3.39 The integrator in Fig. 3.16 has $C_F = 0.01 \, \mu\text{F}$, $R_1 = 10 \, \text{k}\Omega$, and $R_F = 1 \, \text{M}\Omega$. The open-loop voltage gain of the op-amp is $A_o = 5 \times 10^5$. Use Miller’s theorem to find the 3-dB frequency of the integrator.

3.40 Design an integrator as shown in Fig. 3.16 to be operated with an AC signal of 5 kHz and to give a closed-loop voltage gain of $A_f = 10$ at $\omega = 1 \, \text{rad/s}$.

3.41 a. Design a differentiator as shown in Fig. 3.27(a) to satisfy the following specifications: maximum voltage gain of $A_{f_{(\text{max})}} = 20$ and gain-limiting frequency $f_b = 10 \, \text{kHz}$. Determine the values of $R_1$, $R_F$, and $C_1$.

b. Use PSpice/SPICE to check your results by plotting the frequency response in part (a).

3.42 The differentiator in Fig. 3.27(a) has $R_1 = 2 \, \text{k}\Omega$, $R_F = 10 \, \text{k}\Omega$, and $C_1 = 0.01 \, \mu\text{F}$. Determine (a) the differentiator time constant $\tau_d$, (b) the gain-limiting frequency $f_b$, and (c) the maximum closed-loop voltage gain $A_{f_{(\text{max})}}$.

3.43 Design an instrumentation amplifier as shown in Fig. 3.33 to give a differential voltage gain $A_f$ between 500 and 1000.

3.44 Design an instrumentation amplifier as shown in Fig. 3.34 to give a fixed differential voltage gain of $A_f = 750$.

3.45 The noninverting summing amplifier in Fig. 3.35 has $R_a = R_b = R_c = 20 \, \text{k}\Omega$, $R_F = 40 \, \text{k}\Omega$, $R_B = 20 \, \text{k}\Omega$, $v_a = 2 \, \text{V}$, $v_b = -3 \, \text{V}$, $v_c = -2 \, \text{V}$, $V_{CC} = 15 \, \text{V}$, $-V_{EE} = -15 \, \text{V}$, and maximum voltage swing $= \pm 14 \, \text{V}$. Determine the output voltage $V_O$.

3.46 The inverting summing amplifier in Fig. 3.36 has $R_1 = R_2 = R_3 = 20 \, \text{k}\Omega$, $R_F = 40 \, \text{k}\Omega$, $R_x = 5.71 \, \text{k}\Omega$, $v_1 = 2 \, \text{V}$, $v_2 = -3 \, \text{V}$, $v_3 = -2 \, \text{V}$, $V_{CC} = 15 \, \text{V}$, $-V_{EE} = -15 \, \text{V}$, and maximum voltage swing $= \pm 14 \, \text{V}$. Determine the output voltage $V_O$.

3.47 Design an add–subtract summing amplifier as shown in Fig. 3.37 to give an output voltage of the form $v_0 = 5v_a + 7v_b + 3v_c - 2v_1 - v_2 - 6v_3$. The equivalent resistance $R_A$ should be set to 20 k\Omega.

3.48 Design an add–subtract summing amplifier as shown in Fig. 3.37 to give an output voltage of the form $v_0 = 5v_a + 9v_b + 3v_c - 8v_1 - 2v_2 - 6v_3$. The minimum value of any resistance should be $R_{\text{min}} = 20 \, \text{k}\Omega$.

3.49 Design an optocoupler drive circuit as shown in Fig. 3.38 to produce a drive current of 500 mA from a signal voltage of 10 m\text{V}.
3.50 Design a photodetector circuit as shown in Fig. 3.39 to give an output voltage of 1 V at an incident power density of $D_P = 1 \mu W/cm^2$. The current responsivity of the photodiode is $D_i = 1 A/W$, and the active area is $a = 40 \text{ mm}^2$.

3.51 The voltage-to-current converter circuit in Fig. 3.40(a) has $R_1 = R = 10 \text{ k}\Omega$ and $v_S = 200 \text{ mV}$. Determine the load current $i_O$.

3.52 The full-scale current of the moving coil for the DC voltmeter in Fig. 3.41 is $I_M = 200 \mu A$. Determine the value of $R_1$ to give a full-scale reading of $V_S = 300 \text{ V}$.

3.53 Design a DC millivoltmeter as shown in Fig. 3.42. The full-scale current of the moving coil is $I_M = 0.5 \mu A$. Determine the values of $R_1$, $R_F$, and $R_2$ to give a full-scale voltage reading of $V_S = 200 \text{ mV}$.

3.54 Design a negative impedance converter as shown in Fig. 3.43 by determining the component values such that the input resistance will be $Z_{in} = R_{in} = 15 \text{ k}\Omega$.

3.55 a. The noninverting integrator in Fig. 3.45(a) has $V_{CC} = 15 \text{ V}$, $V_{EE} = -15 \text{ V}$, maximum voltage swing $= \pm 14 \text{ V}$, $C = 0.01 \mu F$, and $R_1 = R_F = R = 1 \text{ M}\Omega$. The initial capacitor voltage is $V_{co} = 0$. Draw the waveform for the output voltage if the input is a step voltage described by

$$v_S = 1 \text{ V} \quad \text{for} \quad t \geq 0$$

b. Use PSpice/SPICE to plot the output voltage in part (a).

3.56 Design an inductance simulator as shown in Fig. 3.46 by determining the values of components. The inductance should be $L_e = 2 \text{ mH}$.

3.6 Op-Amp Circuit Design

3.57 A control system requires a proportional controller that will produce $v_O = 5 \text{ V}$ if the error signal $v_e = 0$, $v_O = 10 \text{ V}$ if $v_e \leq -0.1 \text{ V}$, and $v_O = 0$ if $v_e = 0.1 \text{ V}$. These requirements are graphed in Fig. P3.57. Design a circuit that will implement this control strategy to produce $v_O$ from $v_S$ and $v_{ref}$.

3.58 A triggering circuit requires short pulses $v_O$ of approximately 10 V magnitude and pulse width of $t_w = 200 \mu s$, as shown in Fig. P3.58. Design a circuit that will generate triggering pulses. (There is no unique solution.)
3.59 A control system requires a proportional and integral controller that will produce \( v_O = 5 \) V if the signal \( v_{e1} = 0 \), \( v_O = 10 \) V if \( v_{e1} \leq -0.1 \) V, and \( v_O = 0 \) if \( v_{e1} \geq 0.1 \) V, as shown in Fig. P3.59. Design a circuit that will implement this control strategy to produce \( v_O \) from the reference signal \( v_S \) and the feedback signal \( v_{ref} \).

![FIGURE P3.59](image)

3.60 The inverting amplifier shown in Fig. P3.60 can give high voltage gain and requires a narrow range of resistor values. The output voltage should be \( v_O = 12 \) V for \( v_S \leq -0.05 \) V and \( v_O = -12 \) V if \( v_S \geq 0.05 \) V. Design a circuit that will implement this control strategy.

![FIGURE P3.60](image)

3.61 Design an op-amp circuit to obtain a voltage gain of \( A_f = 100 \) V/V with an input resistance of \( R_i \approx 25 \) kΩ. The peak-to-peak output voltage swing should be limited to \( \pm 11 \) V at 25 kHz. Assume DC supply voltages of \( \pm 12 \) V. Use PSpice to verify your design by plotting the frequency response and the transient response with an input signal of 1 mV at 25 kHz.

3.62 Design an op-amp differential amplifier circuit to obtain a voltage gain of \( A_f = 5 \) kV/V with an input resistance of \( R_i \approx 500 \) kΩ. The peak-to-peak output voltage swing should be limited to \( \pm 11 \) V at 25 kHz. Assume DC supply voltages of \( \pm 12 \) V. Use PSpice to verify your design by plotting the frequency response and the transient response with a differential voltage of 1 mV at 25 kHz.

For Probs. 3.63 to 3.66, the op-amp has \( C_i = 1.5 \) pF, \( R_i = 2 \) MΩ, \( R_o = 75 \) Ω, and open-loop voltage gain \( A_o = 2 \times 10^6 \). Use PSpice/SPICE to check your design by plotting the frequency response.

3.63 Design an integrator as shown in Fig. 3.21(a) to give a DC voltage gain \( |A_{PB}| = 20 \) and a high 3-dB frequency \( f_H = 1 \) kHz. Assume \( R_i = 1 \) kΩ and \( R_L = 20 \) kΩ.

3.64 Design a differentiator circuit as shown in Fig. 3.30(a) to give \( f_L = 5 \) kHz and \( f_H = 10 \) kHz. The pass-band gain is \( |A_{PB}| = 20 \).
3.65 An amplifier circuit is shown in Fig. P3.65. Use the short-circuit and zero-value methods to find the low 3-dB frequency $f_L$, the high 3-dB frequency $f_H$, and the pass-band gain $A_{PB}$.

**FIGURE P3.65**

![Amplifier Circuit Diagram](image)

3.66 An amplifier circuit is shown in Fig. P3.66. Use the short-circuit and zero-value methods to find the low 3-dB frequency $f_L$, the high 3-dB frequency $f_H$, and the pass-band gain $A_{PB}$.

**FIGURE P3.66**

![Amplifier Circuit Diagram](image)
CHAPTER 4

SEMICONDUCTOR DIODES

Learning Outcomes
After completing this chapter, students should be able to do the following:

- Describe the ideal and practical characteristics of semiconductor diodes.
- Determine the circuit models of a diode and apply them for analyzing diode circuits.
- Determine the DC and small-signal performances of simple diode circuits.
- Explain the characteristics of zener diodes and their applications as voltage regulators.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$i_D$, $i_d$, $I_D$</td>
<td>Instantaneous DC and AC and quiescent DC diode currents</td>
</tr>
<tr>
<td>$v_D(t)$, $v_o(t)$</td>
<td>Instantaneous DC and AC output voltages</td>
</tr>
<tr>
<td>$V_{o(\text{av})}$, $V_{o(\text{rms})}$</td>
<td>Average and rms output voltages</td>
</tr>
<tr>
<td>$V_{f(pp)}$, $V_{f(p)}$</td>
<td>Peak-to-peak and peak ripple output voltages</td>
</tr>
<tr>
<td>$v_D$, $v_d$, $V_D$</td>
<td>Instantaneous DC and AC and quiescent DC diode voltages</td>
</tr>
<tr>
<td>$V_Z$, $I_Z$</td>
<td>DC zener voltage and current</td>
</tr>
<tr>
<td>$V_{ZK}$, $I_{ZK}$</td>
<td>DC zener knee voltage and current</td>
</tr>
<tr>
<td>$v_Z(t)$, $i_Z(t)$</td>
<td>Instantaneous zener voltage and current</td>
</tr>
</tbody>
</table>
Symbol | Meaning
--- | ---
$v_L(t)$, $i_L(t)$ | Instantaneous zener voltage and current
$r_d$, $R_D$ | Small-signal AC and DC diode resistances
$V_{ZO}$, $R_Z$ | Threshold zener voltage and small-signal zener resistance
$T_J$, $T_A$ | Junction and ambient temperature

# 4.1 Introduction

A diode is a two-terminal semiconductor device. It offers a low resistance on the order of milliohms in one direction and a high resistance on the order of gigaohms in the other direction. Thus a diode permits an easy current flow in only one direction. A diode is the simplest electronic device, and it is the basic building block for many electronic circuits and systems. In this chapter, we will discuss the characteristics of diodes and their models through analysis of a diode circuit.

A diode exhibits a nonlinear relation between the voltage across its terminals and the current through it. However, analysis of a diode can be greatly simplified with the assumption of an ideal characteristic. The results of this simplified analysis are useful in understanding the operation of diode circuits and are acceptable in many practical cases, especially at the initial stage of design and analysis. If more accurate results are required, linear circuit models representing the nonlinear characteristic of diodes can be used. These models are commonly used in evaluating the performance of diode circuits. If better accuracy is required, however, computer-aided modeling and simulation are normally used.

# 4.2 Ideal Diodes

The symbol for a semiconductor diode is shown in Fig. 4.1(a). Its two terminals are the anode and the cathode. If the anode voltage is held positive with respect to the cathode terminal, the diode conducts and offers a small forward resistance. The diode is then said to be forward biased, and it behaves as a short circuit, as shown in Fig. 4.1(b). If the anode voltage is kept negative with respect to the cathode terminal, the diode offers a high resistance. The diode is then said to be reverse biased, and it behaves as an open circuit, as shown in Fig. 4.1(c). Thus, an ideal diode will offer zero resistance and zero voltage drop in the forward direction. In the reverse direction, it will offer infinite resistance and allow zero current.

An ideal diode behaves as a short circuit in the forward region of conduction ($v_D = 0$) and as an open circuit in the reverse region of nonconduction ($i_D = 0$). The $v$-$i$ characteristic of an ideal diode is shown in Fig. 4.1(d). Because the forward voltage tends to be greater than zero, the forward current through the diode tends to be infinite. In practice, however, a diode is connected to other circuit elements, such

![FIGURE 4.1 Characteristics of an ideal diode](image-url)
as resistances, and its forward current is limited to a known value, which will depend on the values of the circuit elements.

**EXAMPLE 4.1**

**Application as a diode OR logic function**  A diode circuit that can generate an OR logic function is shown in Fig. 4.2. A positive logic convention denotes logic 0 for 0 V and logic 1 for a positive voltage, typically 5 V. Show the truth table that illustrates the logic output.

**SOLUTION**

If both inputs have 0 V (i.e., $V_A = 0$ and $V_B = 0$), both diodes will be off, and the output $V_C$ will be 0 (or logic 0) only. If either $V_A$ or $V_B$ (or both) is high ($+5$ V), the corresponding diode ($D_1$ or $D_2$ or both) will conduct, and the output voltage will be high at $V_C = 5$ V. As we will see later, a real diode has a finite voltage drop of approximately 0.7 V, and the output voltage will be approximately $5 - 0.7 = 4.3$ V (or logic 1). The truth table that illustrates the logic functions is shown in Table 4.1. We can define the logic level at any desired value. That is, for example, we can say greater than 3 V for logic 1, and less than 1 V for logic 0.

![Diode OR logic circuit](image)

**TABLE 4.1**  Truth table for Example 4.1

<table>
<thead>
<tr>
<th>$V_A$ (V)</th>
<th>$V_B$ (V)</th>
<th>$V_C$ (V)</th>
<th>Logic Levels</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0 0 0</td>
</tr>
<tr>
<td>0</td>
<td>5</td>
<td>4.3</td>
<td>0 1 1</td>
</tr>
<tr>
<td>5</td>
<td>0</td>
<td>4.3</td>
<td>1 0 1</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>4.3</td>
<td>1 1 1</td>
</tr>
</tbody>
</table>

**EXAMPLE 4.2**

**Application as a diode AND logic function**  A diode circuit that can generate an AND logic function is shown in Fig. 4.3. A positive–logic convention denotes logic 0 for 0 V and logic 1 for a positive voltage, typically 5 V. Show the truth table that illustrates the logic output.

**SOLUTION**

If input $V_A$ or $V_B$ (or both) is 0, the corresponding diode ($D_1$ or $D_2$ or both) will conduct, and the output voltage will be 0. In practice, a diode has a finite voltage drop of approximately 0.7 V, and the output voltage will be approximately 0.7 V (or logic 0). If both inputs are high (i.e., $V_A = 5$ V and $V_B = 5$ V), both diodes will be reverse biased (off), and the output voltage will be high at $V_C = 5$ V. The output will be logic 1. The truth table for an AND logic gate is shown in Table 4.2.
Application as a diode rectifier
The input voltage of the diode circuit shown in Fig. 4.4 is
\[ v_S = V_m \sin \omega t \]. The input voltage has zero DC component—that is, \( V_S = 0 \) and \( v_S = V_S + v_s = v_s \). Draw the waveforms of the output voltage \( v_O \) and the diode voltage \( v_D \).

SOLUTION

During the interval \( 0 \leq \omega t \leq \pi \), the voltage across the diode will be positive, and the diode will behave as a short circuit. This is shown in Fig. 4.5(a). Thus, the output voltage \( v_O \) will be the same as the input voltage \( v_S \), and the diode voltage \( v_D \) will be zero. That is,
\[ v_O = v_S \quad \text{for} \quad 0 \leq \omega t \leq \pi \]
\[ v_D = 0 \]

During the interval \( \pi \leq \omega t \leq 2\pi \), the voltage across the diode will be negative, and the diode will be an open circuit, as shown in Fig. 4.5(b). Thus, the output voltage \( v_O \) will be zero, and the diode voltage \( v_D \) will be the

---

NOTE Although it is possible to use diodes to perform logic functions, diode logic circuits are slow and thus are rarely used in practice. We will see in Chapter 15 that the performance of many logic families is far superior. These examples, however, illustrate the “on” and “off” behaviors and conditions of the diodes.

EXAMPLE 4.3

Application as a diode rectifier The input voltage of the diode circuit shown in Fig. 4.4 is \( v_S = v_s = V_m \sin \omega t \). The input voltage has zero DC component—that is, \( V_S = 0 \) and \( v_S = V_S + v_s = v_s \). Draw the waveforms of the output voltage \( v_O \) and the diode voltage \( v_D \).

SOLUTION

During the interval \( 0 \leq \omega t \leq \pi \), the voltage across the diode will be positive, and the diode will behave as a short circuit. This is shown in Fig. 4.5(a). Thus, the output voltage \( v_O \) will be the same as the input voltage \( v_S \), and the diode voltage \( v_D \) will be zero. That is,
\[ v_O = v_S \quad \text{for} \quad 0 \leq \omega t \leq \pi \]
\[ v_D = 0 \]

During the interval \( \pi \leq \omega t \leq 2\pi \), the voltage across the diode will be negative, and the diode will be an open circuit, as shown in Fig. 4.5(b). Thus, the output voltage \( v_O \) will be zero, and the diode voltage \( v_D \) will be the

---

TABLE 4.2 Truth table for Example 4.2

<table>
<thead>
<tr>
<th>Voltages</th>
<th>Logic Levels</th>
</tr>
</thead>
<tbody>
<tr>
<td>( V_A )</td>
<td>( V_B )</td>
</tr>
<tr>
<td>0 (V)</td>
<td>0 (V)</td>
</tr>
<tr>
<td>0 (V)</td>
<td>5 (V)</td>
</tr>
<tr>
<td>5 (V)</td>
<td>0 (V)</td>
</tr>
<tr>
<td>5 (V)</td>
<td>5 (V)</td>
</tr>
</tbody>
</table>

---

FIGURE 4.3 Diode AND logic circuit

---

FIGURE 4.4 Diode circuit for Example 4.3
4.3 Transfer Characteristics of Diode Circuits

The output voltage of a diode circuit depends on whether the diode is on or off. If the input voltage changes with time, as illustrated in Example 4.3, the output voltage is based on the on or off status of the diode(s). The transfer characteristic of a circuit is the relationship between the output voltage and the input voltage. It shows the manner in which the output voltage varies with the input voltage and is independent of the input waveform. Therefore, once the transfer characteristic is known, the output waveform can be determined directly for any given input waveform. The transfer characteristic is useful in describing the behavior of a circuit.

The output voltages of the circuits in Fig. 4.6 can be described as follows. For Fig. 4.6(a), the output voltage \( v_O \) will be the same as the input voltage when the ideal diode conducts. When the diode is off, the output voltage will be zero. That is,

\[
v_O = \begin{cases} 
  v_S & \text{if } v_S > 0 \\
  0 & \text{if } v_S \leq 0
\end{cases}
\]

The waveforms of the input voltage \( v_S \), the output voltage \( v_O \), and the diode voltage \( v_D \) are shown in Fig. 4.5(c).
For Fig. 4.6(b), the output voltage $v_O$ will become zero when the ideal diode conducts. That is,

$$v_O = \begin{cases} 0 & \text{if } v_S > 0 \\ v_S & \text{if } v_S \leq 0 \end{cases}$$

For Fig. 4.6(c), the output voltage $v_O$ will be the same as the input voltage when the diode conducts. That is,

$$v_O = \begin{cases} v_S & \text{if } v_S > V_B \\ V_B & \text{if } v_S \leq V_B \end{cases}$$

For Fig. 4.6(d), the output voltage $v_O$ will be clamped to $V_B$ (i.e., it will remain fixed at $V_B$) when the diode conducts. When the diode is off, the output voltage will be the same as the input voltage. Otherwise, it will be $V_B$. That is,

$$v_O = \begin{cases} V_B & \text{if } v_S > V_B \\ v_S & \text{if } v_S \leq V_B \end{cases}$$

Typical transfer characteristics are also shown in Fig. 4.6.

**KEY POINT OF SECTION 4.3**

- The transfer characteristic relates the output voltage to the input voltage and does not depend on the magnitude and waveform of the input voltage.
4.4 Practical Diodes

The characteristic of a practical diode that distinguishes it from an ideal one is that the practical diode experiences a finite voltage drop when it conducts. This drop is typically in the range of 0.5 V to 0.7 V. If the input voltage to a diode circuit is high enough, this small drop can be ignored. The voltage drop may, however, cause a significant error in electronic circuits, and the diode characteristic should be taken into account in evaluating the performance of diode circuits. To understand the internal characteristics of a practical diode [1], we need to understand its physical operation, which is covered in Chapter 6.

4.4.1 Characteristic of Practical Diodes

The voltage-versus-current (v-i) characteristic of a practical diode is shown in Fig. 4.7. This characteristic, which can be well approximated by an equation known as the Shockley diode equation [2–4], is given by

\[ i_D = I_S (e^{v_D/nV_T} - 1) \tag{4.1} \]

where
- \( i_D \) = current through the diode, in A
- \( v_D \) = diode voltage with the anode positive with respect to the cathode, in V
- \( I_S \) = leakage (or reverse saturation) current, typically in the range of \( 10^{-6} \) A to \( 10^{-15} \) A
- \( n \) = empirical constant known as the emission coefficient or the ideality factor, whose value varies from 1 to 2

The emission coefficient \( n \) depends on the material and the physical construction of the diode. For germanium diodes, \( n \) is considered to be 1. For silicon diodes, the predicted value of \( n \) is 2 at very small or large currents; but for most practical silicon diodes, the value of \( n \) falls in the range of 1.1 to 1.8.

**FIGURE 4.7** Voltage-versus-current characteristic of practical diode
$V_T$ in Eq. (4.1) is a constant called the thermal voltage, and it is given by

$$V_T = \frac{kT}{q} \quad (4.2)$$

where

- $q =$ electron charge $= 1.6022 \times 10^{-19}$ coulomb (C)
- $T_K =$ absolute temperature in kelvins $= 273 + T_{\text{Celsius}}$
- $k =$ Boltzmann’s constant $= 1.3806 \times 10^{-23}$ J per kelvin

At a junction temperature of 25°C, Eq. (4.2) gives the value of $V_T$ as

$$V_T = \frac{kT}{q} = \frac{(1.3806 \times 10^{-23})(273 + 25)}{1.6022 \times 10^{-19}} = \frac{T_K}{11,605.1} \approx 25.8 \text{ mV}$$

At a specific temperature, the leakage current $I_S$ will remain constant for a given diode. For small-signal (or low-power) diodes, the typical value of $I_S$ is $10^{-9}$ A. We can divide the diode characteristic of Fig. 4.7 into three regions, as follows:

- Forward-biased region, where $v_D > 0$
- Reverse-biased region, where $v_D < 0$
- Breakdown region, where $-V_{ZK} > v_D > 0$

**Forward-Biased Region**

In the forward-biased region, $v_D > 0$. The diode current $i_D$ will be very small if the diode voltage $v_D$ is less than a specific value $V_{TD}$, known as the threshold voltage or the cut-in voltage or the turn-on voltage (typically 0.7 V). The diode conducts fully if $v_D$ is higher than $V_{TD}$. Thus, the threshold voltage is the voltage at which a forward-biased diode begins to conduct fully.

Assume that a small forward voltage of $v_D = 0.1$ V is applied to a diode of $n = 1$. At room temperature, $V_T = 25.8$ mV. From Eq. (4.1), we can find the diode current $i_D$ as

$$i_D = I_S(e^{v_D/nV_T} - 1) = I_S(e^{0.1/(1 \times 0.0258)} - 1) = I_S(48.23 - 1) \approx 48.23I_S \text{ with 2.1% error}$$

Therefore, for $v_D > 0.1$ V, which is usually the case, $i_D >> I_S$, and Eq. (4.1) can be approximated within 2.1% error by

$$i_D = I_S(e^{v_D/nV_T} - 1) \approx I_S e^{v_D/nV_T} \quad (4.3)$$

**Reverse-Biased Region**

In the reverse-biased region, $-V_{ZK} < v_D < 0$. That is, $v_D$ is negative. If $|v_D| >> V_T$, which occurs for $v_D < -0.1$ V, the exponential term in Eq. (4.1) becomes negligibly small compared to unity, and the diode current $i_D$ becomes

$$i_D = I_S(e^{-|v_D|/nV_T} - 1) \approx -I_S \quad (4.4)$$
which indicates that the diode current $i_D$ remains constant in the reverse direction and is equal to $I_S$ in magnitude.

### Breakdown Region

In the breakdown region, the reverse voltage is high—usually greater than 100 V. If the magnitude of the reverse voltage exceeds a specified voltage known as the *breakdown voltage* $V_{BR}$, the corresponding reverse current $I_{BV}$ increases rapidly for a small change in reverse voltage beyond $V_{BR}$. Operation in the breakdown region will not be destructive to the diode provided the power dissipation ($P_D = v_D i_D$) is kept within the safe level specified in the manufacturer’s data sheet. It is often necessary, however, to limit the reverse current in the breakdown region so that the power dissipation falls within a permissible range.

### 4.4.2 Determination of Diode Constants

Diode constants $I_S$ and $n$ can be determined either from experimentally measured $v$-$i$ data or from the $v$-$i$ characteristic. There are a number of steps to be followed. Taking the natural (base $e$) logarithm of both sides of Eq. (4.3), we get

$$\ln i_D = \ln I_S + \frac{v_D}{nV_T}$$

which, after simplification, gives the diode voltage $v_D$ as

$$v_D = nV_T \ln \left( \frac{i_D}{I_S} \right) \quad (4.5)$$

If we convert the natural log of base $e$ to the logarithm of base 10, Eq. (4.5) becomes

$$v_D = 2.3nV_T \log \left( \frac{i_D}{I_S} \right) \quad (4.6)$$

which indicates that the diode voltage $v_D$ is a nonlinear function of the diode current $i_D$. If $V_{D1}$ is the diode voltage corresponding to diode current $I_{D1}$, Eq. (4.5) gives

$$V_{D1} = nV_T \ln \left( \frac{I_{D1}}{I_S} \right) \quad (4.7)$$

Similarly, if $V_{D2}$ is the diode voltage corresponding to the diode current $I_{D2}$, we get

$$V_{D2} = nV_T \ln \left( \frac{I_{D2}}{I_S} \right) \quad (4.8)$$

Therefore, the difference in diode voltages can be expressed by

$$V_{D2} - V_{D1} = nV_T \ln \left( \frac{I_{D2}}{I_S} \right) - nV_T \ln \left( \frac{I_{D1}}{I_S} \right) = nV_T \ln \left( \frac{I_{D2}}{I_{D1}} \right) \quad (4.9)$$
which can be converted to the logarithm of base 10 as

\[ V_{D2} - V_{D1} = 2.3nV_T \log \left( \frac{I_{D2}}{I_{D1}} \right) \]  

(4.10)

This shows that for a decade (i.e., a factor of 10) change in diode current \( I_{D2} = 10I_{D1} \), the diode voltage will change by \( 2.3nV_T \). Thus, Eq. (4.6) can be written as

\[ v_D = 2.3nV_T \log i_D - 2.3nV_T \log I_S \]  

(4.11)

If this equation is plotted on a semilog scale with \( v_D \) on the vertical linear axis and \( i_D \) on the horizontal log axis, the characteristic will be a straight line with a slope of \( +2.3nV_T \) per decade of current, and its equation will have the form of a standard straight-line equation—that is,

\[ y = mx - c \]

where \( c = 2.3nV_T \log I_S \) and \( m = 2.3nV_T \) per decade of current. The plot of Eq. (4.11) is shown in Fig. 4.8. With \( v_D \) in the linear scale and \( i_D \) in the log scale.

Thus, based on the experimental results from an unknown diode, the \( v-i \) characteristic can be plotted on a semilog scale. The values of \( I_S \) and \( n \) can be calculated as follows:

**Step 1.** Plot \( v_D \) against \( i_D \) on a semilog scale, as shown in Fig. 4.8 with \( v_D \) in the linear scale and \( i_D \) in the log scale.

**Step 2.** Find the slope \( m \) per decade of current change on the \( v_D \)-axis.

**Step 3.** Find the emission coefficient \( n \) for the known value of slope \( m \)—that is,

\[ n = \frac{m}{2.3V_T} = \frac{m}{2.3 \times 0.0258} \]

**Step 4.** Find the intercept \( c \) on the \( v_D \)-axis.

**Step 5.** Find the value of \( I_S \) from

\[ 2.3nV_T \log I_S = c \]
Once the values of $I_S$ and $n$ have been determined, the diode voltage $v_D$ can be expressed explicitly as a function of the diode current $i_D$, as in Eq. (4.5).

**EXAMPLE 4.4**

**Finding diode constants** The measured values of a diode at a junction temperature of 25°C are given by

$$V_D = \begin{cases} 0.5 \text{ V} & \text{at } I_D = 5 \mu\text{A} \\ 0.6 \text{ V} & \text{at } I_D = 100 \mu\text{A} \end{cases}$$

Determine (a) the emission coefficient $n$ and (b) the leakage current $I_S$.

**SOLUTION**

$V_{D1} = 0.5 \text{ V at } I_{D1} = 5 \mu\text{A}$, and $V_{D2} = 0.6 \text{ V at } I_{D2} = 100 \mu\text{A}$. At 25°C, $V_T = 25.8 \text{ mV}$.

(a) From Eq. (4.9),

$$V_{D2} - V_{D1} = n V_T \ln \left( \frac{I_{D2}}{I_{D1}} \right) \text{ or } 0.6 - 0.5 = n V_T \ln \left( \frac{100 \mu\text{A}}{5 \mu\text{A}} \right)$$

which gives $n V_T = 0.03338$, and $n = 0.03338$ $V_T = 0.03338 \times 10^{-3} = 1.294$.

(b) From Eq. (4.5),

$$V_{D1} = n V_T \ln \left( \frac{I_{D1}}{I_S} \right) \text{ or } 0.5 = 0.03338 \ln \left( \frac{5 \times 10^{-6}}{I_S} \right)$$

which gives $I_S = 1.56193 \times 10^{-12}$ A.

**4.4.3 Temperature Effects**

The leakage current $I_S$ depends on the junction temperature $T_j$ (in Celsius) and increases at the rate of approximately $+7.2\%$ per degree Celsius for silicon and germanium diodes [1, 5]. Thus, by adding the increments for each degree rise in the junction temperature up to 10°C, we get

$$I_S(T_j = 10) = I_S \left[ 1 + 0.072 + (0.072 + 0.072^2) + (0.072^2 + 0.072^3) \\
+ (0.072^3 + 0.072^4) + (0.072^4 + 0.072^5) + (0.072^5 + 0.072^6) \\
+ (0.072^6 + 0.072^7) + (0.072^7 + 0.072^8) + (0.072^8 + 0.072^9) \\
+ (0.072^9 + 0.072^{10}) \right]$$

$$\approx 2I_S$$
That is, $I_S$ approximately doubles for every $10^\circ\text{C}$ increase in temperature and can be related to any temperature change by

$$I_S(T_j) = I_S(T_o) \times 2^{(T_j-T_o)/10} = I_S(T_o) \times 2^{0.1(T_j-T_o)} \tag{4.12}$$

where $I_S(T_o)$ is the leakage current at temperature $T_o$. Substituting $V_T = kT/q$ in Eq. (4.5) gives the temperature dependence of the forward diode voltage. That is,

$$v_D = \frac{nk(273 + T_j)}{q} \ln \left( \frac{i_D}{I_S} \right) \tag{4.13}$$

which, after differentiation of $v_D$ with respect to $T_j$, gives

$$\frac{\partial v_D}{\partial T_j} = \frac{nk}{q} \ln \left( \frac{i_D}{I_S} \right) - \frac{nk(273 + T_j)}{qI_S} \frac{dI_S}{dT_j} = \frac{v_D}{273 + T_j} - \frac{nV_T}{I_S} \frac{dI_S}{dT_j} \tag{4.14}$$

which decreases with the temperature $T_j$ for a constant $v_D$. At a given diode current $i_D$, the diode voltage $v_D$ decreases with the temperature. The temperature dependence of the forward diode characteristic is shown in Fig. 4.9.

The threshold voltage $V_{TD}$ also depends on the temperature $T_j$. As the temperature increases, $V_{TD}$ decreases, and vice versa. $V_{TD}$, which has an approximately linear relationship to temperature $T_j$, is given by

$$V_{TD}(T_j) = V_{TD}(T_o) + K_{TC}(T_j - T_o) \tag{4.15}$$

where $T_o = \text{junction temperature at } 25^\circ\text{C}$

$T_j = \text{new junction temperature, in } ^\circ\text{C}$

$V_{TD}(T_o) = \text{threshold voltage at junction temperature } T_o$, which is 0.7 V for a silicon diode, 0.3 V for a germanium diode, and 0.3 V for a Schottky diode (discussed in Sec. 6.6)

$V_{TD}(T_j) = \text{threshold voltage at new junction temperature } T_j$

$K_{TC} = \text{temperature coefficient, in V/}^\circ\text{C, which is } -2.5 \text{ mV/}^\circ\text{C for a germanium diode, } -2 \text{ mV/}^\circ\text{C for a silicon diode, and } -1.5 \text{ mV/}^\circ\text{C for a Schottky diode}$
EXAMPLE 4.5

Finding the temperature dependence of threshold voltage The threshold voltage $V_{TD}$ of a silicon diode is 0.7 V at 25°C. Find the threshold voltage $V_{TD}$ at (a) $T_j = 100^\circ$C and (b) $T_j = -100^\circ$C.

SOLUTION

At $T_o = 25^\circ$C, $V_{TD}(T_o) = 0.7$ V. The temperature coefficient for silicon is $K_{TC} = -2$ mV/°C.

(a) At $T_j = 100^\circ$C, from Eq. (4.15),

$$V_{TD}(T_j) = V_{TD}(T_o) + K_{TC}(T_j - T_o)$$

$$= 0.7 - 2 \times 10^{-3} \times (100 - 25) = 0.55$$ V

(b) At $T_j = -100^\circ$C, from Eq. (4.15),

$$V_{TD}(T_j) = V_{TD}(T_o) + K_{TC}(T_j - T_o)$$

$$= 0.7 - 2 \times 10^{-3} \times (-100 - 25) = 0.95$$ V

Thus, a change in the temperature can significantly change the value of $V_{TD}$.

EXAMPLE 4.6

Finding the temperature dependence of diode current The leakage current of a silicon diode is $I_S = 10^{-9}$ A at 25°C, and the emission coefficient is $n = 2$. The operating junction temperature is $T_j = 60^\circ$C. Determine (a) the leakage current $I_S$ and (b) the diode current $i_D$ at $v_D = 0.8$ V.

SOLUTION

$I_S = 10^{-9}$ A at $T_o = 25^\circ$C, $T_j = 60^\circ$C, and $v_D = 0.8$ V.

(a) From Eq. (4.12), the value of $I_S$ at $T_j = 60^\circ$C is

$$I_S(T_j = 60) = I_S(T_o)n^{0.1(T_j - T_o)} = 10^{-9} \times 2^{0.1(60 - 25)} = 11.31 \times 10^{-9}$$ A

(b) At $T_K = 273 + 60 = 333$ K, Eq. (4.2) gives

$$V_T = \frac{kT_K}{q} = \frac{1.3806 \times 10^{-23} \times (273 + 60)}{1.6022 \times 10^{-19}} = 28.69$$ mV

From Eq. (4.3), we can find the diode current $i_D$:

$$i_D \approx I_Se^{e^{qV_T}/kT} = 11.31 \times 10^{-9} \times e^{0.8/(2 \times 0.02869)} = 12.84$$ mA
4.5 Analysis of Practical Diode Circuits

A diode is used as a part of an electronic circuit, and the diode current \( i_D \) becomes dependent on other circuit elements. A simple diode circuit is shown in Fig. 4.10. Applying Kirchhoff’s voltage law (KVL), we can express the source voltage \( V_S \) and the diode current \( i_D \) by

\[
V_S = v_D + R_L i_D
\]

which gives the diode current \( i_D \) as

\[
i_D = \frac{V_S - v_D}{R_L}
\]  
(4.16)

Since the diode will be forward biased, the diode current \( i_D \) is related to the diode voltage \( v_D \) by the Shockley diode equation,

\[
i_D = I_S (e^{v_D/nV_T} - 1)
\]  
(4.17)

which shows that \( i_D \) depends on \( v_D \), which in turn depends on \( i_D \). Thus, Eqs. (4.16) and (4.17) can be solved for \( v_D \) and \( i_D \) by any of the following methods: graphical method, approximate method, or iterative method.

4.5.1 Graphical Method

Let us assume that \( v_D \) is positive. Then Eq. (4.17) represents the diode characteristic in the forward direction. Equation (4.16) is the equation of a straight line with a slope of \(-1/R_L\) and represents the load characteristic known as the load line. If Eqs. (4.16) and (4.17) are plotted on the same graph, as shown in Fig. 4.11,
the diode characteristic will intersect the load line at a point \( Q \), which is the operating point (or quiescent point) of the diode. The coordinates of this \( Q \)-point give the quiescent diode voltage \( \text{V}_{\text{DQ}} \) (or simply \( \text{V}_D \)) and the quiescent diode current \( \text{I}_{\text{DQ}} \) (or simply \( \text{I}_D \)). This graphical approach [6] is not a convenient method of analysis, and thus it is rarely used in the analysis of diode circuits. However, it helps us understand the concept of \( Q \)-point and the mechanism of diode circuit analysis.

### 4.5.2 Approximate Method

To solve Eqs. (4.16) and (4.17) by the approximate method, we assume the diode to have a constant voltage drop equal to the threshold voltage \( \text{V}_{\text{TD}} \). That is, \( \text{v}_D = \text{V}_{\text{TD}} \), and the diode characteristic is approximated as a vertical line, as shown in Fig. 4.12. The threshold voltage \( \text{V}_{\text{TD}} \) of small-signal diodes lies in the range of 0.5 V to 1.0 V. The diode drop for silicon diodes is approximately \( \text{v}_D = \text{V}_{\text{TD}} = 0.7 \text{ V} \), and that for germanium diodes is \( \text{v}_D = \text{V}_{\text{TD}} = 0.3 \text{ V} \). Using the approximate value of \( \text{v}_D \), we can find the diode current \( \text{i}_D \) from Eq. (4.16) as follows:

\[
\text{i}_D = \frac{\text{V}_S - \text{v}_D}{\text{R}_L} = \frac{\text{V}_S - 0.7}{\text{R}_L} \quad \text{(or 0.3 for germanium)}
\]

As an example, let \( \text{V}_S = 10 \text{ V} \), \( \text{v}_D = \text{V}_{\text{TD}} = 0.7 \text{ V} \), and \( \text{R}_L = 1 \text{ kΩ} \). Then the operating \( Q \)-point current \( \text{I}_D \) becomes \( \text{I}_D = \text{i}_D = (10 - 0.7) \text{ V}/(1 \text{ kΩ}) = 9.3 \text{ mA} \).

This method gives an approximate solution and does not take into account the nonlinear characteristic described by Eq. (4.17). This approximation is adequate, however, for many applications and is useful as a starting point for a circuit design.

**NOTE** \( \text{v}_D \) and \( \text{i}_D \) are the variable quantities, whereas \( \text{V}_D \) and \( \text{I}_D \) are their fixed values, respectively.

### 4.5.3 Iterative Method

The iterative method uses an iterative solution to find the values of \( \text{i}_D \) and \( \text{v}_D \) from the load line of Eq. (4.16) and the nonlinear diode characteristic of Eq. (4.17). First a small value of \( \text{v}_D \) is assumed and Eq. (4.16) is used to find an approximate value of \( \text{i}_D \), which is then used to calculate a better
approximation of diode voltage $v_D$ from Eq. (4.17). This completes one iteration; the iterations continue until the desired accuracy has been obtained. The steps can be described as follows:

**Step 1.** Start with an arbitrary point $a$, as shown in Fig. 4.13, and assume a fixed value of $v_D$ (say 0.7 V) at a specified value of $i_D$.

**Step 2.** Find point $b$ by calculating the value of $i_D$ from the load characteristic described by Eq. (4.16).

**Step 3.** Find point $c$ by calculating a modified value of $v_D$ from the diode characteristic described by Eq. (4.17) or Eq. (4.9). This completes one iteration.

**Step 4.** Find point $d$ by calculating the value of $i_D$ from the load characteristic described by Eq. (4.16).

**Step 5.** Find point $e$ by calculating a modified value of $v_D$ from the diode characteristic described by Eq. (4.17). This completes two iterations.

**Step 6.** Find point $f$ by calculating the value of $i_D$ from the load characteristic described by Eq. (4.16).

**Step 7.** Find point $g$ by calculating a modified value of $v_D$ from the diode characteristic described by Eq. (4.17) or Eq. (4.9). This completes three iterations.

This process is continued until the values of $i_D$ and $v_D$ converge to within the range of desired accuracy.

### 4.5.4 Mathematical Method

Equating $i_D$ in Eq. (4.16) with that of Eq. (4.17), we get the following relationship:

$$i_D = \frac{(V_S - v_D/R_L)}{R_L} = I_S (e^{v_D/\eta V_T} - 1) = I_S e^{v_D/\eta V_T}$$

This can be solved for the diode voltage $v_D$ by using computer software such as MATHCAD or MATLAB if the values of $\eta$, $R_L$, $V_T$, and $I_S$ are known. Once the value of $v_D$ is found, the value of $i_D$ can be determined from Eq. (4.16) or Eq. (4.17).

### EXAMPLE 4.7

**Finding the Q-point of a diode circuit** The diode circuit shown in Fig. 4.10 has $R_L = 1$ k$\Omega$ and $V_S = 10$ V. The emission coefficient is $n = 1.84$, the thermal voltage is $V_T = 25.8$ mV, and the leakage current is $I_S = 2.682 \times 10^{-9}$ A. Calculate the Q-point (or the operating point) $V_D$ and $I_D$ by (a) the approximate method, (b) the iterative method with three iterations, and (c) the mathematical method. Assume a default value of $V_D = 0.61$ V as the initial guess.

**SOLUTION**

If the initial guess of $V_D$ is not specified, we can use the default value of $V_D = 0.70$ V. But it should affect the final results as long as the initial guess value is reasonable around 0.7 V.
(a) $V_D = v_D = 0.61$ V. From Eq. (4.16),

$$i_D = \frac{(V_S - v_D) / R_L}{1\ \text{k}\Omega} = 9.39\text{ mA}.$$ 

(b) $R_L = 1\ \text{k}\Omega$, $n = 1.84$, $V_T = 25.8\text{ mV}$, and $v_D = 0.61$ V at $i_D = 1$ mA.

**Iteration 1:** Assume $v_D = 0.61$ V. From Eq. (4.16),

$$i_D = \frac{V_S - v_D}{R_L} = \frac{(10 - 0.61)\ V}{1\ \text{k}\Omega} = 9.39\text{ mA}$$

From Eq. (4.5), the new value of $v_D$ is

$$v_D^{\text{(new)}} = nV_T \ln \left( \frac{i_D}{I_S} \right)$$

$$= 1.84 \times 0.0258 \ln \left( \frac{9.39\text{ mA}}{2.682 \times 10^{-9}} \right) = 0.7153\text{ V}$$

**Iteration 2:** Assume the values of $v_D$ from the previous iteration. That is, set $v_D = v_D^{\text{(new)}} = 0.7153$ V. From Eq. (4.16),

$$i_D^{\text{(new)}} = \frac{V_S - v_D}{R_L} = \frac{(10 - 0.7153)\ V}{1\ \text{k}\Omega} = 9.2847\text{ mA}$$

From Eq. (4.5), the new value of $v_D$ is

$$v_D^{\text{(new)}} = nV_T \ln \left( \frac{i_D^{\text{(new)}}}{I_S} \right)$$

$$= 1.84 \times 0.0258 \ln \left( \frac{9.2847}{2.682 \times 10^{-9}} \right) = 0.7148\text{ V}$$

**Iteration 3:** Assume the values of $v_D$ from the previous iteration. That is, set $v_D = v_D^{\text{(new)}} = 0.7148$ V. From Eq. (4.16),

$$i_D^{\text{(new)}} = \frac{V_S - v_D}{R_L} = \frac{(10 - 0.7148)\ V}{1\ \text{k}\Omega} = 9.285\text{ mA}$$

From Eq. (4.5), the new value of $v_D$ is

$$v_D^{\text{(new)}} = nV_T \ln \left( \frac{i_D^{\text{(new)}}}{I_S} \right)$$

$$= 1.84 \times 0.0258 \ln \left( \frac{9.285}{2.682 \times 10^{-9}} \right) = 0.7158\text{ V}$$

Therefore, after three iterations, $V_D = v_D^{\text{(new)}} = 0.7158$ V and $I_D = i_D^{\text{(new)}} = 9.285$ mA. Note that the results of iteration 3 do not differ significantly from those of iteration 2. In fact, there was no need for iteration 3.

(c) Substituting for the given values in Eq. (4.18),

$$\frac{(V_S - v_D)}{R_L} \approx I_S e^{v_D / \eta V_T},$$

we get $$(10 - v_D) / \text{k}\Omega = 2.682 \times 10^{-9} \times e^{v_D/(1.84 \times 25.8 \times 10^{-3})}$$ which, after solving by MATHCAD software function, gives $V_D = 0.7148$ V and

$$I_D = I_S e^{-\eta V_T} = (2.682 \times 10^{-9}) e^{-0.7148/(1.84 \times 25.8 \times 10^{-3})} = 9.2845\text{ mA}$$

**NOTE:** Four-digit answers were used to control computational errors and the number of iterations needed to reach the solution. In reality, resistors will have tolerances, and such accuracy may not be necessary.
### TABLE 4.3
The values of $V_D$ and $I_D$ obtained by different methods

<table>
<thead>
<tr>
<th>Method</th>
<th>$V_D$ (V)</th>
<th>$I_D$ (mA)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Approximate</td>
<td>0.7</td>
<td>9.39</td>
</tr>
<tr>
<td>Iterative</td>
<td>0.7148</td>
<td>9.2852</td>
</tr>
<tr>
<td>Mathematical</td>
<td>0.7148</td>
<td>9.2845</td>
</tr>
</tbody>
</table>

### KEY POINTS OF SECTION 4.5
- The analysis of a diode circuit involves solving a nonlinear diode equation.
- The graphical method is rarely used.
- The approximate method gives a quick answer but approximate values. The mathematical solution by MATHCAD gives quick but accurate results. In the absence of any computer-aided solution, the iterative method also gives accurate results. Comparisons of the values obtained by these three methods for Example 4.7 are shown in Table 4.3.

### 4.6 Modeling of Practical Diodes

In practice, multiple diodes are used in a circuit. Therefore, diode circuits become complex, and analysis by the graphical or iterative method becomes time-consuming and laborious. To simplify the analysis and design of diode circuits, we can represent a diode by one of the following models: constant-drop DC model, piecewise linear DC model, low-frequency AC model, high-frequency AC model, or SPICE diode model.

#### 4.6.1 Constant-Drop DC Model

The constant-drop DC model assumes that a conducting diode has a voltage drop $v_D$ that remains almost constant and is independent of the diode current. Therefore, the diode characteristic becomes a vertical line at the threshold voltage; that is, $v_D = V_{TD}$. The $Q$-point is determined by adding the load line to the approximate diode characteristic, as shown in Fig. 4.14(a). The diode voltage $v_D$ is expressed by

$$v_D = \begin{cases} 
V_{TD} & \text{for } v_D \geq V_{TD} \\
0 & \text{for } v_D < V_{TD}
\end{cases}$$

The circuit model is shown in Fig. 4.14(b). The typical value of $V_{TD}$ is 0.7 V for silicon diodes and 0.3 V for germanium diodes. With this model, the diode current $i_D$ can be determined from

$$i_D = \frac{V_S - V_{TD}}{R_L}$$

(4.19)
4.6.2 Piecewise Linear DC Model

The voltage drop across a practical diode increases with its current. The diode characteristic can be represented approximately by a fixed voltage drop $V_{TD}$ and a straight line, as shown in Fig. 4.15(a). The straight line $a$ takes into account the current dependence of the voltage drop, and it represents a fixed resistance $R_D$, which remains constant. The line $a$ can pass through at most two points; it is usually drawn tangent to the diode characteristic at the estimated $Q$-point. This model represents the diode characteristic approximately by two piecewise parts: a fixed part and a current-dependent part. A piecewise linear representation of the diode is shown in Fig. 4.15(b). The steps for determining the model parameters are as follows:

**Step 1.** Draw a line tangent to the current-dependent part of the forward diode characteristic at the estimated $Q$-point. A best-fit line through the current-dependent part is generally acceptable.

**Step 2.** Use the intercept on the $v_D$-axis as the fixed drop $V_{TD}$.
Step 3. Choose a suitable current $i_X$ on the $i_D$-axis of the tangent line $a$, and read the corresponding voltage $v_X$ on the $v_D$-axis. $i_X$ is normally chosen to be the maximum diode current; that is, $i_X = i_D(\text{max}) = \frac{V_S}{R_L}$.

Step 4. Calculate the resistance $R_D$, which is the inverse slope of the tangent line.

$$R_D = \frac{\Delta v_D}{\Delta i_D} \bigg|_{\text{at estimated } Q\text{-point}} = \frac{v_X - V_{TD}}{i_X} \quad (4.20)$$

This model determines the value of $R_D$ at the $Q$-point and does not take into account the actual shape of the diode characteristic at other points. Therefore, if the $Q$-point changes as a result of variations in the load resistance $R_L$ or the DC supply voltage $V_S$, the value of $R_D$ will change. However, the piecewise model is quite satisfactory for most applications. Using this model and applying KVL, we find that the diode current $i_D$ in Fig. 4.15(b) is given by

$$V_S = V_{TD} + R_D i_D + R_L i_D \quad (4.21)$$

which gives the diode current $i_D$ as

$$i_D = \frac{V_S - V_{TD}}{R_D + R_L} \quad (4.22)$$

EXAMPLE 4.8

Finding the $Q$-point of a diode circuit and diode model parameters by two different methods

The diode circuit shown in Fig. 4.16(a) has $V_S = 10$ V and $R_L = 1$ kΩ. The diode characteristic is shown in Fig. 4.16(b). Determine the diode voltage $v_D$, the diode current $i_D$, and the load voltage $v_O$ by using (a) the piecewise linear DC model and (b) the constant-drop DC model.

**FIGURE 4.16** Diode circuit for Example 4.8
**SOLUTION**

\[ V_S = 10 \text{ V and } R_L = 1 \text{ k} \Omega \text{. Thus, } i_{D(\text{max})} = \frac{V_S}{R_L} = \frac{10 \text{ V}}{1 \text{ k} \Omega} = 10 \text{ mA} \text{.} \]

(a) If we follow the steps described in Sec. 4.6.2, the tangent or best-fit line gives \( V_{TD} = 0.6 \text{ V} \) and \( v_X = 0.8 \text{ V} \) at \( i_X = i_{D(\text{max})} = 10 \text{ mA} \). From Eq. (4.20), the resistance \( R_D \) of the current-dependent part is

\[ R_D = \frac{v_X - V_{TD}}{i_X} = \frac{(0.8 - 0.6) \text{ V}}{(10 \text{ mA})} = 20 \Omega \]

From Fig. 4.15(b), the diode current is

\[ i_D = \frac{V_S - V_{TD}}{R_L + R_D} = \frac{(10 - 0.6) \text{ V}}{1 \text{ k} \Omega + 20 \Omega} = 9.22 \text{ mA} \]

From Fig. 4.15(b), the diode voltage is

\[ v_D = V_{TD} + R_D i_D = 0.6 + 20 \times 9.22 \times 10^{-3} = 0.784 \text{ V} \]

Thus, the load voltage becomes

\[ v_O = V_S - V_D = 10 - 0.784 = 9.216 \text{ V} \]

**NOTE:** The diode mode parameters are \( V_{TD} = 0.6 \text{ V} \) and \( R_D = 20 \Omega \).

(b) Using Eq. (4.19) for the constant-drop DC model of Fig. 4.14(b), we get the diode current

\[ i_D = \frac{V_S - V_{TD}}{R_L} = \frac{(10 - 0.6) \text{ V}}{1 \text{ k} \Omega} = 9.4 \text{ mA} \]

The load voltage is

\[ v_O = V_S - V_{TD} = 10 - 0.6 = 9.4 \text{ V} \]

for an error of \((9.4 - 9.216) 9.4 = 1.96\%\) compared to the piecewise linear model.

**NOTES:**

1. The diode mode parameters are \( V_{TD} = 0.6 \text{ V} \) and \( R_D = 0 \Omega \).
2. If the supply voltage \( V_S \) is much greater than the diode voltage drop \( v_D \), the constant-drop DC model will give acceptable results. If the diode voltage \( v_D \) is comparable to the supply voltage \( V_S \), the piecewise linear DC model, which gives better results, is generally acceptable in most applications.

### 4.6.3 Low-Frequency Small-Signal Model

In electronic circuits, a DC supply normally sets the DC operating point of electronic devices including diodes, and an AC signal is usually then superimposed on the operating point. Thus, the operating point, which consists of both a DC component and an AC signal, will vary with the magnitude of the AC signal.
Since the $i_D-v_D$ characteristic of a diode is nonlinear, the diode current $i_D$ will also vary nonlinearly with the AC signal voltage. The magnitude of the AC signal is generally small, however, so the operating point changes by only a small amount. Thus, the slope of the characteristic ($\Delta i_D$ versus $\Delta v_D$) can be approximated linearly. Under this condition, we can represent the diode as a resistance in order to determine the response of the circuit to this small AC signal. That is, the nonlinear diode characteristic can be linearized at the operating point. A small-signal model [7] is widely used for the analysis and design of electronic circuits in order to obtain their small-signal behavior.

Figure 4.17(a) shows a diode circuit with a DC source $V_D$, which sets the operating point at $Q$, defined by coordinates $V_D$ and $I_D$. If a small-amplitude sinusoidal voltage $v_d$ is superimposed on $V_D$, the operating point will vary with the time-varying AC signal $v_d$. Therefore, if the diode voltage varies between $(V_D + V_m)$ and $(V_D - V_m)$, the corresponding diode current will vary between $(I_D + \Delta i_D)$ and $(I_D - \Delta i_D)$. This is illustrated in Fig. 4.17(b), in which the change in the AC diode current $i_d$ is assumed to be approximately sinusoidal in response to a sinusoidal voltage $v_d$. However, the diode characteristic is nonlinear and the diode current will be slightly distorted.

Under small-signal conditions, the diode characteristic around the $Q$-point can be approximated by a straight line and modeled by a resistance called the dynamic resistance or AC resistance or small-signal resistance $r_d$, which is defined by

$$\frac{1}{r_d} = g_d = \left. \frac{\Delta i_D}{\Delta v_D} \right|_{\text{at } Q\text{-point}}$$

(4.23)

where $g_d$ is the small-signal diode transconductance and depends on the slope of the diode characteristic at the operating point. Since $r_d$ is determined from the slope of the diode characteristic at the $Q$-point,
its value should be the same as $R_D$ of Fig. 4.15(b) if the tangent line is drawn accurately at the DC operating point.

**Determining $r_d$ by Differentiating**

If the operating point $(V_D, I_D)$ is known for a given diode characteristic and a load line, we can determine the value of $r_d (= R_D)$ directly by considering a change in diode voltage around the operating point. If $\Delta V_D$ and $\Delta I_D$ are small, tending to zero, Eq. (4.23) becomes

$$g_d = \frac{1}{r_d} = \left. \frac{dI_D}{dV_D} \right|_{Q\text{-point}}$$  \hspace{1cm} (4.24)

If $V_D > 0.1$ V, which is usually the case when the diode is operated in the forward direction, then the diode current $I_D$ is related to the diode voltage $V_D$ by

$$I_D = I_S(e^{\frac{V_D}{nVT}} - 1) \approx I_S e^{\frac{V_D}{nVT}}$$ \hspace{1cm} (4.25)

Substituting $I_D$ from Eq. (4.25) into Eq. (4.24) and differentiating $I_D$ with respect to $V_D$ gives

$$g_d = \frac{1}{r_d} = \left. \frac{dI_D}{dV_D} \right|_{Q\text{-point}} = I_S \frac{1}{nVT} e^{\frac{V_D}{nVT}} = \frac{i_D + I_S}{nVT}$$ \hspace{1cm} (4.26)

which gives the AC resistance ($r_d = R_D$) at the operating point $(V_D, I_D)$. That is,

$$r_d = R_D = \frac{1}{g_d} = \frac{nVT}{I_D + I_S} \approx \frac{nVT}{I_D} \quad \text{since} \quad i_D = I_D \quad \text{and} \quad I_D >> I_S$$ \hspace{1cm} (4.27)

$$\approx \frac{0.0258}{I_D} \quad \text{at} \quad 25°C \quad \text{and for} \quad n = 1$$ \hspace{1cm} (4.28)

Notice from Eq. (4.27) that the determination of the AC resistance requires the determination of the diode current $I_D$ at the $Q$-point.

**Determining $r_d$ by Taylor Series Expansion**

Equation (4.27) can also be derived by Taylor series expansion. The instantaneous diode voltage $V_D$ is the sum of $V_D$ and $V_d$. That is,

$$V_D = V_D + V_d$$ \hspace{1cm} (4.29)

Substituting $V_D = V_D + V_d$ into Eq. (4.25) gives the instantaneous diode current $I_D$:

$$I_D \approx I_S e^{\frac{(V_D + V_d)/nVT}{nVT}} = I_S e^{\frac{V_D}{nVT} e^{\frac{V_d}{nVT}}}
\hspace{1cm} = I_D e^{\frac{V_d}{nVT}} \quad \text{since} \quad I_D = I_S e^{\frac{V_D}{nVT}}$$ \hspace{1cm} (4.30)

If the amplitude of the sinusoidal voltage $V_d$ is very small compared to $nVT$, so that $V_d << nVT$, we can use the relation $e^x \approx 1 + x$. Equation (4.30) can be expanded in Taylor series with the first two terms:

$$I_D \approx I_D \left(1 + \frac{V_d}{nVT}\right) = I_D + i_d(t)$$ \hspace{1cm} (4.31)
Thus, the instantaneous diode current $i_D$ has two components: a DC component $I_D$ and a small-signal AC component $i_d$. This is a mathematical derivation of the principle of superposition introduced in Appendix B. From Eq. (4.31), the AC diode current $i_d$ is defined by

$$i_d = \frac{v_d}{nV_T}I_D$$

which gives the small-signal AC resistance $r_d$ as

$$r_d = \frac{v_d}{i_d} = \frac{nV_T}{I_D}$$

which is the same as Eq. (4.27). The value of $r_d$ should ideally be the same as the value of $R_D$ if the tangent line is drawn accurately at the DC operating point.

The small-signal AC model of a diode is shown in Fig. 4.17(c). This model is known as the low-frequency small-signal AC model. It does not take into account the frequency dependency of the diode.

**NOTE**

1. The AC resistance $r_d$ takes into account the shape of the curve and represents the slope of the characteristic at the $Q$-point. If the $Q$-point changes, the value of $r_d$ will also change.
2. $R_D$ is determined from the slope of the diode characteristic at an estimated $Q$-point, whereas $r_d$ is determined from the Shockley diode equation. If $r_d$ and $R_D$ are determined from the two methods, their values should be the same, although there may be a small but generally negligible difference.
3. We will see in Chapters 7 and 8 that the concept of small-signal resistance $r_d$ in Eq. (4.28) can be applied to model the small-signal behavior of transistors.

**EXAMPLE 4.9**

**Small-signal analysis of a diode circuit** The diode circuit shown in Fig. 4.18 has $V_S = 10$ V, $V_m = 50$ mV, and $R_L = 1$ kΩ. Use the $Q$-point found in Example 4.7 by mathematical method to determine the instantaneous diode voltage $v_D$. Assume an emission coefficient of $n = 1.84$.

**SOLUTION**

$V_T = 25.8$ mV, $n = 1.84$, $V_S = 10$ V, and $R_L = 1$ kΩ. The iterations of the $Q$-point analysis in Example 4.7 gave $V_D = 0.7148$ V and $I_D = 9.284$ mA. Using Eq. (4.27), we can find the AC resistance $r_d$ from

$$r_d = \frac{nV_T}{I_D} = \frac{1.84 \times 25.8 \times 10^{-3}}{(9.284 \times 10^{-3})} = 5.11 \, \Omega$$

The AC equivalent circuit is shown in Fig. 4.19. From the voltage divider rule, the AC diode voltage $v_d$ is given by

$$v_d = \frac{r_d}{r_d + R_L}v_m \sin \omega t$$

$$= \frac{5.11}{5.11 + 1 \, \text{k}\Omega} \times 50 \times 10^{-3} \sin \omega t = 0.2542 \times 10^{-3} \sin \omega t$$
Therefore, the instantaneous diode voltage $v_D$ is the sum of $V_D$ and $v_d$. That is,

$$v_D = V_D + v_d$$

$$= 0.7158 + 0.2542 \times 10^{-3} \sin \omega t \text{ V}$$

**EXAMPLE 4.10**

Finding the Q-point of a diode circuit and the diode model parameters from tabular data The diode circuit shown in Fig. 4.10 has $V_S = 15 \text{ V}$ and $R_L = 250 \Omega$. The diode forward characteristic, which can be obtained either from practical measurement or from the manufacturer’s data sheet, is given by the following table:

<table>
<thead>
<tr>
<th>$i_D$ (mA)</th>
<th>0</th>
<th>10</th>
<th>20</th>
<th>30</th>
<th>40</th>
<th>50</th>
<th>60</th>
<th>70</th>
</tr>
</thead>
<tbody>
<tr>
<td>$v_D$ (V)</td>
<td>0.5</td>
<td>0.87</td>
<td>0.98</td>
<td>1.058</td>
<td>1.115</td>
<td>1.173</td>
<td>1.212</td>
<td>1.25</td>
</tr>
</tbody>
</table>

Determine (a) the Q-point ($V_D$, $I_D$), (b) the parameters ($V_{TD}$ and $R_D$) of the piecewise linear DC model, and (c) the small-signal AC resistance $r_d$. Assume that the emission coefficient is $n = 1$ and that $V_T = 25.8 \text{ mV}$.

**SOLUTION**

$V_S = 15 \text{ V}$ and $R_L = 250 \Omega$.

(a) From Eq. (4.16), the load line is described by

$$i_D = \frac{(V_S - v_D)}{R_L}$$

The diode characteristic is defined by a table of data. The Q-point can be determined from the load line and the data table by an iterative method, as discussed in Sec. 4.5.3.

**Iteration 1:** Assume $v_D = 0.7 \text{ V}$. From Eq. (4.16),

$$i_D = \frac{(V_S - v_D)}{R_L} = \frac{(15 - 0.7)}{250} = 57.2 \text{ mA}$$

which lies between 50 mA and 60 mA in the table. Thus, we can see from the table of data that the new value of diode drop $v_D^{(\text{new})}$ lies between 1.173 V and 1.212 V. Let us assume that the diode voltage $v_D(k)$ corresponds...
to the diode current $i_D(k)$ and the diode voltage $v_D(k + 1)$ corresponds to the diode current $i_D(k + 1)$. This is shown in Fig. 4.20. If $i_D$ lies between $i_D(k)$ and $i_D(k + 1)$, then the corresponding value of $v_D$ will lie between $v_D(k)$ and $v_D(k + 1)$. Thus, $v_D^{(\text{new})}$ can be found approximately by linear interpolation from

$$v_D^{(\text{new})} = v_D(k) + \frac{v_D(k + 1) - v_D(k)}{i_D(k + 1) - i_D(k)} [i_D - i_D(k)]$$

$$= 1.173 + \frac{1.212 - 1.173}{60 \text{ mA} - 50 \text{ mA}} (57.2 \text{ mA} - 50 \text{ mA}) = 1.201 \text{ V}$$

**Iteration 2:** Use the value of $v_D$ from the previous iteration; that is, set $v_D = v_D^{(\text{new})} = 1.201 \text{ V}$. From Eq. (4.16),

$$i_D = i_D^{(\text{new})} = \frac{(15 - 1.201)}{250} = 55.2 \text{ mA}$$

From Eq. (4.34), the new value of $v_D$ is

$$v_D^{(\text{new})} = 1.173 + \frac{1.212 - 1.173}{60 \text{ mA} - 50 \text{ mA}} (55.2 \text{ mA} - 50 \text{ mA}) = 1.193 \text{ V}$$

This process is repeated until a stable $Q$-point is found. After two iterations, we have $V_D = v_D^{(\text{new})} = 1.193 \text{ V}$ and $I_D = i_D^{(\text{new})} = 55.2 \text{ mA}$.

**b** Since $R_D$ is the slope of the tangent at the $Q$-point, we get

$$R_D = \frac{\Delta v_D}{\Delta i_D} \mid_{\text{at } Q\text{-point}} = \frac{v_D(k)}{i_D(k + 1) - i_D(k)} = \frac{v_D(k + 1) - v_D}{i_D(k + 1) - i_D}$$

$$= \frac{1.193 - 1.173}{55.2 - 50} = 3.9 \Omega$$

(4.35)

The diode threshold voltage is

$$V_{TD} = V_D - R_D I_D = 1.193 - 3.9 \times 55.2 \text{ mV} = 0.98 \text{ V}$$

**c** From Eq. (4.27), the small-signal AC resistance $r_d$ is

$$r_d = \frac{nV_I}{I_D} = \frac{1 \times 25.8 \text{ mV}}{55.2 \times 10^{-3}} = 0.5 \Omega$$

**d** Since $R_D$ is the slope of the tangent at the $Q$-point, we get

$$R_D = \frac{\Delta v_D}{\Delta i_D} \mid_{\text{at } Q\text{-point}} = \frac{v_D(k) - v_D(k + 1)}{i_D(k + 1) - i_D(k)}$$

$$= \frac{1.193 - 1.173}{55.2 - 50} = 3.9 \Omega$$

(4.35)

The diode threshold voltage is

$$V_{TD} = V_D - R_D I_D = 1.193 - 3.9 \times 55.2 \text{ mV} = 0.98 \text{ V}$$

**e** From Eq. (4.27), the small-signal AC resistance $r_d$ is

$$r_d = \frac{nV_I}{I_D} = \frac{1 \times 25.8 \text{ mV}}{55.2 \times 10^{-3}} = 0.5 \Omega$$

![Figure 4.20](image-url)
4.6.4 PSpice/SPICE Diode Model

PSpice/SPICE uses a voltage-dependent current source, as shown in Fig. 4.21(a). \( r_s \) is the series resistance, known as the bulk (or parasitic) resistance. It is due to the resistance of the semiconductor and is dependent on the amount of doping. It should be noted that Fig. 4.21(a) is a nonlinear diode model, whereas the constant-drop DC model, the piecewise linear DC model, and the low-frequency AC model are linear or piecewise linear models.

At first PSpice/SPICE finds the DC biasing point and then calculates the parameter of the small-signal model shown in Fig. 4.21(b). \( C_j \) is a nonlinear function of the diode voltage \( v_D \), and its value equals \( C_j = dq_j/dv_D \), where \( q_j \) is the depletion layer charge. (The junction capacitances and the high-frequency model are discussed in Sec. 6.8.) PSpice/SPICE generates the small-signal parameters from the operating point and adjusts the values of \( r_d \) and \( C_j \) for the forward or reverse condition. The diode characteristic can be described in PSpice/SPICE in either a model statement or a tabular representation.

Model Statement

The PSpice/SPICE model statement of a diode has the general form

\[
\text{.MODEL DNAME D (P1=A1 P2=A2 P3=A3 \ldots \ldots \ldots \ldots P\textit{N}=\textit{AN})}
\]

where DNAME is the model name, which can begin with any character but is normally limited to eight characters. D is the type symbol for diodes. P1, P2, \ldots and A1, A2, \ldots are the model parameters and their values, respectively. The model parameters can be found in the PSpice/SPICE library file or can be determined from the data sheet [8, 9]. For example, a typical statement for diode D1N4148 is as follows:

\[
\text{.MODEL D1N4148 D(IS=2.682N N=1.836 RS=.5664 IKF=44.17M XTI=3 EG=1.11}
\]

\[+\text{ CJO=4P M=.3333 VJ=.5 FC=.5 ISR=1.565N NR=2 BV=100 IBV=100U TT=11.54N)\]

![FIGURE 4.21 PSpice/SPICE diode model](a) Large-signal model (b) Small-signal model

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
Tabular Representation

The TABLE representation is available only in PSpice. It allows the $v$-$i$ characteristic to be described, and it has the general form

$$E<\text{name}> \ N+ \ N- \ \text{TABLE} \ \{<\text{expression}>\} = <<(\text{input}) \ \text{value}>, \ (\text{output}) \ \text{value}>>$$

$E<\text{name}>$ is the name of a voltage-controlled voltage source, and $N+$ and $N-$ are the positive and negative nodes of the voltage source, respectively. The keyword TABLE indicates that the relation is described by a table of data. The table consists of pairs of values: $(\text{input}) \ \text{value}$ and $(\text{output}) \ \text{value}$. The first value in a pair is the input, and the second value is the corresponding output. The $<\text{expression}>$ is the input value and is used to find the corresponding output from the lookup table. If an input value falls between two entries, the output is found by linear interpolation. If the input falls outside the table’s range, the output is assumed to remain constant at the value corresponding to the smallest or the largest input.

The diode characteristic is represented by a current-controlled voltage source—say ED. That is, the diode is replaced by a voltage source of ED in series with a dummy voltage source VX of 0. VX acts as an ammeter and measures the diode current. This is shown in Fig. 4.22. ED is related to $I_D$ [i.e., $I(VX)$] by a table. The PSpice representation for the diode characteristic in Example 4.10 is shown here:

$$VX \ 2 \ 3 \ \text{DC} \ 0 V \ ; \ \text{measures the diode current ID}$$

$$ED \ 1 \ 2 \ \text{TABLE} \ \{I(VX)\} = (0, 0.5) \ (10m, 0.87) \ (20m, 0.98) \ (30m, 1.058)$$
$$+ (40m, 1.15) \ (50m, 1.173) \ (60m, 1.212) \ (70m, 1.25) \ (80m, 1.5) \ (300m, 3.0)$$

EXAMPLE 4.11

PSpice/SPICE diode model and analysis The diode circuit shown in Fig. 4.18 has $V_S = 10 V$, $V_m = 50 mV$ at $1 kHz$, $R_L = 1 k\Omega$, and $V_T = 25.8 mV$. Assume an emission coefficient of $n = 1.84$.

(a) Use PSpice/SPICE to generate the $Q$-point and the small-signal parameters and to plot the instantaneous output voltage $v_O = v_D$.

(b) Compare the results with those of Example 4.9. Assume model parameters of diode D1N4148:

$$I_S = 2.682 N \ CJO = 4 P \ M = .3333 \ VJ = .5 \ BV = 100 \ IBV = 100 U \ TT = 11.54 N$$

SOLUTION

$$V_S = 10 V, \ V_m = 50 mV, \ \text{and} \ R_L = 1 k\Omega$$

(a) From Example 4.7, the $Q$-point values are $V_D = 0.7148 V$ and $I_D = 9.284 mA$. The diode circuit for PSpice simulation is shown in Fig. 4.23. PSpice simulation gives the following biasing point and small-signal parameters:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$I_D$</td>
<td>9.28E-03 mA</td>
</tr>
<tr>
<td>$V_D$</td>
<td>7.18E-01 V</td>
</tr>
<tr>
<td>$V_T$</td>
<td>718 mV</td>
</tr>
<tr>
<td>$r_d$</td>
<td>5.53 Ω</td>
</tr>
<tr>
<td>$C_j$</td>
<td>2.10E-09 nF</td>
</tr>
</tbody>
</table>
The PSpice plot of the transient response is shown in Fig. 4.24, which gives $V_D = 718.35$ mV and $v_{d(\text{peak})} = v_{o(\text{peak})} = 600.52 \ \mu\text{V}/2 = 300.3 \ \mu\text{V}$. Thus,

$$v_d = 300.3 \times 10^{-6} \sin \omega t \quad \text{and} \quad v_{d(\text{peak})} = 300.3 \ \mu\text{V}$$

(b) Example 4.9 gives $V_D = 0.7148$ V, $I_D = 9.284$ mA, $r_d = 5.11 \ \Omega$, $v_d = 254.2 \times 10^{-6} \sin \omega t$, and $v_{d(\text{peak})} = 254.2 \ \mu\text{V}$, which agree closely with the PSpice results.

---

**FIGURE 4.23** Diode circuit for PSpice simulation

**FIGURE 4.24** PSpice plot for Example 4.11

---

**KEY POINTS OF SECTION 4.6**

- The constant-drop DC model assumes a fixed voltage drop of the diode. It gives a quick but approximate result. It is best suited for finding the approximate behavior of a circuit, especially at the initial design stage.
- The piecewise linear DC model breaks the nonlinear diode characteristic into two parts: a fixed DC voltage and a current-dependent voltage drop across a fixed resistance. The resistance is determined by drawing a best-fit line through the estimated $Q$-point on the current-dependent part. This model is commonly used for the analysis of diode circuits, and it gives reasonable results for most applications.
- The low-frequency AC model represents the behavior of a diode in response to a variation of the $Q$-point caused by a small signal. It is modeled by a small-signal resistance drawn as a tangent at the $Q$-point and is dependent on the diode current. The resistance can be approximated by that of the piecewise linear model. Thus this model can be regarded as an extension of the piecewise linear DC model.
- The high-frequency AC model represents the frequency response of the diode by incorporating two junction capacitances (diffusion and depletion layer) into the low-frequency AC model. The depletion layer capacitance is dependent on the diode reverse voltage. But the diffusion capacitance is directly proportional to the diode current and is present only in the forward direction.
- PSpice/SPICE generates a complex but accurate model. However, it is necessary to define the PSpice/SPICE model parameters, which can be obtained from the PSpice/SPICE library or from the manufacturer. These parameters can also be determined from the diode characteristic.
4.7 Zener Diodes

If the reverse voltage of a diode exceeds a specific voltage called the breakdown voltage, the diode will operate in the breakdown region. In this region the reverse diode current increases rapidly. The diode voltage remains almost constant and is practically independent of the diode current. However, operation in the breakdown region will not be destructive if the diode current is limited to a safe value by an external circuitry so that the power dissipation within the diode is within permissible limits specified by the manufacturer and the diode does not overheat.

A diode especially designed to have a steep characteristic in the breakdown region is called a zener diode. The symbol for a zener diode is shown in Fig. 4.25(a), and its v-i characteristic appears in Fig. 4.25(b).

\[
V_{ZK} \quad \text{is the knee voltage, and} \quad I_{ZK} \quad \text{is its corresponding current. A zener diode is specified by its breakdown voltage, called the zener voltage (or reference voltage) } V_Z, \quad \text{at a specified test current} \quad I_{ZT} \quad \text{.} \quad I_{Z,max} \quad \text{is the maximum current that the zener diode can withstand and still remain within permissible limits for power dissipation.} \quad I_{Z(min)} \quad \text{is the minimum current, slightly below the knee of the characteristic curve, at which the diode exhibits the reverse breakdown. That is,} \quad I_{Z(min)} \approx I_{ZK}. \]

The forward and reverse characteristics of a zener diode are represented by an arrow symbol. The arrow points toward the positive current \( i_D \). In the forward direction, the zener diode behaves like a normal diode; its equivalent circuit is shown in Fig. 4.25(c). In the reverse direction, it offers a very high resistance, acting like a normal reverse-biased diode if \( |v_D| < V_Z \) and like a low-resistance diode if \( |v_D| > V_Z \). For example, let us consider a zener diode with a nominal voltage \( V_Z = 5 \text{ V} \pm 2 \text{ V} \). For \( 3 \text{ V} < |v_D| < 5 \text{ V} \) in the reverse direction, the diode will normally exhibit a zener effect. For \( 5 \text{ V} < V_Z < 7 \text{ V} \), the breakdown could be due to the zener effect, the avalanche effect, or a combination of the two.

The reverse (zener) characteristic of Fig. 4.25(b) can be approximated by a piecewise linear model with a fixed voltage \( V_{ZO} \) and an ideal diode in series with resistance \( R_Z \). The equivalent circuit of the zener action is shown in Fig. 4.25(d) for \( |v_D| > V_Z \). \( R_Z \) depends on the inverse slope of the zener characteristic and is defined as

\[
R_Z = \frac{\Delta V_Z}{\Delta i_Z} \text{ at } v_Z = \frac{\Delta v_D}{\Delta i_D} \text{ for } v_D < 0 \text{ and } i_D < 0 \quad (4.36)
\]
\( R_Z \) is also called the zener resistance. The value of \( R_Z \) remains almost constant over a wide range of the zener characteristic. However, its value changes rapidly in the vicinity of the knee point. Thus, a zener diode should be operated away from the knee point. The typical value of \( R_Z \) is a few tens of ohms, but it increases with current \( i_D \). At the knee point of the zener characteristic, \( R_Z \) has a high value, typically 3 k\( \Omega \). The zener current \( i_Z (= -i_D) \) can be related to \( V_{ZO} \) and \( R_Z \) by
\[
\Delta v_Z = \Delta V_{ZO} - R_Z \Delta i_Z
\]

### 4.7.1 Zener Regulator

A zener diode may be regarded as offering a variable resistance whose value changes with the current so that the voltage drop across the terminals remains constant. Therefore, it is also known as a voltage reference diode. The value of \( R_Z \) is very small. Thus, the zener voltage \( v_Z \) is almost independent of the reverse diode current \( i_D \). Because of the constant voltage characteristic in the breakdown region, a zener diode can be employed as a voltage regulator. A regulator maintains an almost constant output voltage even though the DC supply voltage and the load current may vary over a wide range. A zener voltage regulator is shown in Fig. 4.26(a). A zener voltage regulator is also known as a shunt regulator because the zener diode is connected in shunt (or parallel) with the load \( R_L \). The value of current-limiting resistance \( R_s \) should be such that the diode can operate in the breakdown region over the entire range of input voltages \( v_S \) and variations of the load current \( i_L \).

If the zener diode is replaced by its piecewise linear model with \( V_{ZO} \) and \( R_Z \), the equivalent circuit shown in Fig. 4.26(b) is created. If the supply voltage \( v_S \) varies, then the zener current \( i_Z \) will vary because of the presence of \( R_Z \), thereby causing a variation of the output voltage. This variation of the output voltage is defined by a factor called the line regulation, which is related to \( R_s \) and \( R_Z \):
\[
\text{Line regulation} = \frac{\Delta v_O}{\Delta v_S} = \frac{R_Z}{R_Z + R_s}
\]

If the load current \( i_L \) increases, then the zener current \( i_Z \) will decrease because of the presence of \( R_Z \), thereby causing a decrease of the output voltage. This variation of the output voltage is defined by a factor called the load regulation, which is related to \( R_s \) and \( R_Z \):
\[
\text{Load regulation} = \frac{\Delta v_O}{\Delta i_L} = -(R_Z \parallel R_s)
\]
Any change in the zener voltage $V_{ZO}$ will increase the output voltage. The variation of the output voltage is defined by a factor called the zener regulation, which is related to $R_s$ and $R_Z$:

$$Zener\ regulation = \frac{\Delta V_O}{\Delta V_{ZO}} = \frac{R_s}{R_Z + R_s}$$

Thus, applying the superposition theorem, we can find the effective output voltage $v_O$ of the regulator in Fig. 4.26(b) as follows:

$$v_O = \frac{\Delta v_O}{\Delta V_{ZO}} \Delta V_{ZO} + \frac{\Delta v_O}{\Delta v_S} \Delta v_S + \frac{\Delta v_O}{\Delta i_L} \Delta i_L$$

$$= \frac{R_s}{R_Z + R_s} \Delta V_{ZO} + \frac{R_Z}{R_Z + R_s} \Delta v_S - (R_Z \parallel R_s) \Delta i_L$$

(4.40)

**EXAMPLE 4.12**

**Design of a zener regulator** The parameters of the zener diode for the voltage regulator circuit of Fig. 4.26(a) are $V_Z = 4.7$ V at test current $I_{ZT} = 53$ mA, $R_Z = 8$ Ω, and $R_{ZK} = 500$ Ω at $I_{ZK} = 1$ mA. The supply voltage is $v_S = V_S = 12 \pm 2$ V, and $R_s = 220$ Ω.

(a) Find the nominal value of the output voltage $v_O$ under no-load condition $R_L = \infty$.

(b) Find the maximum and minimum values of the output voltage for a load resistance of $R_L = 470$ Ω.

(c) Find the nominal value of the output voltage $v_O$ for a load resistance of $R_L = 100$ Ω.

(d) Find the minimum value of $R_L$ for which the zener diode operates in the breakdown region.

**SOLUTION**

Using Eq. (4.37), we have

$$V_{ZO} = V_Z - R_Z I_{ZT} = 4.7 \text{ V} - 8 \Omega \times 53 \text{ mA} = 4.28 \text{ V}$$

(a) For $R_L = \infty$, the zener current is

$$i_Z = \frac{V_S - V_{ZO}}{R_Z + R_s} = \frac{12 - 4.28}{8 + 220} = 33.86 \text{ mA}$$

The output voltage is

$$v_O = V_{ZO} + R_Z i_Z = 4.28 \text{ V} + 8 \Omega \times 33.86 \text{ mA} = 4.55 \text{ V}$$

(b) A change in the supply voltage by $\Delta v_S = \pm 2$ V will cause a change in the output voltage, which we can find from Eq. (4.38):

$$\Delta v_{O(supply)} = \frac{\Delta v_S R_Z}{R_Z + R_s} = \frac{\pm 2 \times 8}{8 + 220} = \pm 0.18 \text{ mV}$$

The nominal value of the load current is $i_L = V_Z/R_L = 4.7/470 = 10$ mA. A change in the load current by $\Delta i_L = 10$ mA will also cause a change in the output voltage, which we can find from Eq. (4.39):

$$\Delta v_{O(\text{load})} = -(R_Z \parallel R_s) i_L = -(8 \Omega \parallel 220 \Omega) \times 10 \text{ mA} = -77.19 \text{ mV}$$
4.7.2 Design of a Zener Regulator

If \( i_Z \) is the zener current and \( i_L \) is the load current, the value of resistance \( R_s \) can be found from

\[
R_s = \frac{V_S - V_{ZO} - R_Z i_Z}{i_Z + i_L} \quad \text{for} \quad V_S = V_S
\]  

To ensure that the zener diode operates in the breakdown region under the worst-case conditions, the regulator must be designed to do the following:

1. To ensure that the zener current will exceed \( i_{Z(\text{min})} \) when the supply voltage is minimum \( V_{S(\text{min})} \) and the load current is maximum \( i_{L(\text{max})} \). Applying Eq. (4.43), we can find \( R_s \) from

\[
R_s = \frac{V_{S(\text{min})} - (V_{ZO} + R_Z i_{Z(\text{min})})}{i_{Z(\text{min})} + i_{L(\text{max})}}
\]  

2. To ensure that the zener current will not exceed \( i_{Z(\text{max})} \) when the supply voltage is maximum \( V_{S(\text{max})} \) and the load current is minimum \( i_{L(\text{min})} \). Using Eq. (4.43), we can find \( R_s \) from

\[
R_s = \frac{V_{S(\text{max})} - (V_{ZO} + R_Z i_{Z(\text{min})})}{i_{Z(\text{max})} + i_{L(\text{min})}}
\]
Equating $R_s$ in Eq. (4.44) to $R_s$ in Eq. (4.45), we get the relationship of the maximum zener current in terms of the variations in $V_S$ and $i_L$. That is,

\[
(V_{S(\text{min})} - V_{ZO} - R_Z i_{Z(\text{min})})(i_{Z(\text{max})} + i_{L(\text{min})}) = (V_{S(\text{max})} - V_{ZO} - R_Z i_{Z(\text{max})})(i_{Z(\text{min})} + i_{L(\text{max})})
\]

As a rule of thumb, the minimum zener current $i_{Z(\text{min})}$ is normally limited to 10% of the maximum zener current $i_{Z(\text{max})}$ to ensure operation in the breakdown region. That is,

\[
i_{Z(\text{min})} = 0.1 \times i_{Z(\text{max})}
\]

**EXAMPLE 4.13**

**Design of a zener regulator** The parameters of a 6.3-V zener diode for the voltage regulator circuit of Fig. 4.26(a) are $V_Z = 6.3$ V at $i_{ZT} = 40$ mA and $R_Z = 2$ Ω. The supply voltage $v_S = V_S$ can vary between 12 V and 18 V. The minimum load current is 0 mA. The minimum zener diode current $i_{Z(\text{min})}$ is 1 mA. The power dissipation $P_{Z(\text{max})}$ of the zener diode must not exceed 750 mW at 25°C. Determine (a) the maximum permissible value of the zener current $i_{Z(\text{max})}$, (b) the value of $R_s$ that limits the zener current $i_{Z(\text{max})}$ to the value determined in part (a), (c) the power rating $P_R$ of $R_s$, and (d) the maximum load current $i_{L(\text{max})}$.

**SOLUTION**

$V_Z = 6.3$ V at $i_{ZT} = 40$ mA, $i_{L(\text{min})} = 0$ and $i_{Z(\text{min})} = 1$ mA. Using Eq. (4.37), we have

$V_{ZO} = V_Z - R_Z i_{ZT} = 6.3 - 2 \times 40$ mA = 6.22 V

(a) The maximum power dissipation $P_{Z(\text{max})}$ of a zener diode is

\[
P_{Z(\text{max})} = i_{Z(\text{max})} V_Z = 0.75 \text{ W}
\]

or

\[
i_{Z(\text{max})} = \frac{P_{Z(\text{max})}}{V_Z} = \frac{0.75}{6.3} = 119 \text{ mA}
\]

(b) The zener current $i_{Z}$ becomes maximum when the supply voltage is maximum and the load current is minimum—that is, $V_{S(\text{max})} = 18$ V, $i_{L(\text{min})} = 0$, and $i_{Z(\text{max})} = 119$ mA. From Eq. (4.45),

\[
R_s = \frac{V_{S(\text{max})} - V_{ZO} - R_Z i_{Z(\text{max})}}{i_{Z(\text{max})} + i_{L(\text{min})}} = \frac{18 \text{ V} - 6.22 \text{ V} - 2 \Omega \times 119 \text{ mA}}{119 \text{ mA} + 0} = 96.96 \text{ Ω}
\]

(c) The power rating $P_R$ of $R_s$ is

\[
P_R = (i_{Z(\text{max})} + i_{L(\text{min})})(V_{S(\text{max})} - V_{ZO} - R_Z i_{Z(\text{max})}) = 119 \text{ mA} \times (18 \text{ V} - 6.22 \text{ V} - 2 \Omega \times 119 \text{ mA}) = 1.373 \text{ W}
\]

The worst-case power rating of $R_s$ will occur when the load is shorted. That is,

\[
P_{R(\text{max})} = \frac{V_{S(\text{max})}^2}{R_s} = \frac{18^2}{96.99} = 3.34 \text{ W}
\]
(d) \( i_L \) must be maintained at the maximum when \( V_S \) is minimum and \( i_Z \) is minimum—that is, \( V_S(\text{min}) = 12 \) V and \( i_Z(\text{min}) = 1 \) mA. From Eq. (4.44), we get

\[
I_L(\text{max}) = \frac{V_S(\text{min}) - V_Z - R_Z i_Z(\text{min})}{R_s} - i_Z(\text{min}) = \frac{12 \text{ V} - 6.22 \text{ V} - 2 \Omega \times 1 \text{ mA}}{96.99 \Omega} - 1 \text{ mA} = 58.57 \text{ mA}
\]

**EXAMPLE 4.14**

**Design of a zener regulator and PSpice/SPICE verification** The parameters of the zener diode for the voltage regulator in Fig. 4.26(a) are \( V_Z = 4.7 \) V at \( i_Z(\text{T}) = 20 \) mA, \( R_Z = 19 \Omega \), \( i_ZK = 1 \) mA, and \( P_Z(\text{max}) = 400 \) mW at 4.7 V. The supply voltage \( V_S = V_S \) varies from 20 V to 30 V, and the load current \( i_L \) changes from 5 mA to 50 mA.

(a) Determine the value of resistance \( R_s \) and its power rating.

(b) Use PSpice/SPICE to check your results by plotting the output voltage \( v_O \) against the supply voltage \( v_S \).

Assume PSpice model parameters of zener diode D1N750:

\[
I_S=880.5E-18 \quad N=1 \quad CJO=175P \quad VJ=.75 \quad BV=4.7 \quad IBV=20.245M
\]

**SOLUTION**

\( V_Z = 4.7 \) V, \( P_Z(\text{max}) = 400 \) mW, \( i_L(\text{min}) = 5 \) mA, \( i_L(\text{max}) = 50 \) mA, \( V_S(\text{min}) = 20 \) V, and \( V_S(\text{max}) = 30 \) V. Using Eq. (4.37), we have

\[
V_{ZO} = V_Z - R_Z i_Z(\text{T}) = 4.7 - 19 \times 20 \text{ mA} = 4.32 \text{ V}
\]

Also, \( i_Z(\text{max}) = \frac{P_Z(\text{max})}{V_Z} = \frac{400 \text{ mW}}{4.7 \text{ V}} = 85.1 \text{ mA} \) (from specifications)

(a) Since the minimum value of the zener current is not specified, we can assume for all practical purposes that

\[
i_Z(\text{min}) = 0.1 \times i_Z(\text{max}) = 0.1 \times 85.1 \text{ mA} = 8.51 \text{ mA}
\]

From Eq. (4.44) and Fig. 4.26(b), we can find the value of \( R_s \):

\[
R_s = \frac{V_S(\text{min}) - V_{ZO} - R_Z i_Z(\text{min})}{i_Z(\text{min}) + i_L(\text{max})} = \frac{20 \text{ V} - 4.32 \text{ V} - 19 \Omega \times 8.51 \text{ mA}}{8.51 \text{ mA} + 50 \text{ mA}} = 265 \Omega
\]

From Eq. (4.45), we can find

\[
R_s(i_Z(\text{max}) + i_L(\text{min})) = V_S(\text{max}) - V_{ZO} - R_Z i_Z(\text{max})
\]

which can be solved to find the actual value of the maximum zener current \( i_Z(\text{max}) \):

\[
i_Z(\text{max}) = \frac{V_S(\text{max}) - V_{ZO} - R_s i_L(\text{min})}{R_s + R_Z} = \frac{30 \text{ V} - 4.32 \text{ V} - 265 \Omega \times 5 \text{ mA}}{(265 + 19) \Omega} = 85.76 \text{ mA}
\]
The power rating \( P_R \) of \( R_s \) is

\[
P_R = (i_{Z\text{max}} + i_{L\text{min}})(V_{S\text{max}} - V_Z)
\]

\[
= (85.76 \text{ mA} + 5 \text{ mA})(30 \text{ V} - 4.7 \text{ V}) = 2.3 \text{ W}
\]

The worst-case power rating will be

\[
P_{R\text{max}} = \frac{V_{S\text{max}}^2}{R_s} = \frac{30^2}{265} = 3.4 \text{ W}
\]

From \( i_{L\text{min}} = 5 \text{ mA} \) and \( i_{L\text{max}} = 50 \text{ mA} \), we find that the corresponding maximum and minimum values of the load resistance are

\[
R_{L\text{max}} = \frac{V_Z}{i_{L\text{max}}} = \frac{4.7 \text{ V}}{50 \text{ mA}} = 94 \Omega
\]

\[
R_{L\text{min}} = \frac{V_Z}{i_{L\text{min}}} = \frac{4.7 \text{ V}}{5 \text{ mA}} = 940 \Omega
\]

The zener voltage regulator for the PSpice simulation is shown in Fig. 4.27. The zener diode is normally modeled by setting the diode parameter \( BV \equiv V_Z \) in the PSpice model.

(b) The PSpice plot of the output voltage \( v_O \) against supply voltage \( v_S \) is shown in Fig. 4.28. The zener action begins at an output voltage of \( v_O = 4.74 \text{ V} \), which is close to the expected value of 4.7 V.

---

**4.7.3 Zener Limiters**

The zener characteristic shown in Fig. 4.25(b) can be approximated by the piecewise linear characteristic shown in Fig. 4.29(a). In the forward direction, a zener diode behaves like a normal diode, and it can be represented by a piecewise linear model with voltage \( V_{TD} \) and resistance \( R_D \). The model of a zener diode...
The current through a zener diode can be expressed as follows:

\[ i_D = \begin{cases} 
0 & \text{for } -V_{ZO} < v_D < V_{TD} \\
\frac{v_D}{R_D} - \frac{V_{TD}}{R_0} & \text{for } v_D \geq V_{TD} \\
\frac{v_D}{R_Z} + \frac{V_{ZO}}{R_Z} & \text{for } v_D \leq -V_{ZO} 
\end{cases} \]

The values of \( R_Z \) and \( R_D \) are very small, typically 20 \( \Omega \), and can be neglected for most analysis. The characteristic of Fig. 4.29(b) can be represented by the ideal zener characteristic shown in Fig. 4.29(c). Thus, a zener diode forms a natural limiter. By replacing the zener diode by its ideal characteristic (i.e., neglecting \( R_D \) and \( R_Z \)), we can simplify the circuit model of Fig. 4.29(b) to the circuit shown in Fig. 4.30(a). For a positive supply voltage \( v_S \geq V_{TD} \), the output voltage \( v_O \) will be limited to \( V_{TD} \). However, a negative input supply \( v_S \leq -V_{ZO} \) will limit the output voltage \( v_O \) to \( -V_{ZO} \). The approximate transfer characteristic of a zener limiter is shown in Fig. 4.30(b). This is an unsymmetrical limiter.

A symmetrical limiter can be obtained by connecting two zener diodes in series such that one diode opposes the other, as shown in Fig. 4.31(a). By replacing each zener diode by its model, shown in Fig. 4.29(b), we can create the equivalent circuit of a zener limiter, as shown in Fig. 4.31(b). If \( v_S > (V_{TD} + V_{ZO}) \), ideal diodes \( D_2 \) and \( D_3 \) behave as short circuits and can be replaced by an equivalent...
single diode in series with a voltage $V_{TD} + V_Z$ and a resistance $R_D + R_Z$. Similarly, when $v_S < -(V_{TD} + V_Z)$, diodes $D_1$ and $D_4$ can be replaced by a diode in series with a voltage $V_{TD} + V_Z$ and a resistance $R_D + R_Z$. This arrangement is shown in Fig. 4.31(c). If we assume an ideal zener diode such that the values of $R_D$ and $R_Z$ are negligible, Fig. 4.31(c) can be reduced to Fig. 4.31(d). The transfer characteristic ($v_O$ versus $v_S$) of a symmetrical zener limiter is shown in Fig. 4.31(e).

**EXAMPLE 4.15**

*Small-signal analysis of a zener limiter and PSpice/SPICE verification* The parameters of the zener diodes in the symmetrical zener limiter of Fig. 4.31(a) are $R_D = 50 \, \Omega$, $V_{TD} = 0.7 \, V$, $R_Z = 20 \, \Omega$, and $V_Z = 4.7 \, V$ at $I_{ZT} = 20 \, mA$. The value of current-limiting resistance $R_s$ is $1 \, k\Omega$. The input voltage to the limiter is AC rather than DC and is given by $v_S = v_s = 15 \sin (2000 \pi t)$. 

---

*FIGURE 4.31* Symmetrical zener limiter
(a) Determine the instantaneous output voltage \( v_O \) and the peak zener diode current \( I_p (\text{diode}) \).

(b) Use PSpice/SPICE to plot the instantaneous output voltage \( v_O \). Assume PSpice/SPICE model parameters of zener diode D1N750:

\[
\begin{align*}
 IS &= 880.5 \times 10^{-18} \\
 N &= 1 \\
 CJO &= 175 \times 10^{-12} \\
 VJ &= 0.7 \\
 BV &= 4.7 \\
 IBV &= 20.245 \times 10^6
\end{align*}
\]

**SOLUTION**

(a) \( R_D = 50 \Omega, V_{TD} = 0.7 \, \text{V}, R_Z = 20 \, \Omega, V_Z = 4.7 \, \text{V}, R_s = 1 \, \text{k}\Omega, \) and \( v_s = 15 \sin (2000\pi t) \). Using Eq. (4.37), we have

\[
V_{ZO} = V_Z - R_Z I_ZT = 4.7 \, \text{V} - 20 \, \Omega \times 20 \, \text{mA} = 4.3 \, \text{V}
\]

There are four possible intervals, depending on the value of \( v_s \).

If \( 15 \sin 2000\pi t = V_{ZO} + V_{TD} = 5 \), then

\[
2000\pi t = \sin^{-1} \left( \frac{5}{15} \right) = 0.34 \, \text{rad}
\]

**Interval 1:** This interval is valid for \( 0 \leq v_s \leq (V_{ZO} + V_{TD}) \).

\[
i_D = 0
\]

\[
v_O = v_s = 15 \sin (2000\pi t) \quad \text{for} \quad 0 \leq 2000\pi t \leq 0.34 \quad \text{and} \quad (\pi - 0.34) \leq 2000\pi t \leq \pi
\]

**Interval 2:** This interval is valid for \( v_s \geq (V_{ZO} + V_{TD}) \). From Fig. 4.31(c), we can find the instantaneous diode current \( i_D \):

\[
i_D = \frac{v_s}{R_s + R_D + R_Z} - \frac{V_{ZO} + V_{TD}}{R_s + R_D + R_Z}
\]

\[
= \frac{15 \sin (2000\pi t)}{1 \, \text{k}\Omega + 50 \, \Omega + 20 \, \Omega} - \frac{(4.3 + 0.7) \, \text{V}}{1 \, \text{k}\Omega + 50 \, \Omega + 20 \, \Omega} = [14.02 \sin (2000\pi t) - 4.67] \, \text{mA}
\]

The instantaneous output voltage \( v_O \) is given by

\[
v_O = V_{ZO} + V_{TD} + (R_D + R_Z) i_D
\]

Substituting for \( i_D \), we get

\[
v_O = (4.3 + 0.7) + (50 + 20) \times [14.02 \sin (2000\pi t) - 4.67] \times 10^{-3}
\]

\[
= 4.67 + 0.981 \sin (2000\pi t) \quad \text{for} \quad 0.34 \leq 2000\pi t \leq (\pi - 0.34)
\]

**Interval 3:** This interval is valid for \( 0 \leq v_s \leq -(V_{ZO} + V_{TD}) \).

\[
i_D = 0
\]

\[
v_O = v_s = -15 \sin (2000\pi t) \quad \text{for} \quad -0.34 \leq 2000\pi t \leq 0 \quad \text{and} \quad -\pi \leq 2000\pi t \leq (\pi + 0.34)
\]
4.7.4 Temperature Effects on Zener Diodes

Any change in junction temperature generally changes the zener voltage $V_Z$. The temperature coefficient is approximately $+2 \text{mV/}^\circ\text{C}$, which is the same as but opposite that of a forward-biased diode. However, if a zener diode is connected in series with a forward-biased diode, as shown in Fig. 4.34, the temperature coefficients of the two diodes tend to cancel each other. This cancellation greatly reduces the overall temperature coefficients, and the effect of temperature changes is minimized.

**Interval 4:** This interval is valid for $v_s \leq -(V_{ZO} + V_{TD})$.

\[
\begin{align*}
  i_D &= -[14.02 \sin (2000\pi t) - 4.67] \text{ mA} \\
  v_O &= 4.67 - 0.981 \sin (2000\pi t) \quad \text{for } (-\pi + 0.34) \leq 2000\pi t \leq -0.34
\end{align*}
\]

The peak diode current $i_{p(diode)}$ occurs at $2000\pi t = \pi/2$. That is,

\[
i_{p(diode)} = 14.02 \sin \left(\frac{\pi}{2}\right) - 4.67 \text{ mA} = 14.02 \text{ mA} - 4.67 \text{ mA} = 9.35 \text{ mA}
\]

(b) The symmetrical zener limiter for PSpice simulation is shown in Fig. 4.32. The PSpice plot of instantaneous output voltage $v_O$ is shown in Fig. 4.33, which gives $+5.435 \text{ V}$, compared to the expected value of $4.67 + 0.981 = 5.65 \text{ V}$ (from the expression of $v_O$ for the interval 2).

![Symmetrical zener limiter for PSpice simulation](image1)

![PSpice plots for Example 4.15](image2)

**FIGURE 4.32** Symmetrical zener limiter for PSpice simulation

**FIGURE 4.33** PSpice plots for Example 4.15

4.7.4 Temperature Effects on Zener Diodes

Any change in junction temperature generally changes the zener voltage $V_Z$. The temperature coefficient is approximately $+2 \text{mV/}^\circ\text{C}$, which is the same as but opposite that of a forward-biased diode. However, if a zener diode is connected in series with a forward-biased diode, as shown in Fig. 4.34, the temperature coefficients of the two diodes tend to cancel each other. This cancellation greatly reduces the overall temperature coefficients, and the effect of temperature changes is minimized.

![Zener diode in series with a forward-biased diode](image3)

**FIGURE 4.34** Zener diode in series with a forward-biased diode
EXAMPLE 4.16

Finding the temperature effect of a zener regulator with PSpice/SPICE  The zener voltage of the regulator in Fig. 4.26(a) is $V_Z = 4.7$ V. The current-limiting resistance $R_s$ is 1 kΩ, and the load resistance $R_L$ is very large, tending to infinity. The supply voltage $v_S$ varies from 0 to 20 V. Use PSpice/SPICE to plot the output voltage $v_O$ against the input voltage $v_S$ for junction temperatures $T_j = 25^\circ$C and $T_j = 100^\circ$C. Assume PSpice/SPICE model parameters of zener diode D1N750:

\[ IS=880.5E-18 \quad N=1 \quad CJO=175P \quad VJ=.75 \quad BV=4.7 \quad IBV=20.245M \]

SOLUTION

The zener diode regulator for PSpice simulation is shown in Fig. 4.35. The PSpice plots of the output voltage $v_O$ against the supply voltage $v_S$ are shown in Fig. 4.36, which shows that the junction temperature affects the zener voltage slightly. For example, at $v_S = 5$ V, $v_O = 4.2336$ V at $25^\circ$C and $4.2285$ V at $100^\circ$C.

FIGURE 4.35 Zener diode regulator for PSpice simulation

FIGURE 4.36 PSpice plots for Example 4.16

KEY POINTS OF SECTION 4.7

- A zener diode behaves like a normal diode in the forward direction. In the reverse direction, it maintains an almost constant voltage under varied load conditions if its voltage is greater than the zener voltage.
- A practical zener diode has a finite zener resistance, and the zener voltage will vary slightly with the zener current.
- Any change in junction temperature generally causes a change in the zener voltage.
4.8 Light-Emitting Diodes

A light-emitting diode (LED) is a special type of semiconductor diode that emits light when it is forward biased. The light intensity is approximately proportional to the forward diode current $i_D$. Light-emitting diodes are normally used in low-cost applications such as calculators, cameras, appliances, and automobile instrument panels.

4.9 Power Rating

Under normal operation, the junction temperature of a diode will rise as a result of power dissipation. Semiconductor materials have low melting points. The junction temperature, which is specified by the manufacturer, is normally limited to a safe value in the range of 150°C–200°C for silicon diodes and in the range of 60–110°C for germanium diodes. The power dissipation of a diode can be found from

$$ P_D = I_D V_D $$

(4.50)

The power dissipation of a small-signal diode is low (on the order of milliwatts), and the junction temperature does not normally rise above the maximum permissible value specified by the manufacturer. However, power diodes are normally mounted on a heat sink. The function of the heat sink is to dissipate heat on the ambient (i.e., the material surrounding the device) in order to keep the junction temperature of power diodes below the maximum permissible value. The steady-state rise in the junction temperature with respect to the ambient temperature has been found, by experiment, to be proportional to the power dissipation. That is,

$$ \Delta T = T_j - T_a = \theta_{ja} P_D $$

(4.51)

where $T_j =$ junction temperature, in °C

$T_a =$ ambient temperature, in °C

$\theta_{ja} =$ thermal resistance from junction to ambient, in °C/W

If the power dissipation $P_D$ exceeds the maximum permissible value, the junction temperature will rise above the maximum allowable temperature. Excessive power dissipation can damage a diode. The permissible junction power dissipation $P_D$ can be found by rearranging Eq. (4.51) to give

$$ P_D = \frac{-T_a}{\theta_{ja}} + \frac{T_j}{\theta_{ja}} $$

(4.52)

which indicates that the permissible power dissipation will increase if the ambient temperature $T_a$ can be reduced below the normal temperature of 25°C. However, in practice, power dissipation is limited to the value that corresponds to the permissible diode current. This limiting power $P_{Dm}$ corresponds to the value of $P_D$ at $T_j = 25°C$ and is specified by the manufacturer. Thus,

$$ P_D = \begin{cases} 
P_{Dm} \text{ (at } T_j = 25°C) & \text{for } T_a \leq 25°C \\
-\frac{T_a}{\theta_{ja}} + \frac{T_{jm}}{\theta_{ja}} & \text{for } T_a > 25°C 
\end{cases} $$

(4.53)

where $T_{jm} =$ maximum junction temperature.
As the junction temperature increases, the permissible power dissipation is reduced. A power derating curve is given by the manufacturer. The derating curve indicates the required adjustment in the power as the junction temperature increases above a specified temperature. A typical power dissipation–temperature derating curve is shown in Fig. 4.37. In the absence of such a characteristic, the values of $T_{jm}$ and $P_{Dm}(T_a = 25°C)$ are usually provided.

**KEY POINTS OF SECTION 4.9**

- Power and temperature ratings are important parameters of a diode, and they are related to each other.
- The maximum power rating of a diode is specified at an ambient temperature. The diode must be derated if the operating ambient temperature is above this specified value. The diode can handle higher power if the operating ambient temperature is below the specified temperature.

**EXAMPLE 4.17**

**Finding the power dissipation of a diode** A diode is operated at a $Q$-point of $V_D = 0.7$ V and $I_D = 1$ A. The diode parameters are $P_D = 1$ W at $T_a = 50°C$ and $P_{derating} = 6.67$ mW/°C. The ambient temperature is $T_a = 25°C$, and the maximum permissible junction temperature is $T_{jm} = 200°C$. Calculate (a) the junction temperature $T_j$, (b) the maximum permissible junction dissipation $P_{Dm}$, and (c) the permissible junction dissipation $P_D$ at an ambient temperature of $T_a = 75°C$.

**SOLUTION**

From Eq. (4.50), the junction power dissipation at the $Q$-point is

$$P_D = I_D V_D = 1 \text{ A} \times 0.7 \text{ V} = 0.7 \text{ W}$$

From Eq. (4.51), the thermal resistance from junction to ambient is

$$\theta_{ja} = \frac{\Delta T}{P_D} = \frac{T_{jm} - T_a}{P_D} = \frac{200 - 50}{1} = 150°C/\text{W}$$

(a) From Eq. (4.51), the junction temperature at the $Q$-point is

$$T_j = T_a + \theta_{ja} P_D = 25 + 150 \times 0.7 = 130°C$$
4.10 Diode Data Sheets

Diode ratings specify the current, voltage, and power-handling capabilities. This information is supplied by the manufacturer in data (or specifications) sheets. Typical data sheets for general-purpose diodes of types 1N4001 through 1N4007 are shown in Fig. 4.38. The important parameters of diodes of type 1N4001 are as follows:

1. Type of device with generic number or manufacturer’s part number: 1N4001.
2. Peak inverse voltage (or peak repetitive reverse voltage) \( P_{IV} \) = \( V_{RRM} \) = 50 V.
3. Operating and storage junction temperature range \( T_j \) = -65°C to +175°C.
4. Maximum reverse current \( I_R \) (at DC rated reverse voltage) at \( P_{IV} \) = 10 \( \mu \)A at \( T_j \) = 25°C and 50 \( \mu \)A at \( T_j \) = 100°C.
5. Maximum instantaneous forward voltage drop \( v_D \) = \( v_F \) = 1.1 V at \( T_j \) = 25°C.
6. Average rectified forward current \( I_{F(AV)} \) = 1 A at \( T_a \) = 75°C.
7. Repetitive peak current \( I_{FRM} \) is not quoted for 1N4001.
8. Nonrepetitive peak surge current \( I_{FSM} \) = 30 A for one cycle.
9. Average forward voltage drop \( V_{F(AV)} \) = \( V_D \) = 0.8 V.
10. DC power dissipation \( P_D \) = \( V_{F(AV)} I_{F(AV)} \) (not quoted for 1N4001).

Typical data sheets for zener diodes of types 1N4728A through 1N4764A are shown in Fig. 4.39. The important parameters of zener diodes of type 1N4732 are as follows:

1. Type of device with generic number or manufacturer’s part number: 1N4732.
2. Nominal zener voltage (avalanche breakdown voltage) \( V_Z \) = 4.7 V.
3. Operating and storage junction temperature range \( T_j \) = -65°C to +200°C.
4. Zener test current \( I_{ZT} \) = 53 mA.
5. Zener impedance \( Z_{ZT} \) = 8 \( \Omega \).
6. Knee current \( I_{ZK} \) = 1 mA.
7. Nonrepetitive peak surge current \( I_{FSM} \) = 970 A for one cycle.
8. DC power dissipation \( P_D \) = 1 W at \( T_a \) = 50°C.
9. Power derating curve: Above 50°C, \( P_D \) is derated by 6.67 mW/°C.

**NOTE** To allow a safety margin, designers should ensure that the operating values of voltage, current, and power dissipation are at least 20% to 30% less than the published maximum ratings. For military applications, the derating could be up to 50%.
Axial Lead Standard Recovery Rectifiers

This data sheet provides information on subminiature, axial lead mounted rectifiers for general-purpose low-power applications.

**MAXIMUM RATINGS**

<table>
<thead>
<tr>
<th>Rating</th>
<th>Symbol</th>
<th>V(DS)</th>
<th>I(T)</th>
<th>T(D)</th>
<th>T(RMS)</th>
<th>VRRM</th>
<th>VRRM</th>
<th>VRRM</th>
<th>VRRM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reverse Voltage</td>
<td>V(DS)</td>
<td>1500</td>
<td>500</td>
<td>250</td>
<td>200</td>
<td>400</td>
<td>500</td>
<td>1000</td>
<td>1500</td>
</tr>
<tr>
<td>Reverse Current</td>
<td>I(T)</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>Reverse Recovery Time</td>
<td>T(RMS)</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
</tr>
<tr>
<td>Temperature Range</td>
<td>VRRM</td>
<td>-40°C to +175°C</td>
<td>-40°C to +175°C</td>
<td>-40°C to +175°C</td>
<td>-40°C to +175°C</td>
<td>-40°C to +175°C</td>
<td>-40°C to +175°C</td>
<td>-40°C to +175°C</td>
<td>-40°C to +175°C</td>
</tr>
</tbody>
</table>

**ELECTRICAL CHARACTERISTICS**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Type</th>
<th>Max.</th>
</tr>
</thead>
<tbody>
<tr>
<td>VCEO</td>
<td>V</td>
<td>6.8 V</td>
</tr>
<tr>
<td>I</td>
<td>mA</td>
<td>10 mA</td>
</tr>
<tr>
<td>f</td>
<td>kHz</td>
<td>50 kHz</td>
</tr>
</tbody>
</table>

**Mechanical Characteristics**

- Case: Epoxy, Molded
- Weight: 0.4 gram (approximately)
- Finish: All External Surfaces Corrosion Resistant and Terminal Leads are Readily Solderable
- Lead and Mounting Surface Temperature for Soldering Purposes: 260°C Max. for 10 Seconds, 1/16 in. from case
- Polarity: Cathode Indicated by Polarity Band

**FIGURE 4.38** Data sheet for diodes (Copyright of ON Semiconductor. Used by permission.)
FIGURE 4.39  Data sheet for zener diodes (Copyright of Motorola. Used by permission.)
Updated information on the product can be found at www.onsemi.com
Semiconductor Diodes

1N4728, A thru 1N4764, A

**ELECTRICAL CHARACTERISTICS** (TA = 25°C unless otherwise noted) Vf = 1.2 V max., ic = 200 mA for all types.

<table>
<thead>
<tr>
<th>JEDEC Type No.</th>
<th>Nominal Zener Voltage VZ @ 1ZT (Volts Notes 2 and 3)</th>
<th>Test Current IZT mA</th>
<th>Maximum Zener Impedance (Note 4) Z2Z @ IZT Ohms</th>
<th>Leakage Current IR mA Max</th>
<th>Surge Current @ TA = 25°C iS-A = mA Volts</th>
</tr>
</thead>
<tbody>
<tr>
<td>1N4728</td>
<td>3.3</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4729</td>
<td>3.6</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4730</td>
<td>3.9</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4731</td>
<td>4.3</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4732</td>
<td>4.7</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4734</td>
<td>5.1</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4735</td>
<td>5.6</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4736</td>
<td>6.2</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4737</td>
<td>6.8</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4738</td>
<td>7.5</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4739</td>
<td>8.2</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4744</td>
<td>15.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4746</td>
<td>18.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4747</td>
<td>22.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4750</td>
<td>27.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4751</td>
<td>36.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4753</td>
<td>39.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4754</td>
<td>43.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4756</td>
<td>47.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4759</td>
<td>56.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4760</td>
<td>68.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4761</td>
<td>75.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4762</td>
<td>82.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4763</td>
<td>91.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1N4764</td>
<td>100.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>

* Indicates JEDEC Registered Data.

**NOTE 1 – Tolerance and Type Number Designation.** The JEDEC type numbers listed have a standard tolerance on the nominal zener voltage of ±10%. A standard tolerance of ±5% on individual units is also available and is indicated by suffixing “AT” to the standard type number.

**NOTE 2 – Specials Available Include:**
A. Nominal voltage ratings for the voltages shown and tighter voltage tolerances.
B. Matched sets.

For detailed information on price, availability, and delivery, contact your nearest Motorola representative.

**APPLICATION NOTE**

Since the actual voltage available from a given zener diode is temperature dependent, it is necessary to determine the junction temperature under any set of operating conditions in order to calculate its value. The following procedure is recommended: Lead Temperature, T L, should be determined from:

\[ T_L = \theta_{JA} PD + T_A \]

\[ \theta_{JA} \] is the lead-to-ambient thermal resistance (°C/W) and PD is the power dissipation. The value for \( \theta_{JA} \) will vary and depends on the device mounting method, \( \theta_{JA} \) is generally 30 to 40°C/W for the various clips and tie points in common use and for printed circuit board wiring.

The temperature of the lead can also be measured using a thermocouple placed on the lead as close as possible to the tie point. The thermal mass connected to the tie point is normally large enough so that it will not significantly respond to heat surges generated in the diode as a result of zener operation once steady-state conditions are achieved. Using the measured value of \( T_L \), the junction temperature may be determined by:

\[ T_J = T_L + \Delta T_J \]

\( \Delta T_J \) is the increase in junction temperature above the lead temperature and may be found as follows:

\[ \Delta T_J = \frac{\theta_{PD}}{R_D} \]

where \( \theta_{PD} \) may be determined from Figure 3 for dc power conditions.

For worst-case design, using expected limits of R \( \theta \), limits of PD and the extremes of T_J will be estimated. Changes in voltage, VZ, can then be found from:

\[ \Delta V = \theta_{ZT} \Delta T_J \]

\( \theta_{ZT} \) is the zener voltage temperature coefficient, is found from Figure 2.

Under high power-surge operation, the zener voltage will vary with time and may be affected significantly by the zener resistance. For best regulation, keep current excursions as low as possible.

**FIGURE 4.39 Continued**
Summary

A diode is a two-terminal semiconductor device. It offers a very low resistance in the forward direction and a very high resistance in the reverse direction. The analysis of diode circuits can be simplified by assuming an ideal diode model in which the resistance in the forward-biased condition is zero and the resistance in the reverse direction is very large, tending to infinity.

A practical diode exhibits a nonlinear characteristic, analysis of which requires a graphical or iterative method. In order to linearize the diode characteristic to apply linear circuit laws, a practical diode is normally represented by (a) a constant DC drop $V_{TD}$, (b) a piecewise linear DC model, (c) a small-signal AC resistance $r_d$, or (d) a high-frequency AC model.

In a zener diode, the reverse breakdown is controlled, and the zener voltage is the reverse breakdown voltage. The diode characteristic depends on the operating temperature, and the leakage current almost doubles for every 10°C increase in the junction temperature.

References


Review Questions

1. What is a diode?
2. What is the characteristic of an ideal diode?
3. What is a rectifier?
4. What is doping?
5. What is the depletion region of a diode?
6. What are the forward and reverse characteristics of a practical diode?
7. What is the forward-biased region of a diode?
8. What is the reverse-biased region of a diode?
9. What is the breakdown region of a diode?
10. What is the effect of junction temperature on the diode characteristic?
11. What are the three methods for analyzing diode circuits?
12. What is the low-frequency AC model of a diode?
13. What is the AC resistance of a diode?
14. What is the high-frequency AC model of a diode?
15. What is the PSpice/SPICE model of a diode?
16. What is a zener diode?
17. What is zener voltage?
18. What is a shunt regulator?
19. What is zener resistance?
20. What is the bulk resistance of a diode?

Problems

The symbol indicates that a problem is a design problem.

4.2 Ideal Diodes

4.1 The diode circuit shown in Fig. P4.1 has \( R = 30 \, k\Omega \) and \( V_{DD} = 10 \, V \). Determine the voltage \( v_O \) and the current \( i_O \) if (a) \( v_S = 5 \, V \) and (b) \( v_S = 12 \, V \). Assume a diode drop of \( V_D = 0.7 \, V \).

![FIGURE P4.1](image)

4.2 The diode circuit shown in Fig. P4.2 has \( R_1 = 30 \, k\Omega \), \( R_2 = 10 \, k\Omega \), \( R_3 = 80 \, k\Omega \), and \( R_4 = 20 \, k\Omega \). Determine the voltage \( v_O \) and the current \( i_O \). Assume a diode drop of \( V_D = 0.7 \, V \) and \( V_D = 10 \, V \).

![FIGURE P4.2](image)
4.3 The diode circuit shown in Fig. P4.3 has \( R_1 = 30 \, \text{k}\Omega, R_2 = 10 \, \text{k}\Omega, R_3 = 80 \, \text{k}\Omega, \) and \( R_4 = 20 \, \text{k}\Omega. \) Determine the voltage \( v_0 \) and the current \( i_0. \) Assume a diode drop of \( V_D = 0.7 \, \text{V} \) and \( V_1 = 15 \, \text{V}. \)

**FIGURE P4.3**

4.4 The diode circuit shown in Fig. P4.4 has \( R_1 = 1 \, \text{k}\Omega, R_2 = 2 \, \text{k}\Omega, V_1 = 12 \, \text{V}, \) and \( V_2 = 15 \, \text{V}. \) Determine the diode currents \( i_1 \) and \( i_2. \) Assume a diode drop of \( V_D = 0.7 \, \text{V}. \)

**FIGURE P4.4**

4.5 The diode circuit shown in Fig. P4.5 has \( R_1 = 5 \, \text{k}\Omega, R_2 = 15 \, \text{k}\Omega, E_1 = 5 \, \text{V}, \) and \( v_S = 15 \, \text{V}. \) Determine the voltage \( v_0 \) and the current \( i_0. \) Assume a diode drop of \( V_D = 0.7 \, \text{V}. \)

**FIGURE P4.5**

4.6 The diode circuit shown in Fig. P4.6 has \( R_1 = 5 \, \text{k}\Omega, R_2 = 15 \, \text{k}\Omega, E_1 = 10 \, \text{V}, E_2 = 15 \, \text{V}, \) and \( v_S = 15 \, \text{V}. \) Determine the voltage \( v_0 \) and the current \( i_0. \) Assume a diode drop of \( V_D = 0.7 \, \text{V}. \)

**FIGURE P4.6**

4.7 Find the voltage \( v_0 \) and the current \( i_0 \) of the diode circuit in Fig. P4.7 if \( R_1 = 5 \, \text{k}\Omega, R_2 = 10 \, \text{k}\Omega, V_1 = 5 \, \text{V}, \) and \( V_2 = 10 \, \text{V}. \) Assume diode voltage drop \( V_D = 0.7 \, \text{V}. \)
4.8 Find the voltage \( v_O \) and the current \( i_O \) of the diode circuit in Fig. P4.8 if \( R_1 = 1 \, \text{k}\Omega \), \( R_2 = 5 \, \text{k}\Omega \), \( V_1 = 10 \, \text{V} \), and \( V_2 = 5 \, \text{V} \). Assume diode voltage drop \( V_D = 0.7 \, \text{V} \).

4.9 Find the voltage \( v_O \) and the current \( i_O \) of the diode circuit in Fig. P4.9 if \( R_1 = 1 \, \text{k}\Omega \) and \( V_1 = 5 \, \text{V} \). Assume diode voltage drop \( V_D = 0.7 \, \text{V} \).

4.3 Transfer Characteristics of Diode Circuits

4.10 Plot the transfer characteristic (\( v_O \) versus \( v_S \)) of the diode circuit in Fig. P4.10 if the input voltage \( v_S \) is varied from 0 to 10 \, \text{V} \) in increments of 2 \, \text{V} \). Assume \( R_1 = 5 \, \text{k}\Omega \), \( R_2 = 1 \, \text{k}\Omega \), \( V_1 = 10 \) to 5 \, \text{V} \), and a diode voltage drop \( V_D = 0.7 \, \text{V} \).
4.11 Plot the transfer characteristic ($v_O$ versus $v_S$) of the diode circuit in Fig. P4.11 if the input voltage $v_S$ is varied from $-10$ V to $10$ V in increments of $2$ V. Assume $R_1 = 5$ kΩ, $R_2 = 1$ kΩ, $V_1 = 5$ V, $V_2 = 5$ V, and a diode voltage drop $V_D = 0.7$ V.

**FIGURE P4.10**

![Diode Circuit Diagram](image)

4.12 The measured values of a diode at junction temperature $T_j = 25^\circ$C are

$$V_D = \begin{cases} 
0.65 \text{ V} & \text{at } I_D = 10 \mu\text{A} \\
0.8 \text{ V} & \text{at } I_D = 1 \text{ mA}
\end{cases}$$

Determine (a) the emission coefficient $n$ and (b) the leakage current $I_S$.

4.13 The threshold voltage of a silicon diode is $V_{TD} = 0.75$ V at $25^\circ$C. Find the threshold voltage $V_{TD}$ at (a) $T_j = 125^\circ$C and (b) $T_j = -150^\circ$C.

4.14 The leakage current of a silicon diode is $I_S = 5 \times 10^{-14}$ A at $T_j = 25^\circ$C, and the emission coefficient is $n = 1.8$. The junction temperature is $T_j = 90^\circ$C. Determine (a) the leakage current $I_S$ and (b) the diode current $i_D$ at a diode voltage of $v_D = 0.9$ V.

4.5 Analysis of Practical Diode Circuits

4.15 The diode circuit shown in Fig. 4.10 has $R_L = 4$ kΩ and $V_S = 15$ V. The emission coefficient is $n = 1.8$. Use the iterative method to calculate the $Q$-point (or operating point), whose coordinates are $V_D$ and $I_D$. Assume an approximate diode drop of $V_D = 0.75$ V at $i_D = 0.1$ mA. Assume a junction temperature of $25^\circ$C. Use three iterations only.

4.16 Repeat Prob. 4.15 using the approximate method with $V_D = 0.75$ V.
4.17 The diode circuit shown in Fig. 4.10 has \( R_L = 1 \, \text{k}\Omega \) and \( V_S = 10 \, \text{V} \). The diode characteristic is described by

\[
i_D = K V_D^2 = 5 \times 10^{-4} V_D^2 \quad (i_D \text{ in amps and } v_D \text{ in volts})
\]

Determine the values of \( V_D \) and \( I_D \) at the \( Q \)-point (or operating point) by using (a) the iterative method and (b) the mathematical method.

### 4.6 Modeling of Practical Diodes

4.18 The diode circuit shown in Fig. 4.16(a) has \( V_S = 15 \, \text{V} \) and \( R_L = 2.5 \, \text{k}\Omega \). The diode characteristic is shown in Fig. 4.16(b). Determine the diode voltage \( V_D \), the diode current \( I_D \), and the load voltage \( v_O \) by using (a) the piecewise linear DC model and (b) the constant-drop DC model.

4.19 The diode circuit shown in Fig. 4.18(a) has \( V_S = 12 \, \text{V} \), \( V_m = 150 \, \text{mV} \), and \( R_L = 5 \, \text{k}\Omega \). Assume emission coefficient \( n = 1.8 \), diode voltage drop \( v_D = 0.75 \, \text{V} \) at \( i_D = 0.5 \, \text{mA} \), and \( V_T = 25.8 \, \text{mV} \) at a junction temperature of 25°C. Determine (a) the \( Q \)-point \((V_D, I_D)\), (b) the parameters \((V_{TD}, R_D)\) of the piecewise linear DC model, and (c) the instantaneous diode voltage \( v_D \).

4.20 The diode circuit shown in Fig. 4.18(a) has \( V_S = 12 \, \text{V} \), \( V_m = 150 \, \text{mV} \), \( R_L = 5 \, \text{k}\Omega \), and \( V_T = 25.8 \, \text{mV} \). Assume an emission coefficient of \( n = 2 \) and the diode saturation current \( I_S = 2.682 \times 10^{-9} \). Use PSpice/SPICE to (a) calculate the \( Q \)-point and small-signal parameters and (b) plot the instantaneous output voltage \( v_O = v_D \). Assume PSpice/SPICE model parameters of diode 1N4148:

\[
IS=2.682N \quad CJO=4P \quad M=.3333 \quad VJ=.5 \quad BV=100 \quad IBV=100U \quad TT=11.54N \quad N=1.8
\]

4.21 The diode circuit shown in Fig. 4.10 has \( V_S = 18 \, \text{V} \) and \( R_L = 1.5 \, \text{k}\Omega \). The diode forward characteristic, which can be obtained from practical measurements, can be represented by the following data:

<table>
<thead>
<tr>
<th>( i_D ) (mA)</th>
<th>( v_D ) (V)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>0.87</td>
</tr>
<tr>
<td>15</td>
<td>0.98</td>
</tr>
<tr>
<td>30</td>
<td>1.058</td>
</tr>
<tr>
<td>45</td>
<td>1.115</td>
</tr>
<tr>
<td>60</td>
<td>1.173</td>
</tr>
<tr>
<td>75</td>
<td>1.212</td>
</tr>
<tr>
<td>90</td>
<td>1.25</td>
</tr>
<tr>
<td>105</td>
<td></td>
</tr>
</tbody>
</table>

Determine (a) the \( Q \)-point \((V_D, I_D)\), (b) the small-signal DC resistance \( R_D \) and threshold voltage \( V_{TD} \), and (c) the small-signal AC resistance \( r_d \). Assume \( n = 1 \) and \( V_T = 25.8 \, \text{mV} \).

4.22 The diode circuit shown in Fig. 4.10 has \( R_L = 1 \, \text{k}\Omega \) and \( V_S = 10 \, \text{V} \). The diode characteristic is described by

\[
i_D = K V_D^2 = 5 \times 10^{-4} V_D^2 \quad (i_D \text{ in amps and } v_D \text{ in volts})
\]

Determine (a) the diode voltage \( V_D \), (b) the diode current \( I_D \), and (c) the load voltage \( V_O \).

4.23 The characteristic of the diode in Fig. P.4.23 is described by

\[
i_D = K V_D^2 = 5 \times 10^{-4} V_D^2 \quad (i_D \text{ in amps and } v_D \text{ in volts})
\]

Determine (a) the values of \( V_D \) and \( I_D \) at the \( Q \)-point (or operating point), (b) the small-signal ac resistance \( r_d \), and (c) the rms output voltage \( V_{o(rms)} \). Assume that the capacitor C offers a negligible impedance at the operating frequency.

**FIGURE P4.23**

![Diode Circuit Diagram](image-url)
4.24 The characteristic of the diode circuit in Fig. P2.23 follows the Shockley diode equation with a leakage current of $I_S = 2.682 \times 10^{-9}$ A at 25°C and an emission coefficient of $n = 1.8$. Use PSpice/SPICE to (a) calculate the $Q$-point and the small-signal parameters and (b) plot the instantaneous output voltage $v_O = v_D$. Assume these PSpice/SPICE model parameters:

$$IS = 2.682N, M = 0.3333, VJ = 0.5, BV = 100\text{ V}, IBV = 100U, TT = 11.54N, CJO = 10\text{ PF}, N = 1.8$$

4.25 A diode circuit is shown in Fig. P4.25. The diode characteristic is given by

$$i_D = 5 \times 10^{-2} v_D^2$$

Determine (a) the values of $V_D$ and $I_D$ at the $Q$-point (or operating point), (b) the small-signal AC resistance $r_d$, (c) the threshold voltage $V_{TD}$, and (d) the rms output voltage $V_{O(rms)}$.

**FIGURE P4.25**

4.26 The characteristic of the diode in Fig. P4.25 follows the Shockley diode equation with a leakage current of $I_S = 2.682 \times 10^{-9}$ A at 25°C and an emission coefficient of $n = 1.8$. Use PSpice/SPICE to (a) calculate the $Q$-point and the small-signal parameters and (b) plot the instantaneous output voltage $v_O$. Assume these PSpice/SPICE model parameters:

$$IS = 2.682N, M = 0.3333, VJ = 0.5, BV = 100\text{ V}, IBV = 100U, TT = 11.54N, CJO = 10\text{ PF}, N = 1.8$$

4.27 A diode circuit is shown in Fig. P4.27. Use PSpice/SPICE to (a) determine the operating diode voltages and currents and (b) find the small-signal parameters of the diodes. The supply voltage $V_S$ is 12 V. Use default values for the PSpice/SPICE model parameters of 1N4148 diodes.

**FIGURE P4.27**

4.28 A diode circuit is shown in Fig. P4.28. Use PSpice/SPICE to (a) determine the operating diode voltages and currents and (b) find the small-signal parameters of the diodes. The supply voltage $V_S$ is 12 V. Use default values for the PSpice/SPICE model parameters of 1N4148 diodes.
4.34 A DC voltmeter is constructed using a DC meter, as shown in Fig. P4.34. The full-scale deflection of the meter is 150 \( \mu \text{A} \), and the internal resistance \( R_m \) of the meter is 100 \( \Omega \). The zener voltage \( V_Z \) is 10 V, and the zener resistance \( R_Z \) is negligible. The voltmeter is required to measure 220 V at a full-scale deflection.
a. Design the voltmeter by determining the values of $R_1$ and $R_2$.

b. Use PSpice/SPICE to check your design by plotting the meter current $I_m$ against the supply voltage $V_S$. Assume these PSpice/SPICE model parameters:

**4.35**

The zener voltage of the unsymmetrical regulator in Fig. 4.26(a) is $V_Z = 6.3$ V at $I_{ZT} = 20$ mA. The current-limiting resistance $R_s$ is 1.5 kΩ, and the load resistance $R_L$ is very large, tending to infinity. The supply voltage $v_s$ varies from 0 to 30 V. Use PSpice/SPICE to plot the output voltage $v_o$ against the input voltage $v_s$ for $T_j = 25°C$ and $T_j = 150°C$. Assume these PSpice/SPICE model parameters:

**4.36**

The zener voltage of the symmetrical regulator in Fig. 4.31(a) is $V_Z = 6.3$ V at $I_{ZT} = 20$ mA. The current-limiting resistance $R_s$ is 1.5 kΩ, and the load resistance $R_L$ is very large, tending to infinity. The supply voltage $v_s$ varies from 0 to 30 V. Use PSpice/SPICE to plot the output voltage $v_o$ against the input voltage $v_s$ for $T_j = 25°C$ and $T_j = 150°C$. Assume these PSpice/SPICE model parameters:

**4.37** Two zener diodes are connected as shown in Fig. P4.37. The diode current in the forward direction is described by

$$i_D = I_S(e^{v_D/V_T} - 1)$$

where $V_T = 0.026$ and $I_S = 5 \times 10^{-15}$ A. The supply voltage $v_s$ is 7.5 V. The zener voltage $V_z$ of each diode is 6.7 V, and the zener resistance $R_Z$ is negligible. The forward voltage drop $V_{TD}$ of each diode is 0.8 V. Determine (a) the expression for each diode voltage $v_{D1}$ and $v_{D2}$, (b) the operating diode voltages $V_{D1}$ and $V_{D2}$, and (c) the diode current $I_D$.

**4.38** A zener regulator is shown in Fig. P4.38. Use PSpice/SPICE to plot the transfer characteristic between $v_o$ and $v_s$. $v_s$ varies from −18 V to 18 V in increments of 0.5 V. The PSpice/SPICE model parameters of the zener diodes are
4.39 The zener regulator shown in Fig. P4.39 has $R_1 = 10 \, \text{k}\Omega$, $R_2 = 100 \, \text{k}\Omega$, and $R_3 = 100 \, \text{k}\Omega$. Determine the voltage $v_O$ and the power ratings of all elements if $v_S = 15 \, \text{V}$. The zener parameters are $V_{Z1} = 7 \, \text{V}$, $R_{Z1} = 0$, and $V_{Z2} = 5 \, \text{V}$, $R_{Z2} = 0$.

4.40 The zener regulator shown in Fig. P4.40 has $R_1 = 1 \, \text{k}\Omega$, $R_2 = 5 \, \text{k}\Omega$, and $R_3 = 10 \, \text{k}\Omega$. Determine the output voltage $v_O$ and its ripple voltage if the supply voltage $v_S$ varies from 20 V to 30 V. The zener parameters are $V_{Z1} = 12 \, \text{V}$, $R_{Z1} = 40 \, \text{\Omega}$, and $V_{Z2} = 7.5 \, \text{V}$, $R_{Z2} = 25 \, \text{\Omega}$.

4.8–4.9 *Light-Emitting Diodes and Power Ratings*

4.41 Design an LED circuit so that the diode current $I_D$ is 1 mA. Assume an emission coefficient of $n = 2$, a leakage current $I_S = 10^{-10} \, \text{A}$, and $V_T = 25.8 \, \text{mV}$ at a junction temperature of 25°C.

4.42 A diode is operated at a $Q$-point of $V_D = 0.7 \, \text{V}$ and $I_D = 1 \, \text{A}$. The diode parameters are $P_D = 1.5 \, \text{W}$ at $T_A = 50^\circ\text{C}$ and $P_{\text{derating}} = 6.67 \, \text{mW}/0^\circ\text{C}$. The ambient temperature is $T_A = 25^\circ\text{C}$, and the maximum permissible
junction temperature is $T_{JM} = 250^\circ C$. Calculate (a) the junction temperature $T_J$, (b) the maximum permissible junction dissipation $P_{DM}$, and (c) the permissible junction dissipation $P_{DM}$ at an ambient temperature of $T_A = 65^\circ C$.

4.43 A diode is operated at a $Q$-point of $V_D = 0.625 \, V$ and $I_D = 100 \, mA$. The diode parameters are $P_D = 200 \, mW$ at $T_A = 50^\circ C$ and $P_{\text{derating}} = 6.67 \, mW/0^\circ C$. The ambient temperature is $T_A = 25^\circ C$, and the maximum permissible junction temperature is $T_{JM} = 200^\circ C$. Calculate (a) the junction temperature $T_J$, (b) the maximum permissible junction dissipation $P_{DM}$, and (c) the permissible junction dissipation $P_{DM}$ at an ambient temperature of $T_A = 75^\circ C$.

4.44 A diode is operated at a $Q$-point of $V_D = 0.75 \, V$ and $I_D = 200 \, mA$. The diode parameters are $P_D = 500 \, mW$ at $T_A = 50^\circ C$ and $P_{\text{derating}} = 6.67 \, mW/0^\circ C$. The ambient temperature is $T_A = 25^\circ C$, and the maximum permissible junction temperature is $T_{JM} = 250^\circ C$. Calculate (a) the junction temperature $T_J$, (b) the maximum permissible junction dissipation $P_{DM}$, and (c) the permissible junction dissipation $P_{DM}$ at an ambient temperature of $T_A = 55^\circ C$. 
CHAPTER 5

APPLICATIONS OF DIODES

Learning Outcomes
After completing this chapter, students should be able to do the following:

- Select a diode rectifier to meet DC output voltage requirements.
- Design diode rectifiers to produce a DC supply voltage from an AC supply.
- Calculate the values of filter components in order to limit the ripple content on the DC output to a specified value.
- List some applications of diodes in wave-shaping of signals.
- Describe the diode applications as clippers, clampers, voltage multipliers, and transfer function synthesis.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$I_{D(\text{av})}$, $I_{D(\text{rms})}$</td>
<td>Average and rms diode currents</td>
</tr>
<tr>
<td>$I_{O(\text{av})}$, $I_{O(\text{rms})}$</td>
<td>Average and rms output currents</td>
</tr>
<tr>
<td>$I_p$, $I_s$</td>
<td>rms primary and secondary currents of an input transformer</td>
</tr>
<tr>
<td>$P_{o(\text{dc})}$, $P_{o(\text{ac})}$</td>
<td>Average and AC output powers</td>
</tr>
<tr>
<td>RF, PF</td>
<td>Output ripple factor and power factor</td>
</tr>
<tr>
<td>$v_D(t)$, $i_D(t)$, $V_D$</td>
<td>Instantaneous diode voltage, diode current, and DC diode voltage drop</td>
</tr>
<tr>
<td>$v_i(t)$, $v_O(t)$, $v_r(t)$</td>
<td>Instantaneous input supply, output, and ripple voltages</td>
</tr>
</tbody>
</table>
5.1 Introduction

We saw in Chapter 4 that a diode offers a very low resistance in one direction and a very high resistance in the other direction, thus permitting an easy current flow in only one direction. This chapter will illustrate the applications of diodes in wave-shaping circuits. For the sake of simplicity, we will assume ideal diodes—that is, diodes in which the voltage drop across the diode is zero rather than the typical value of 0.7 V.

5.2 Diode Rectifier

The most common applications of diodes are as rectifiers. A rectifier that converts an AC voltage to a unidirectional voltage is used as a DC power supply for many electronic circuits, such as those in radios, calculators, and stereo amplifiers. A rectifier is also called an AC–DC converter. Rectifiers can be classified on the basis of AC input supply into two types: single-phase rectifiers, in which the AC input voltage is a single-phase source, and three-phase rectifiers, in which the AC input voltage is a three-phase source [1]. Three-phase rectifiers, which are normally used in high-power applications, are outside the scope of this book. The following single-phase rectifiers are commonly used in electronic circuits: single-phase half-wave rectifiers, single-phase full-wave center-tapped rectifiers, and single-phase full-wave bridge rectifiers. For simplicity, we will assume ideal diodes in the following analysis and derivations; that is, the DC voltage drop across a diode is zero rather than a typical value of $V_D = 0.7$ V.

5.2.1 Single-Phase Half-Wave Rectifiers

The circuit diagram of a single-phase half-wave rectifier is shown in Fig. 5.1(a). Let us consider a sinusoidal input voltage $v_S = v_s = V_m \sin \omega t$, where $\omega = 2\pi f$ and $f$ is the frequency of the input voltage. Thus, there is no DC component on the input voltage; that is, $v_S = 0$ and $v_S = V_S + v_S = v_S$. Since $v_S$ is positive from $\omega t = 0$ to $\pi$ and negative from $\omega t = \pi$ to $2\pi$, the operation of the rectifier can be divided into two intervals: interval 1 and interval 2.

Interval 1 is the interval $0 \leq \omega t \leq \pi$ during the positive half-cycle of the input voltage. Diode D1 conducts and behaves like a short circuit, as shown in Fig. 5.1(b). The input voltage appears across the load resistance $R_L$. That is, the output voltage becomes

$$v_O = V_m \sin \omega t \quad \text{for} \quad 0 \leq \omega t \leq \pi$$

If we include the DC diode drop $V_D (= 0.7$ V$)$, the peak output voltage $V_m$ will be reduced to $(V_m - V_D)$ and the instantaneous output voltage will become

$$v_O = (V_m - V_D) \sin \omega t \quad \text{for} \quad 0 \leq \omega t \leq \pi$$
Interval 2 is the interval \( \pi \leq \omega t \leq 2\pi \) during the negative half-cycle of the input voltage. Diode D\(_1\) is reverse biased and behaves like an open circuit, as shown in Fig. 5.1(b). The output voltage \( v_O \) becomes zero. That is,

\[
v_O = 0 \quad \text{for} \quad \pi \leq \omega t \leq 2\pi
\]

The waveforms of the input voltage, the output voltage, and the diode voltage are shown in Fig. 5.1(c). The output voltage will be reduced due to the diode drop of approximately 0.7 V as shown by the dotted lines. When diode D\(_1\) conducts, its voltage becomes zero. When the diode is reverse biased, the diode current becomes zero and the diode has to withstand the input voltage. The peak inverse voltage (PIV) the diode must withstand is equal to the peak input voltage \( V_m \). The voltage on the anode side of the diode is AC, whereas on the cathode side it is DC. That is, the diode converts AC voltage to DC. The average output voltage \( V_{o(\text{av})} \) is found using the following equation:

\[
V_{o(\text{av})} = \frac{1}{2\pi} \int_{0}^{\pi} V_O d(\omega t) = \frac{1}{2\pi} \int_{0}^{\pi} V_m \sin \omega t \, d(\omega t) = \frac{V_m}{\pi} = 0.318V_m
\] (5.1)

Therefore, the average load current \( I_{o(\text{av})} \) for a resistive load can be found from

\[
I_{o(\text{av})} = \frac{V_{o(\text{av})}}{R_L} = \frac{V_m}{\pi R_L} = \frac{0.318V_m}{R_L}
\] (5.2)
The rms output voltage \( V_{o(rms)} \) is given by

\[
V_{o(rms)} = \left[ \frac{1}{2\pi} \int_0^\pi V_O^2 d(\omega t) \right]^{1/2} = \left[ \frac{1}{2\pi} \int_0^\pi V_m^2 \sin^2 \omega t d(\omega t) \right]^{1/2}
\]

\[= \frac{V_m}{2} = 0.5V_m \tag{5.3}\]

and the rms load current \( I_{o(rms)} \) is given by

\[
I_{o(rms)} = \frac{V_{o(rms)}}{R_L} = \frac{0.5V_m}{R_L} \tag{5.4}\]

Notice from Fig. 5.1(c) that the output voltage \( v_O \) is pulsating and contains ripples. In practice, a filter is normally required at the rectifier output to smooth out the DC output voltage. We often know the ripple content of the output voltage. The output voltage can be viewed as consisting of two components: ripple voltage and average voltage. The instantaneous ripple voltage \( v_r \), which is the difference between \( v_O \) and \( V_{o(av)} \), is shown in Fig. 5.1(d). The value of \( v_r \) can be expressed as

\[
v_r = \begin{cases} v_S - V_{o(av)} = V_m \sin \omega t - V_{o(av)} & \text{for } 0 \leq \omega t \leq \pi \\ -V_{o(av)} & \text{for } \pi \leq \omega t \leq 2\pi \end{cases} \tag{5.5}\]

Let \( V_{r(rms)} \) be the rms ripple voltage. Then \( V_{r(rms)} \) can be related to \( V_{o(av)} \) and \( V_{o(rms)} \) by

\[
V_{r(rms)}^2 + V_{o(av)}^2 = V_{r(rms)}^2 \]

or

\[
V_{r(rms)}^2 = V_{o(rms)}^2 - V_{o(av)}^2 \tag{5.6}\]

Substituting \( V_m \) from Eq. (5.1) into Eq. (5.3), we get \( V_{o(rms)} = \pi V_{o(av)}/2 \), which is then applied to Eq. (5.6) to give \( V_{r(rms)}^2 \):

\[
V_{r(rms)} = \left[ \frac{\pi^2}{4} V_{o(av)}^2 - V_{o(av)}^2 \right]^{1/2} = V_{o(av)} \left[ \frac{\pi^2}{4} - 1 \right]^{1/2} = 1.21V_{o(av)} \tag{5.7}\]

The ripple content of the output voltage is measured by a factor known as the ripple factor (RF), which is defined by

\[
RF = \frac{V_{r(rms)}}{V_{o(av)}} = \frac{1.21V_{o(av)}}{V_{o(av)}} = 1.21 \text{ or } 121\% \tag{5.8}\]

**NOTE** This numerical value of RF = 121% is valid only for the single-phase half-wave rectifier.

The AC output power \( P_{o(ac)} \) is the average power and is defined as

\[
P_{o(ac)} = \frac{1}{2\pi} \int_0^{2\pi} i_o^2 R_L d(\omega t) = I_{o(rms)}^2 R_L = V_{o(rms)} I_{o(rms)} = \left( \frac{V_m}{2} \right)^2 \frac{1}{R_L} \tag{5.9}\]
which will be the same as the input power $P_{in}$ if we assume there is no power loss in the rectifier. That is, the input power is given by

$$P_{in} = P_{o(ac)} = \left(\frac{V_m}{2}\right)^2 \frac{1}{R_L} \tag{5.10}$$

The DC output power $P_{o(dc)}$ is defined by

$$P_{o(dc)} = V_{o(av)}I_{o(av)} = \frac{V_m^2}{\pi^2 R_L} \tag{5.11}$$

It is generally smaller than $P_{o(ac)}$ because the rms values are larger than the average (DC) values. The effectiveness of a rectifier in delivering DC output power is generally measured by the rectification efficiency $\eta_R$, which is defined as

$$\eta_R = \frac{P_{o(dc)}}{P_{o(ac)}} = \frac{V_{o(av)}I_{o(av)}}{V_{o(rms)}I_{o(rms)}} = \frac{(V_m/\pi)^2/R_L}{(V_m^2/2)^2/R_L} = \frac{4}{\pi^2} = 40.5\% \tag{5.12}$$

If we assume there is no power loss in the rectifier, then the input power factor (PF), which is a measure of the power drawn from the input power supply, is related to the input power ($P_{in}$) by

$$V_sI_sPF = P_{in} = P_{o(ac)} \tag{5.13}$$

This gives the input power factor as given by

$$PF = \frac{P_{o(ac)}}{V_sI_s} = \frac{(V_m/2)^2/R_L}{(V_m/\sqrt{2})(V_m/2R_L)} = \frac{\sqrt{2}}{2} = 0.707 \tag{5.14}$$

where $V_s$ and $I_s$ are the rms input supply voltage and the input supply current, respectively.

**NOTE** These numerical values of $\eta_R = 40.5\%$ and $PF = 0.707$ are valid only for the single-phase half-wave rectifier.

Rectifiers are generally supplied through a transformer from a fixed AC input voltage of 120 V (rms) in order to satisfy the output voltage requirement. This arrangement is shown in Fig. 5.2(a). Let us assume an ideal transformer. Then the primary rms voltage $V_p$ is related to the secondary rms voltage $V_s$ by the turns ratio $n$, as follows:

$$\frac{V_p}{V_s} = \frac{N_p}{N_s} = n \tag{5.15}$$

where $N_p$ is the number of turns of the primary winding and $N_s$ is the number of turns of the secondary winding.
Assuming there is no power loss in the transformer, the input (primary) side power must equal the output (secondary) side power. That is,

\[ V_p I_p = V_s I_s \]  

which, after we use the relationship between the voltages on the primary side and the secondary side, gives the relationship between the primary side current to the secondary side current as

\[ I_s = n I_p \]  

\[ (5.17) \]

**NOTES**

1. If the rectifier is connected to a battery charger, \( P_{o(d)} \) is the useful power transferred to the battery. Since \( P_{o(ac)} \) is greater than \( P_{o(d)} \), \( P_{loss} = P_{o(ac)} - P_{o(d)} \) will be responsible for heating the battery. For a resistive load, however, the AC power \( P_{o(ac)} \) becomes the average output power and will produce the effective heat.

2. The average current through the input side of an ideal transformer will be \( I_{o(\text{av})} / n \). A transformer is normally designed to operate from a sinusoidal AC source so that the magnetic core of the transformer is set and reset in every cycle. The unidirectional DC current flow through the transformer may cause the transformer core to saturate. Therefore, this circuit is suitable only for very low-power applications, typically tens of watts.

3. Unless noted otherwise, the AC input voltage is always specified in rms values, so \( V_m = \sqrt{2} V_s \).

**EXAMPLE 5.1**

**Finding the performance parameters of a single-phase half-wave rectifier** The single-phase half-wave rectifier of Fig. 5.2(a) is supplied from a 120-V, 60-Hz source through the step-down transformer of Fig. 5.2(b) with turns ratio \( n = 10:1 \). The load resistance \( R_L \) is 5 \( \Omega \). Determine (a) the average output voltage \( V_{o(\text{av})} \), (b) the average load current \( I_{o(\text{av})} \), (c) the rms load voltage \( V_{o(rms)} \), (d) the rms load current \( I_{o(rms)} \), (e) the ripple factor RF of the output voltage, (f) the rms ripple voltage \( V_{r(rms)} \), (g) the average diode current \( I_{D(\text{av})} \), (h) the rms diode current \( I_{D(rms)} \), (i) the peak inverse voltage PIV of the diode, (j) the average output power \( P_{o(ac)} \), (k) the DC output power \( P_{o(d)} \), (l) the frequency \( f_r \) of the output ripple voltage, and (m) the input power factor PF.
The primary transformer voltage is \( V_p = 120 \, \text{V} \). From Eq. (5.15), the secondary transformer voltage is \( V_s = V_p/n = 120/10 = 12 \, \text{V} \). The peak input voltage of the rectifier is

\[
V_m = \sqrt{2} V_s = \sqrt{2} \times 12 = 16.97 \, \text{V}
\]

(a) From Eq. (5.1),

\[
V_{o(\text{av})} = 0.318 V_m = 0.318 \times 16.97 = 5.4 \, \text{V}
\]

(b) From Eq. (5.2),

\[
I_{o(\text{av})} = \frac{V_{o(\text{av})}}{R_L} = \frac{5.4}{5} = 1.08 \, \text{A}
\]

(c) From Eq. (5.3),

\[
V_{o(\text{rms})} = 0.5V_m = 0.5 \times 16.97 = 8.49 \, \text{V}
\]

(d) From Eq. (5.4),

\[
I_{o(\text{rms})} = \frac{V_{o(\text{rms})}}{R_L} = \frac{8.49}{5} = 1.7 \, \text{A}
\]

(e) From Eq. (5.8), RF = 1.21, or 121%.

(f) From Eq. (5.8),

\[
V_{r(\text{rms})} = RF \times V_{o(\text{av})} = 1.21 \times 5.4 = 6.53 \, \text{V}
\]

(g) The average diode current \( I_{D(\text{av})} \) will be the same as that of the load. That is, \( I_{D(\text{av})} = I_{o(\text{av})} = 1.08 \, \text{A} \).

(h) The rms diode current \( I_{D(rms)} \) will be the same as that of the load. That is, \( I_{D(rms)} = I_{o(rms)} = 1.7 \, \text{A} \).

(i) PIV = \( V_m = 16.97 \, \text{V} \).

(j) From Eq. (5.9),

\[
P_{o(ac)} = I_{o(rms)}^2 R_L = (1.7)^2 \times 5 = 14.45 \, \text{W}
\]

(k) From Eq. (5.11),

\[
P_{o(dc)} = V_{o(\text{av})} I_{o(\text{av})} = 5.4 \times 1.08 = 5.83 \, \text{W}
\]

(l) Notice from Fig. 5.1(d) that the frequency of the output ripple voltage is the same as the input frequency, \( f_i = f = 60 \, \text{Hz} \).

(m) \( V_o = 12 \, \text{V}, I_o = I_{o(rms)} = 1.7 \, \text{A} \), and \( P_{\text{in}} = P_{o(ac)} = 14.45 \, \text{W} \)

From Eq. (5.14),

\[
\text{PF} = \frac{P_{o(ac)}}{V_o I_o} = \frac{14.45}{12 \times 1.7} = 0.7071
\]
EXAMPLE 5.2

Fourier components of the output voltage of a single-phase half-wave rectifier The single-phase half-wave rectifier of Fig. 5.1(a) is connected to a source of $V_s = 120$ V, 60 Hz. Express the instantaneous output voltage $v_O(t)$ by a Fourier series. Assume ideal diodes with zero voltage drops.

SOLUTION

The output voltage $v_O$ can be described by

$$v_O = \begin{cases} V_m \sin \omega t & \text{for } 0 \leq \omega t \leq \pi \\ 0 & \text{for } \pi \leq \omega t \leq 2\pi \end{cases}$$

which can be expressed by a Fourier series as

$$v_O(\theta) = V_{O(\text{av})} + \sum_{n=1,2,\ldots}^{\infty} (a_n \sin n\theta + b_n \cos n\theta) \quad \text{where } \theta = \omega t = 2\pi ft \quad (5.18)$$

$$V_{O(\text{av})} = \frac{1}{2\pi} \int_{0}^{2\pi} v_O d\theta = \frac{1}{2\pi} \left[ \int_{0}^{\pi} V_m \sin \theta \, d\theta + \int_{\pi}^{2\pi} 0 \, d\theta \right] = \frac{V_m}{\pi}$$

$$a_n = \frac{1}{\pi} \int_{0}^{2\pi} v_O \sin n\theta \, d\theta = \frac{1}{\pi} \int_{0}^{\pi} V_m \sin \theta \sin n\theta \, d\theta$$

$$= \begin{cases} \frac{V_m}{2} & \text{for } n = 1 \\ 0 & \text{for } n = 2, 3, 4, 5, \ldots, \infty \end{cases}$$

$$b_n = \frac{1}{\pi} \int_{0}^{2\pi} v_O \cos n\theta \, d\theta = \frac{1}{\pi} \int_{0}^{\pi} V_m \sin \theta \cos n\theta \, d\theta$$

$$= \begin{cases} -\frac{2V_m}{\pi} \left( \frac{1}{n^2 - 1} \right) & \text{for } n = 2, 4, 6, \ldots, \infty \\ 0 & \text{for } n = 1, 3, 5, \ldots, \infty \end{cases}$$

When the values of $a_n$ and $b_n$ are inserted into Eq. (5.18), the expression for the instantaneous output voltage $v_O$ becomes

$$v_O(t) = \frac{V_m}{\pi} + \frac{V_m}{2} \sin \omega t - \frac{2V_m}{3\pi} \cos 2\omega t - \frac{2V_m}{15\pi} \cos 4\omega t - \frac{2V_m}{35\pi} \cos 6\omega t - \cdots \quad (5.19)$$

$$= -\frac{2V_m}{(2n - 1)(2n + 1)\pi} \cos 2n\omega t \quad \text{for } n = 1, 3, 5, \ldots, \infty$$

where $V_m = \sqrt{2} \times 120 = 169.7$ V and $\omega = 2\pi \times 60 = 377$ rad/s.

NOTE: Equation (5.19) contains sine and cosine components, which are known as harmonics. Except for the sine term, only the even harmonics are present, and their magnitudes decrease with the order of the harmonic frequency.
EXAMPLE 5.3

Application of the single-phase rectifier as a battery charger  A single-phase rectifier can be employed as a battery charger, as shown in Fig. 5.3(a). The battery capacity is 100 Wh, and the battery voltage is $E = 12$ V. The average charging current should be $I_{o(\text{av})} = 5$ A. The primary AC input voltage is $V_p = 120$ V (rms), 60 Hz, and the transformer has a turns ratio of $n = 2:1$.

(a) Calculate the angle $\delta$ over which the diode conducts, the current-limiting resistance $R$, the power rating $P_R$ of $R$, the charging time $h$ in hours, the rectification efficiency $\eta_R$, and the peak inverse voltage PIV of the diode.

(b) Use PSpice/SPICE to plot $P_o(\text{ac})$ and $P_o(\text{dc})$ as a function of time. Assume model parameters of diode D1N4148:

\begin{align*}
I_S &= 2.682 \text{mA} \\
C_J0 &= 4 \mu\text{F} \\
M &= 0.3333 \\
V_J &= 0.5 \\
B_V &= 100 \\
I_{BV} &= 100 \mu\text{A} \\
T_T &= 11.54 \text{us}
\end{align*}

SOLUTION

(a) If the secondary input voltage is $v_S > E$, diode D$_1$ will conduct. The angle $\theta_1$ at which the diode starts conducting can be found from the condition

$$V_m \sin \theta_1 = E$$

or

$$\theta_1 = \sin^{-1} \left( \frac{E}{V_m} \right)$$

(b) Waveforms

FIGURE 5.3  Battery charger
Diode D₁ will be turned off when \( v_S \leq E \) at

\[ \theta_2 = \pi - \theta_1 \]

The charging current \( i_0 \), which is shown in Fig. 5.3(b), can be found from

\[ i_0 = \frac{v_S - E}{R} = \frac{V_m \sin \theta - E}{R} \quad \text{for} \quad \theta_1 \leq \theta \leq \theta_2 \]  \hspace{1cm} (5.21)

Since \( V_s = V_p/2 = 120/2 = 60 \) V,

\[ V_m = \sqrt{2} V_s = \sqrt{2} \times 60 = 84.85 \) V

From Eq. (5.20), \( \theta_1 = \sin^{-1} (12/84.85) = 8.13^\circ \), or 0.1419 rad. Thus,

\[ \theta_2 = 180 - 8.13 = 171.87^\circ \]

The interval over which the diode will conduct is called the conduction angle and is given by

\[ \delta = \theta_2 - \theta_1 = 171.87 - 8.13 = 163.74^\circ \]

The average charging current \( I_{D(\text{av})} \) is

\[ I_{D(\text{av})} = \frac{1}{2\pi} \int_{\theta_1}^{\theta_2=\pi-\theta_1} \frac{V_m \sin \theta - E}{R} d\theta \]

\[ = \frac{1}{2\pi R} \left( 2V_m \cos \theta_1 + 2E\theta_1 - \pi E \right) \]  \hspace{1cm} (5.22)

which gives the limiting resistance \( R \) as

\[ R = \frac{1}{2\pi I_{D(\text{av})}} \left( 2V_m \cos \theta_1 + 2E\theta_1 - \pi E \right) \]

\[ = \frac{1}{2\pi \times 5} \left( 2 \times 84.85 \cos 8.13^\circ + 2 \times 12 \times 0.1419 - \pi \times 12 \right) = 4.26 \Omega \]

The rms battery current \( I_{o(\text{rms})} \) is

\[ I_{o(\text{rms})}^2 = \frac{1}{2\pi} \int_{\theta_1}^{\theta_2=\pi-\theta_1} \frac{(V_m \sin \theta - E)^2}{R^2} d\theta \]

\[ = \frac{1}{2\pi R^2} \left[ \left( \frac{V_m^2}{2} + E^2 \right)(\pi - 2\theta_1) + \frac{V_m^2}{2} \sin 2\theta_1 - 4V_m E \cos \theta_1 \right] \]  \hspace{1cm} (5.23)

\[ = 67.31 \) A²

which gives \( I_{o(\text{rms})} = \sqrt{67.31} = 8.2 \) A. The power rating of \( R \) is

\[ P_R = I_{o(\text{rms})}^2 R = 8.2^2 \times 4.26 = 286.4 \) W
The power delivered to the battery $P_{o(dc)}$ is

$$P_{o(dc)} = EI_{o(av)} = 12 \times 5 = 60 \text{ W}$$

For 100 Wh,

$$hP_{o(dc)} = 100$$

or

$$h = \frac{100}{P_{o(dc)}} = \frac{100}{60} = 1.667 \text{ h}$$

The rectification efficiency $\eta_R$ is

$$\eta_R = \frac{\text{Power delivered to the battery}}{\text{Total input power}} = \frac{P_{o(dc)}}{P_{o(dc)} + P_R} = \frac{60}{60 + 286.4} = 17.32\%$$

The peak inverse voltage PIV of the diode is

$$\text{PIV} = V_m + E = 84.85 + 12 = 96.85 \text{ V}$$

(b) The battery charger circuit for PSpice simulation is shown in Fig. 5.4. Since inductance is proportional to the square of the number of turns, the primary and the secondary leakage inductances of the input transformer are selected with a ratio of $2^2$ (or 4) to 1. That is, $L_1 = 40 \text{ mH}$ and $L_2 = 10 \text{ mH}$ for a linear transformer.

The PSpice plots of $I_{o(rms)}$, $P_{o(dc)}$, and $P_{o(rms)}$ are shown in Fig. 5.5, which gives $I_{o(rms)} \approx 7.3 \text{ A}$, $P_{o(dc)} \approx 53.5 \text{ W}$, and $P_{o(rms)} \approx 86.7 \text{ W}$. The value of $I_{o(rms)}$ is equal to the rms current through resistance $R$—that is, $I(R)$. These plots reach their steady-state values after a transient interval of approximately 80 ms.

![FIGURE 5.4 Battery charger circuit for PSpice simulation](image1)

![FIGURE 5.5 PSpice plots for Example 5.3](image2)

### 5.2.2 Single-Phase Full-Wave Center-Tapped Rectifier

For a half-wave rectifier, the average (or DC) voltage is only 0.318 $V_m$. A full-wave rectifier has double this output voltage, and it can be constructed by combining two half-wave rectifiers, as shown in Fig. 5.6(a). Since $v_S$ is positive from $\omega t = 0$ to $\pi$ and negative from $\omega t = \pi$ to $2\pi$, the operation of the rectifier can be divided into two intervals: interval 1 and interval 2.
Interval 1 is the interval $0 \leq \omega t \leq \pi$ during the positive half-cycle of the input voltage. Diode $D_2$ is reverse biased and behaves like an open circuit, as shown in Fig. 5.6(b). The peak inverse voltage PIV of diode $D_2$ is $2V_m$. Diode $D_1$ conducts and behaves like a short circuit. The half-secondary voltage $v_S = V_m \sin \omega t$ appears across the load resistance $R_L$. That is, the output voltage becomes

$$v_O = V_m \sin \omega t \quad \text{for } 0 \leq \omega t \leq \pi$$

Interval 2 is the interval $\pi \leq \omega t \leq 2\pi$ during the negative half-cycle of the input voltage. Diode $D_1$ is reverse biased and behaves like an open circuit, as shown in Fig. 5.6(c). The peak inverse voltage PIV of diode $D_1$ is also $2V_m$. Diode $D_2$ conducts and behaves like a short circuit. The negative of the half-secondary voltage $v_S = V_m \sin \omega t$ appears across the load resistance $R_L$. That is, the output voltage becomes

$$v_O = -V_m \sin \omega t \quad \text{for } \pi \leq \omega t \leq 2\pi$$

The instantaneous output voltage $v_O$ during interval 2 is identical to that for interval 1. The waveforms for the input and output voltages are shown in Fig. 5.6(d). Now we need to find the average voltage and the ripple content. Similar to that of the half-wave rectifier, the output voltage of a full-wave rectifier can be viewed as consisting of two components: ripple voltage and average voltage. The instantaneous ripple voltage $v_r$, which is the difference between $v_O$ and $V_{o(av)}$, is shown in Fig. 5.6(e). The average output voltage $V_{o(av)}$ with two identical positive pulses can be found from the following equation:

$$V_{o(av)} = \frac{2}{2\pi} \int_0^\pi v_O \, d(\omega t) = \frac{2}{2\pi} \int_0^\pi V_m \sin \omega t \, d(\omega t) = \frac{2V_m}{\pi} = 0.636V_m \quad (5.25)$$

It is twice the average output voltage of a half-wave rectifier, $V_{o(av)} = 0.318V_m$. Therefore, the average load current $I_{o(av)}$ for a resistive load can be found from Eq. (5.25):

$$I_{o(av)} = \frac{V_{o(av)}}{R_L} = \frac{2V_m}{\pi R_L} = \frac{0.636V_m}{R_L} \quad (5.26)$$

The rms output voltage $V_{o(rms)}$ is given by

$$V_{o(rms)} = \left[ \frac{2}{2\pi} \int_0^\pi v_O^2 \, d(\omega t) \right]^{1/2} = \left[ \frac{2}{2\pi} \int_0^\pi V_m^2 \sin^2 \omega t \, d(\omega t) \right]^{1/2}$$

$$= \frac{V_m}{\sqrt{2}} = 0.707V_m$$

compared to $V_{o(rms)} = 0.5V_m$ for a half-wave rectifier. Therefore, the rms load current $I_{o(rms)}$ is given by

$$I_{o(rms)} = \frac{V_{o(rms)}}{R_L} = \frac{0.707V_m}{R_L} \quad (5.28)$$

To find the ripple factor, we have to find the amount of ripple content. The instantaneous ripple voltage $v_r$, which is shown in Fig. 5.6(e), can be expressed as

$$v_r = \begin{cases} 
  v_S - V_{o(av)} = V_m \sin \omega t - V_{o(av)} & \text{for } 0 < \omega t < \pi \\
  -V_m \sin \omega t - V_{o(av)} & \text{for } \pi \leq \omega t \leq 2\pi 
\end{cases}$$
Let $V_{r(rms)}$ be the rms ripple voltage. Then $V_{r(rms)}$ can be related to $V_{o(av)}$ and $V_{o(rms)}$ by the mean square values. That is,

$$V_{r(rms)}^2 + V_{o(rms)}^2 = V_{o(av)}^2$$

or

$$V_{r(rms)}^2 = V_{o(rms)}^2 - V_{o(av)}^2$$  \hspace{1cm} (5.29)

Substituting $V_m$ from Eq. (5.25) into Eq. (5.27), we get $V_{o(rms)} = \pi V_{o(av)}/(2\sqrt{2})$, which, when substituted into Eq. (5.29), gives

$$V_{r(rms)} = \left[ \frac{\pi^2}{8} V_{o(av)}^2 - V_{o(av)}^2 \right]^{1/2} = V_{o(av)} \left[ \frac{\pi^2}{8} - 1 \right]^{1/2} = 0.483 V_{o(av)}$$  \hspace{1cm} (5.30)

which is much less than $V_{r(rms)} = 1.21 V_{o(av)}$ for a half-wave rectifier.

The ripple factor RF of the output voltage, which is a measure of the ripple content, can be found from

$$RF = \frac{V_{r(rms)}}{V_{o(av)}} = \frac{0.483 V_{o(av)}}{V_{o(av)}} = 0.483, \text{ or } 48.3\%$$  \hspace{1cm} (5.31)
which is much lower than $RF = 1.21 = 121\%$ for a half-wave rectifier.

The $AC$ output power $P_{o(ac)}$ is the average power and is defined as

$$P_{o(ac)} = \frac{1}{2\pi} \int_0^{2\pi} i_D^2 R_L d(\omega t) = I_{o(rms)}^2 R_L = V_{o(rms)} I_{o(rms)} = \frac{V_m^2}{2R_L} \tag{5.32}$$

If we assume there is no power loss in the rectifier, the input power can be found from

$$P_{in} = P_{o(ac)} = \left(\frac{V_m}{\sqrt{2}}\right)^2 \frac{1}{R_L} \tag{5.33}$$

The $DC$ output power $P_{o(dc)}$ can be found from

$$P_{o(dc)} = V_{o(av)} I_{o(av)} = \frac{4V_m^2}{\pi^2 R_L} \tag{5.34}$$

It is generally smaller than $P_{o(ac)}$. The ratio of $P_{o(dc)}$ to $P_{o(ac)}$, which is the rectification efficiency $\eta_R$, can be found from

$$\eta_R = \frac{P_{o(dc)}}{P_{o(ac)}} = \frac{V_{o(av)} I_{o(av)}}{V_{o(rms)} I_{o(rms)}} = \frac{(2V_m/\pi)^2/R_L}{(V_{oav}/\sqrt{2})^2/R_L} = \frac{8}{\pi^2} = 81\% \tag{5.35}$$

which is twice the value of $\eta_R = 40.5\%$ for a half-wave rectifier.

If we assume there is no power loss in the rectifier, the input power factor can be found from

$$PF = \frac{P_{o(ac)}}{2V_m I_s} = \frac{(V_m/\sqrt{2})^2/R_L}{2 \times (V_m/\sqrt{2})(V_m/2R_L)} = \frac{\sqrt{2}}{2} = 0.7071 \tag{5.36}$$

**NOTE** This numerical value of $\eta_R = 81\%$ is valid only for the single-phase full-wave rectifier.

The peak inverse voltage $PIV$ of the diodes is $2V_m$. A full-wave rectifier develops twice the average output voltage of a half-wave rectifier for the same peak secondary voltage; however, it requires a center-tapped transformer. This circuit is suitable for low-power applications only—typically tens of watts.

**EXAMPLE 5.4**

Finding the performance parameters of a single-phase full-wave rectifier The single-phase full-wave center-tapped rectifier of Fig. 5.6(a) is supplied from a 120-V, 60-Hz source through a step-down center-tapped transformer with turns ratio $n = 10:2$. The load resistance $R_L$ is 5 $\Omega$. Determine (a) the average output voltage $V_{o(av)}$, (b) the average load current $I_{o(av)}$, (c) the rms load voltage $V_{o(rms)}$, (d) the rms load current $I_{o(rms)}$, (e) the ripple factor RF of the output voltage, (f) the rms ripple voltage $V_{r(rms)}$, (g) the average diode current $I_{D(av)}$, (h) the rms diode current $I_{D(rms)}$, (i) the peak inverse voltage $PIV$ of the diodes, (j) the average output power $P_{o(ac)}$, (k) the DC output power $P_{o(dc)}$, (l) the frequency $f_r$ of the output ripple voltage, and (m) the input power factor $PF$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
The rms voltage of the transformer primary is $V_p = 120$ V. From Eq. (5.15) the rms voltage of the transformer secondary is $V_s = 2V_p/n = 120 \times 2/10 = 24$ V. The rms voltage of the transformer half-secondary is $V_s = 24/2 = 12$ V. The peak voltage of each half-secondary is

$$V_m = \sqrt{2} \times 12 = 16.97 \text{ V}$$

(a) From Eq. (5.25),

$$V_{o(\text{av})} = 0.636V_m = 0.636 \times 16.97 = 10.8 \text{ V}$$

(b) From Eq. (5.26),

$$I_{o(\text{av})} = \frac{V_{o(\text{av})}}{R_L} = \frac{10.8}{5} = 2.16 \text{ A}$$

(c) From Eq. (5.27),

$$V_{o(\text{rms})} = 0.707V_m = 0.707 \times 16.97 = 12 \text{ V}$$

(d) From Eq. (5.28),

$$I_{o(\text{rms})} = \frac{V_{o(\text{rms})}}{R_L} = \frac{12}{5} = 2.4 \text{ A}$$

(e) From Eq. (5.31), RF = 0.483, or 48.3%.

(f) From Eq. (5.30),

$$V_{e(\text{rms})} = 0.483V_{o(\text{av})} = 0.483 \times 10.8 = 5.22 \text{ V}$$

(g) Since the average load current is supplied by two diodes, the average diode current $I_{D(\text{av})}$ will be one-half of the load current. That is, $I_{D(\text{av})} = I_{o(\text{av})}/2 = 2.16/2 = 1.08 \text{ A}$. 

(h) Since the load current is shared by two diodes, the rms load current $I_{o(\text{rms})}$ will be $\sqrt{2}$ times the rms diode current. That is, $I_{D(\text{rms})} = I_{o(\text{rms})}/\sqrt{2} = 2.4/\sqrt{2} = 1.7 \text{ A}$. 

(i) PIV = $2V_m = 2 \times 16.97 = 33.94 \text{ V}$. 

(j) From Eq. (5.32),

$$P_{o(\text{ac})} = I_{o(\text{rms})}^2R_L = (2.4)^2 \times 5 = 28.8 \text{ W}$$

(k) From Eq. (5.34),

$$P_{o(\text{dc})} = V_{o(\text{av})}I_{o(\text{av})} = 10.8 \times 2.16 = 23.33 \text{ W}$$

(l) The output voltage contains two pulses per cycle of the input voltage. That is, $f_i = 2f = 2 \times 60 = 120 \text{ Hz}$. 

(m) $V_s = 12$ V, $I_s = I_{o(\text{rms})}/\sqrt{3} = 1.7 \text{ A}$, and $P_m = P_{o(\text{ac})} = 28.8 \text{ W}$. 

From Eq. (5.36),

$$\text{PF} = \frac{P_m}{2V_sI_s} = \frac{28.8}{2 \times 12 \times 1.7} = 0.7071$$
EXAMPLE 5.5

Fourier components of the output voltage of a single-phase full-wave rectifier  The single-phase full-wave rectifier of Fig. 5.7 is supplied from a 120-V, 60-Hz source through a step-down center-tapped transformer with a turns ratio of \( n = 10:2 \).

(a) Express the instantaneous output voltage \( v_O(t) \) by a Fourier series.

(b) Use PSpice/SPICE to calculate the harmonic components of the output voltage. Assume default diode parameters.

NOTE:  The voltage-controlled voltage source representation of the input transformer will give only the correct input and output voltage waveforms, but not the correct value of the input current. To get the actual input current, we should consider the power balances such that the primary volt-amp is equal to the secondary volt-amp, \( V_p I_p = 2 V_s I_s \). This will require connecting two back-to-back current-controlled current sources (not shown) across the primary side [2].

SOLUTION

(a) \( 2V_s = 120 \times 2/10 = 24 \text{ V} \), and \( V_s = 12 \text{ V} \). \( V_m = \sqrt{2} V_s = \sqrt{2} \times 12 = 16.97 \text{ V} \). The output voltage \( v_O \) can be described by

\[
v_O = \begin{cases} 
V_m \sin \omega t & \text{for } 0 \leq \omega t \leq \pi \\
-V_m \sin \omega t & \text{for } \pi \leq \omega t \leq 2\pi 
\end{cases}
\]

which can be expressed by a Fourier series as

\[
v_O(\theta) = V_{O(\text{av})} + \sum_{n=1,2,...}^{\infty} (a_n \sin n\theta + b_n \cos n\theta) \quad \text{where } \theta = \omega t = 2\pi f t = 377t
\]

\[
V_{O(\text{av})} = \frac{1}{2\pi} \int_0^{2\pi} v_O \, d\theta = \frac{2}{2\pi} \int_0^{\pi} V_m \sin \theta \, d\theta = \frac{2V_m}{\pi}
\]

FIGURE 5.7  Single-phase full-wave rectifier circuit for PSpice simulation
When the values of $a_n$ and $b_n$ are inserted into Eq. (5.18), the expression for the instantaneous output voltage $v_O$ becomes

$$v_O(t) = \frac{2V_m}{\pi} - \frac{4V_m}{3\pi} \cos 2ot - \frac{4V_m}{15\pi} \cos 4ot - \frac{4V_m}{35\pi} \cos 6ot - \cdots$$

$$- \frac{4V_m}{(2n-1)(2n+1)} \cos 2n\omega t \quad \text{for } n = 1, 2, 3, \ldots, \infty \quad (5.37)$$

where $V_m = \sqrt{2} \times 120 = 16.97$ V and $\omega = 2\pi \times 60 = 377$ rad/s.

Equation (5.25) gives $V_{O(\text{av})} = 2V_m/\pi = 2 \times 16.97/\pi = 10.8$ V. From Eq. (5.37), we can find the peak magnitudes of harmonic components are

$$V_{2(\text{peak})} = \frac{4V_m}{3\pi} = 4 \times \frac{16.97}{3\pi} = 7.2 \text{ V}$$

$$V_{4(\text{peak})} = \frac{4V_m}{15\pi} = 4 \times \frac{16.97}{15\pi} = 1.44 \text{ V}$$

$$V_{6(\text{peak})} = \frac{4V_m}{35\pi} = 4 \times \frac{16.97}{35\pi} = 0.617 \text{ V}$$

$$V_{8(\text{peak})} = \frac{4V_m}{63\pi} = 4 \times \frac{16.97}{63\pi} = 0.343 \text{ V}$$

Note that the output voltage $v_O$ contains only even harmonics, and the second harmonic is the dominant one at a ripple frequency of $f_r = 2f = 120$ Hz.

(b) The single-phase full-wave center-tapped rectifier circuit for PSpice simulation is shown in Fig. 5.7. The center-tapped transformer is modeled by a voltage-controlled voltage source.

The PSpice results of Fourier analysis are as follows. The hand-calculated values are shown in parentheses on the right.
5.2.3 Single-Phase Full-Wave Bridge Rectifier

A single-phase full-wave bridge rectifier is shown in Fig. 5.8(a). It requires four diodes. The advantages of this rectifier are that it requires no transformer in the input side and the PIV rating of the diodes is $V_m$. The disadvantages are that it does not provide electrical isolation and it requires more diodes than the center-tapped version. However, an input transformer is normally used to satisfy the output voltage requirement.

Since $v_S$ is positive from $\pi/2$ to $\pi$ and negative from $\pi$ to $2\pi$, the circuit operation can be divided into two intervals: interval 1 and interval 2.

Interval 1 is the interval $0 \leq \omega t \leq \pi$ during the positive half-cycle of the input voltage $v_S$. Diodes $D_3$ and $D_4$ are reverse biased, as shown in Fig. 5.8(b). The peak inverse voltage PIV of diodes $D_3$ and $D_4$ is $V_m$. Diodes $D_1$ and $D_2$ conduct and behave like short circuits. The input voltage $v_S = V_m \sin \omega t$ appears across the load resistance $R_L$. That is, the output voltage becomes

$$v_O = V_m \sin \omega t \quad \text{for} \quad 0 \leq \omega t \leq \pi$$

Interval 2 is the interval $\pi \leq \omega t \leq 2\pi$ during the negative half-cycle of the input voltage $v_S$. Diodes $D_1$ and $D_2$ are reverse biased, as shown in Fig. 5.8(c). The peak inverse voltage PIV of diodes $D_1$ and $D_2$ is $V_m$. Diodes $D_3$ and $D_4$ conduct and behave like short circuits. The negative of voltage $v_S = V_m \sin \omega t$ appears across the load resistance $R_L$. That is, the output voltage becomes

$$v_O = -V_m \sin \omega t \quad \text{for} \quad \pi \leq \omega t \leq 2\pi$$

The waveforms for the input and output voltages are shown in Fig. 5.8(d). The output voltage will be reduced due to DC diode drop as shown by the dotted lines. The output ripple voltage is shown in Fig. 5.8(e). The equations that were derived earlier for a single-phase full-wave center-tapped transformer are also valid for the bridge rectifier.

### FOURIER COMPONENTS OF TRANSIENT RESPONSE $V(\mathrm{RL}:2)$

<table>
<thead>
<tr>
<th>HARMONIC NO</th>
<th>FREQUENCY (HZ)</th>
<th>FOURIER COMPONENT</th>
<th>NORMALIZED COMPONENT</th>
<th>PHASE (deg)</th>
<th>NORMALIZED PHASE (deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6.000E+01</td>
<td>2.727E-02</td>
<td>1.000E+00</td>
<td>-8.789E+01</td>
<td>0.000E+00</td>
</tr>
<tr>
<td>2</td>
<td>1.200E+02</td>
<td>6.312E+00</td>
<td>2.315E+02</td>
<td>-9.018E+01</td>
<td>-2.289E+00</td>
</tr>
<tr>
<td>3</td>
<td>1.800E+02</td>
<td>2.705E-02</td>
<td>9.922E-01</td>
<td>-8.373E+01</td>
<td>4.161E+00</td>
</tr>
<tr>
<td>4</td>
<td>2.400E+02</td>
<td>1.199E+00</td>
<td>4.398E+01</td>
<td>-9.065E+01</td>
<td>-2.763E+00</td>
</tr>
<tr>
<td>5</td>
<td>3.000E+02</td>
<td>2.658E-02</td>
<td>9.750E-01</td>
<td>-7.975E+01</td>
<td>8.142E+00</td>
</tr>
<tr>
<td>6</td>
<td>3.600E+02</td>
<td>4.806E-01</td>
<td>1.763E+01</td>
<td>-9.477E+01</td>
<td>-3.882E+00</td>
</tr>
<tr>
<td>7</td>
<td>4.200E+02</td>
<td>2.576E-02</td>
<td>9.448E-01</td>
<td>-7.603E+01</td>
<td>1.186E+01</td>
</tr>
<tr>
<td>8</td>
<td>4.800E+02</td>
<td>2.478E-01</td>
<td>9.089E+00</td>
<td>-9.391E+01</td>
<td>-6.015E+00</td>
</tr>
<tr>
<td>9</td>
<td>5.400E+02</td>
<td>2.447E-02</td>
<td>8.973E-01</td>
<td>-7.254E+01</td>
<td>1.535E+01</td>
</tr>
</tbody>
</table>

**NOTE:** The calculated values do not take into account the diode voltage drops, whereas the PSpice simulation assumes a real diode characteristic. This accounts for the differences between the PSpice and the hand-calculated values.
If we assume there is no power loss in the rectifier, the input power factor can be found from

\[
PF = \frac{P_{o(ac)}}{V_s I_s} = \frac{(V_m/\sqrt{2})^2/R_L}{(V_m/\sqrt{2})(V_m/((\sqrt{2} R_L))} = 1.0
\]  

\[ (5.38) \]

**EXAMPLE 5.6**

**Performance parameters of a single-phase full-wave bridge rectifier** The single-phase full-wave bridge rectifier of Fig. 5.8(a) is supplied from a 120-V, 60-Hz source through a transformer with turns ratio \( n = 10:1 \). The load resistance \( R_L \) is 5 \( \Omega \). Determine (a) the average output voltage \( V_{o(\text{av})} \), (b) the average load current \( I_{o(\text{av})} \), (c) the rms load voltage \( V_{o(\text{rms})} \), (d) the rms load current \( I_{o(\text{rms})} \), (e) the ripple factor RF of the output voltage, (f) the rms ripple voltage \( V_{r(\text{rms})} \), (g) the average diode current \( I_{D(\text{av})} \), (h) the rms diode current \( I_{D(\text{rms})} \), (i) the peak inverse voltage PIV of the diode, (j) the average (or AC) output power \( P_{o(ac)} \), (k) the DC output power \( P_{o(dc)} \), (l) the frequency \( f_r \) of the output ripple voltage, and (m) the input power factor PF.
**SOLUTION**

The rms voltage of the transformer primary is $V_p = 120\, \text{V}$. From Eq. (5.15), the rms voltage of the transformer secondary is $V_s = V_p/n = 120/10 = 12\, \text{V}$. The peak voltage of the secondary is

$$V_m = \sqrt{2} \times 12 = 16.97\, \text{V}$$

(a) From Eq. (5.25),

$$V_{o(\text{av})} = 0.636 V_m = 0.636 \times 16.97 = 10.8\, \text{V}$$

(b) From Eq. (5.26),

$$I_{o(\text{av})} = \frac{V_{o(\text{av})}}{R_L} = \frac{10.8}{5} = 2.16\, \text{A}$$

(c) From Eq. (5.27),

$$V_{o(\text{rms})} = 0.707 V_m = 0.707 \times 16.97 = 12\, \text{V}$$

(d) From Eq. (5.28),

$$I_{o(\text{rms})} = \frac{V_{o(\text{rms})}}{R_L} = \frac{12}{5} = 2.4\, \text{A}$$

(e) From Eq. (5.31), $RF = 0.483$, or 48.3%.

(f) From Eq. (5.30),

$$V_{(\text{rms})} = 0.483 V_{o(\text{av})} = 0.483 \times 10.8 = 5.22\, \text{V}$$

(g) The load current flows through one of the top diodes ($D_1$ or $D_3$), the load, and then one of the bottom diodes ($D_2$ or $D_4$). Thus, the same current flows through two diodes, which are conducting. The time-average diode current $I_{D(\text{av})}$ will be one-half of the load current. That is, $I_{D(\text{av})} = I_{o(\text{av})}/2 = 2.16/2 = 1.08\, \text{A}$.

(h) The rms diode current $I_{D(\text{rms})}$ will be $1/\sqrt{2}$ times the rms load current. That is, $I_{D(\text{rms})} = I_{o(\text{rms})}/\sqrt{2} = 2.4/\sqrt{2} = 1.7\, \text{A}$

(i) $PIV = V_m = 16.97\, \text{V}$

(j) From Eq. (5.32),

$$P_{o(\text{ac})} = I_{o(\text{rms})}^2 R_L = (2.4)^2 \times 5 = 28.8\, \text{W}$$

(k) From Eq. (5.34),

$$P_{o(\text{dc})} = V_{o(\text{av})} I_{o(\text{av})} = 10.8 \times 2.16 = 23.33\, \text{W}$$

(l) $f_2 = 2f = 2 \times 60 = 120\, \text{Hz}$

(m) $V_s = 12\, \text{V}, I_s = I_{o(\text{rms})} = 2.4\, \text{A},$ and $P_{in} = P_{o(\text{ac})} = 28.8\, \text{W}$
Applications of Diodes

Transfer (output versus input) characteristic of a single-phase bridge rectifier

A single-phase bridge rectifier is shown in Fig. 5.9. The load resistance $R_L$ is 4.5 kΩ. The source resistance $R_s$ is 500 Ω.

(a) Determine the transfer characteristic ($v_O$ versus $v_S$) of the rectifier.

(b) Use PSpice/SPICE to plot the transfer characteristic for $v_S$ from 10 V to 10 V. Assume model parameters of diode D1N4148:

$\begin{align*}
&I_S=2.682N \\
&C_{JO}=4P \\
&M=.3333 \\
&V_J=.5 \\
&BV=100 \\
&IBV=100U \\
&TT=11.54N
\end{align*}$

SOLUTION

(a) $R_L = 4.5$ kΩ and $R_s = 500$ Ω. When the input voltage $v_S$ is positive, only diodes D1 and D2 conduct. The output voltage $v_O$ can be obtained by applying the voltage divider rule. That is,

$$v_O = \frac{v_S R_L}{R_L + R_s} = \frac{v_S (4.5 \text{ kΩ})}{4.5 \text{ kΩ} + 500 \text{ Ω}} = 0.9v_S \quad \text{for } v_S > 0$$

If the input voltage $v_S$ is negative, only diodes D3 and D4 conduct. The output voltage $v_O$ can be obtained from

$$v_O = \frac{-v_S R_L}{R_L + R_s} = \frac{-v_S (4.5 \text{ kΩ})}{4.5 \text{ kΩ} + 500 \text{ Ω}} = -0.9v_S \quad \text{for } v_S < 0$$

The transfer characteristic is shown in Fig. 5.10(a).

(b) The PSpice plot of $v_O$ against $v_S$ is shown in Fig. 5.10(b). The dead zone around 0 (between 0.82 V and −0.671 V) is due to the voltage drops across the diodes.

FIGURE 5.9 Single-phase bridge rectifier circuit for PSpice simulation

From Eq. (5.38),

$$PF = \left( \frac{P_{in}}{V_s \times I_s} \right) = \left( \frac{28.8}{12 \times 2.4} \right) = 1.0$$

NOTE: The results of Examples 5.4 and 5.6 are identical, except that the PIV of a bridge rectifier is PIV = $V_m = 16.97$ V whereas the PIV of a center-tapped rectifier is PIV = $2V_m = 33.94$ V for the same $V_{o(av)} = 10.8$ V. The bridge rectifier has the best power factor.
**EXAMPLE 5.8**

**Application of a single-phase bridge rectifier as an AC voltmeter** An AC voltmeter is constructed by using a DC meter and a bridge rectifier, as shown in Fig. 5.11(a). The meter has a resistance of \( R_m = 100 \Omega \), and its average current is \( I_m = 100 \text{ mA} \) for a full-scale deflection. The current-limiting resistance is \( R_s = 1 \text{ k}\Omega \).

(a) Determine the rms value of the AC input voltage \( V_s \) that will give a full-scale deflection if the input voltage \( v_S \) is sinusoidal.

(b) If this meter is used to measure the rms value of an input voltage with a triangular waveform, as shown in Fig. 5.11(b), calculate the necessary correction factor \( K \) to be applied to the meter reading.
SOLUTION

\[ V_{m} = 100 \ \Omega, \ R_s = 1 \ k\Omega, \ \text{and} \ I_m = 100 \ mA. \]

(a) The peak value \( V_m \) of a sinusoidal voltage is related to its rms value \( V_s \) by \( V_m = \sqrt{2} V_s \). The average meter voltage \( V_{m(\text{av})} \) can be found by applying the voltage divider rule between resistances \( R_s \) and \( R_m \):

\[
V_{m(\text{av})} = \frac{R_m}{R_s + R_m} V_{o(\text{av})}
\]

where \( V_s(\text{rms}) \) is the rms value of \( v_s \).

Using \( V_{o(\text{av})} = 2V_m / \pi \) from Eq. (5.25), we can find the average meter current \( I_{m(\text{av})} \) from

\[
I_{m(\text{av})} = \frac{V_{o(\text{av})}}{R_s + R_m} = \frac{1}{R_s + R_m} \frac{2V_m}{\pi} = \frac{2 \sqrt{2} V_s}{\pi (R_s + R_m)} \tag{5.39}
\]

The meter reading \( \theta_1 \), which is proportional to the average meter current \( I_{m(\text{av})} \), must measure the rms input voltage. That is,

\[
\theta_1 = K_1 I_{m(\text{av})} = V_s \tag{5.40}
\]

where \( K_1 \) is a meter scale factor. Substituting \( I_{m(\text{av})} \) from Eq. (5.39), we get \( K_1 \):

\[
K_1 = \frac{2 \sqrt{2} V_s}{\pi (R_s + R_m)}
\]

which gives the constant \( K_1 \) as

\[
K_1 = \frac{\pi (R_s + R_m)}{2 \sqrt{2}} = \frac{\pi (1 \times 10^3 + 100)}{2 \sqrt{2}} = 1221.8 \ \text{V/A} \tag{5.41}
\]

Therefore, using Eq. (5.40), we can find the rms input voltage \( V_s \) that will give the full-scale deflection:

\[
V_s = K_1 I_{m(\text{av})} = 1221.8 \times 100 \times 10^{-3} = 122.2 \ \text{V}
\]

(b) If a triangular waveform \( v_S \) with a peak value of \( V_m \) is applied to the bridge rectifier, the output voltage \( V_O \) is as shown in Fig. 5.11(c). The rms input voltage \( V_s \) of the triangular voltage with four identical triangular areas can be found from

\[
V_s = \left[ \frac{4}{2 \pi} \int_0^{\pi/2} \frac{V_m}{\pi/2} \theta \ d\theta \right]^{1/2} = \frac{V_m}{\sqrt{3}} \quad \text{(after the integration is completed)} \tag{5.42}
\]

The average output voltage \( V_{o(\text{av})} \) with four identical triangular areas can be found from

\[
V_{o(\text{av})} = \frac{4}{2 \pi} \int_0^{\pi/2} \frac{V_m}{\pi/2} \theta \ d\theta = \frac{V_m}{2} \tag{5.43}
\]
5.3 Output Filters for Rectifiers

In Eqs. (5.19) and (5.37), the rectifier output voltage \( v_O(t) \) has a DC component \( \frac{V_m}{\pi} \) or \( \frac{2V_m}{\pi} \) and other cosine components at various frequencies. The magnitudes of the cosine components are called the harmonics. The output should ideally be pure DC; these harmonics are undesirable. Filters are normally used to smooth out the output voltage. Since the input supply to these filters is DC, they are known as DC filters. Three types of DC filters are normally used: \( L \) filters, \( C \) filters, and \( LC \) filters. \( L \) filters and \( LC \) filters are generally used for high-power applications, such as DC power supplies. In integrated circuits, \( C \) filters are usually used.
5.3.1 **L Filters**

An inductor, which is an energy storage element, tries to maintain a constant current through the load so that the variation in the output voltage is low. Let us assume that an inductor with zero internal resistance is connected in series with the load resistance \( R_L \) of a bridge rectifier. This arrangement is shown in Fig. 5.12(a). At the ripple frequencies, the inductance offers a high impedance and the load current ripple is reduced. The equivalent circuits for the DC and harmonic components are shown in Fig. 5.12[(b) and (c)], respectively. The load impedance is given by

\[
Z = R_L + j(nωL) = \sqrt{R_L^2 + (nωL)^2} \angle \phi_n
\]  

where \( \phi_n = \tan^{-1}\left(\frac{nωL}{R_L}\right) \)

Dividing the frequency-dependent components of the output voltage \( v_O \) in Eq. (5.37) by the impedance \( Z \) of Eq. (5.47) gives the instantaneous load current \( i_O \):

\[
i_O(t) = I_{o(av)} - \frac{4V_m}{\pi} \left[ \sum_{n=2,4,6} \frac{1}{(n-1)(n+1)} \frac{\cos nωt - \phi_n}{\sqrt{R_L^2 + (nωL)^2}} \right]
\]  

where \( I_{o(av)} \) is obtained by dividing \( V_{o(av)} \) by the load resistance \( R_L \). That is,

\[
I_{o(av)} = \frac{V_{o(av)}}{R_L} = \frac{2V_m}{\pi R_L}
\]

Let us consider the first two harmonic components only, ignoring the higher-order ones. Let \( I_{o2(rms)} \) and \( I_{o4(rms)} \) be the rms currents of the second and fourth harmonic components, respectively. Since these currents are in rms values, the resultant rms ripple current \( I_{r(rms)} \) can be found by adding the mean square values of \( I_{o2(rms)} \) and \( I_{o4(rms)} \). That is,  

\[
I_{r(rms)}^2 = I_{o2(rms)}^2 + I_{o4(rms)}^2
\]
Using this relationship and dividing the peak values in Eq. (5.49) by $\sqrt{2}$ to convert to the rms values, we can get the rms ripple current $I_{\text{r(rms)}}$ of Eq. (5.50):

$$I_{\text{r(rms)}}^2 = \frac{1}{2} \left( \frac{4V_m}{\pi[R_L^2 + (2\omega L)^2]^{1/2}} \right)^2 \times \frac{1}{3}$$

$$+ \frac{1}{2} \left( \frac{4V_m}{\pi[R_L^2 + (4\omega L)^2]^{1/2}} \right)^2 \times \frac{1}{15} + \cdots \quad (5.51)$$

**EXAMPLE 5.9**

**Designing an output L filter** The single-phase bridge rectifier of Fig. 5.12(a) is directly supplied from a 120-V, 60-Hz source without any input transformer. The average output voltage is $V_{\text{o(av)}} = 158$ V. The load resistance is $R_L = 500$ $\Omega$.

(a) Design an L filter so that the rms ripple current $I_{\text{r(rms)}}$ is limited to less than 5% of $I_{\text{o(av)}}$. Assume that the second harmonic $I_{\text{o2(rms)}}$ is the dominant one and that the effects of higher-order harmonics are negligible.

(b) Use PSpice/SPICE to check your design by plotting the output current. Use diode default parameters of 1N4148 diodes.

**SOLUTION**

(a) Since $V_m = \sqrt{2}V_s = \sqrt{2} \times 120 = 169.7$ V,

$$I_{\text{o(av)}} = \frac{V_{\text{o(av)}}}{R_L} = \frac{158}{500} = 316 \text{ mA}$$

$$I_{\text{r(rms)}} = 5\% \text{ of } I_{\text{o(av)}} = 0.05 \times 316 \text{ mA} = 15.8 \text{ mA}$$

Assume that the ripple current is approximately sinusoidal. Then, the peak ripple current is $\sqrt{2}$ times the value of $I_{\text{r(rms)}}$. That is,

$$I_{\text{r(peak)}} = \sqrt{2} \times I_{\text{r(rms)}} = \sqrt{2} \times 15.8 \text{ mA} = 22.34 \text{ mA}$$

The peak-to-peak ripple current $I_{\text{r(pp)}}$ is twice the value of $I_{\text{r(peak)}}$. Thus,

$$I_{\text{r(pp)}} = 2 \times I_{\text{r(peak)}} = 2 \times 22.34 \text{ mA} = 44.69 \text{ mA}$$

Let us consider only the lowest-order harmonic—that is, $n = 2$. Equation (5.51) yields

$$I_{\text{r(rms)}} \approx I_{\text{o2(rms)}} = \frac{4V_m}{\sqrt{2\pi[R_L^2 + (2\omega L)^2]^{1/2}}} \times \frac{1}{3}$$

The ripple factor $RF_i$ of the output current is given by

$$RF_i = \frac{I_{\text{r(rms)}}}{I_{\text{o2(rms)}}} \approx \frac{I_{\text{o2(rms)}}}{I_{\text{o2(rms)}}} = \frac{4V_m}{\sqrt{2\pi[R_L^2 + (2\omega L)^2]^{1/2}}} \times \frac{1}{3} \times \frac{\pi R_L}{2V_m}$$

$$= \frac{4/(\sqrt{2} \times 3 \times 2)}{\sqrt{1 + (2\omega L/R_L)^2}} = \frac{0.4714}{\sqrt{1 + (2\omega L/R_L)^2}}$$

$$= \frac{0.4714}{\sqrt{1 + (2\omega 500/500)^2}} = \frac{0.4714}{\sqrt{1 + (2\omega 500)^2}}$$

$$= \frac{0.4714}{\sqrt{1 + (2\omega 500)^2}}$$
which can be solved to find the value of \( L \) for the known values of \( R_L = 500 \, \Omega \), \( f = 60 \, \text{Hz} \), and \( \text{RF}_1 = 5\% = 0.05 \). That is,

\[
0.05 = \frac{0.4714}{\sqrt{1 + (2\omega L/R_L)^2}}
\]

\[
0.4714^2 = (0.05)^2 \times \left[ 1 + \left( \frac{2 \times 2 \times 60 \times \pi L}{500} \right)^2 \right]
\]

\[ L = 6.22 \, \text{H} \]

(b) The bridge rectifier circuit with an \( L \) filter for PSpice simulation is shown in Fig. 5.13. PSpice allows us to find the current through resistors, \( I(R_L) \). It is not necessary to have a fictitious voltage source \( V_X = 0 \).

The PSpice plot of load current \( i_O \), shown in Fig. 5.14, gives the peak-to-peak ripple current as \( I_{r(pp)} = 166.67 - 150.82 = 15.85 \, \text{mA} \), compared to the calculated value of \( I_{r(pp)} = 22.34 \, \text{mA} \). The difference between the values is a result of neglecting the higher-order harmonics in determining the value of \( L \) and also the fact that PSpice uses real diodes rather than ideal ones with zero forward resistance. With ideal diodes, PSpice would give 27.9 mA. The DC current from PSpice is \( I_{r(\text{av})} = (166.61 + 150.82)/2 = 158.72 \, \text{mA} \), which is below the calculated value of 316 mA. This is caused by the fact that the effect of inductor \( L \) was not included in the calculated values.
5.3.2 C Filters

A capacitor is also an energy storage element; it tries to maintain a constant voltage, thereby preventing any change in voltage across the load. A capacitor C can be connected across the load to maintain a continuous output voltage $v_O$, as shown in Fig. 5.15(a) [3]. Under steady-state conditions, the capacitor will have a finite initial voltage. When the magnitude of the instantaneous supply voltage $v_S$ is greater than that of the instantaneous capacitor voltage $v_C$, the diodes (D1 and D2 or D3 and D4) will conduct and the capacitor will be charged from the supply. However, if the magnitude of the voltage $v_S$ falls below that of the instantaneous capacitor voltage $v_C$, the diodes (D1 and D2 or D3 and D4) will be reverse biased and the capacitor C will discharge through the load resistance $R_L$. The capacitor voltage $v_C$ will vary between a minimum value $V_o(\text{min})$ and a maximum value $V_o(\text{max})$. The waveforms of the output voltage $v_O$ and ripple voltage $v_r$ are shown in Fig. 5.15(b). If $f$ is the supply frequency, the period of the input voltage is $T = 1/f$.

For a single-phase half-wave rectifier, the period of the output ripple voltage is the same as the period $T$ of the supply voltage. However, for a single-phase full-wave rectifier, the period of the output ripple voltage is $T/2$. The output operation can be divided into two intervals: interval 1 for charging and interval 2 for discharging.

The equivalent circuit during charging is shown in Fig. 5.15(c). The capacitor charges almost instantaneously to the supply voltage $v_S$. The capacitor C will be charged approximately to the peak supply voltage $V_m$, so $v_C(\omega t = \pi/2) = V_m$. Figure 5.15(d) shows the equivalent circuit during discharging. The capacitor discharges exponentially through $R_L$. When one of the diode pairs is conducting, the

![Figure 5.15](image-url)
TABLE 5.1 Terms for measuring output ripple voltage

<table>
<thead>
<tr>
<th>Definition of Terms</th>
<th>Relationship</th>
</tr>
</thead>
<tbody>
<tr>
<td>The peak value of output voltage</td>
<td>( V_{\text{omax}} = V_m )</td>
</tr>
<tr>
<td>The peak-to-peak output ripple voltage</td>
<td>( V_{\text{pp}} = V_{\text{omax}} - V_{\text{omin}} = V_m - V_{\text{omin}} )</td>
</tr>
<tr>
<td>The ripple factor of the output voltage</td>
<td>( RF = \frac{V_{\text{pp}}}{V_m} = \frac{V_m - V_{\text{omin}}}{V_m} = 1 - \frac{V_{\text{omin}}}{V_m} )</td>
</tr>
<tr>
<td>The minimum value of output voltage</td>
<td>( V_{\text{omin}} = V_m(1 - RF) )</td>
</tr>
</tbody>
</table>

capacitor C draws a pulse of charging current from the AC supply, as shown in Fig. 5.15(e). As a result, the rectifier generates harmonic currents into the AC supply. For high-power applications, an input filter is normally required to reduce the amount of harmonic injection into the AC supply. Thus, a rectifier with a \( C \) filter is used only for low-power applications [4].

The output ripple voltage, which is the difference between maximum voltage \( V_{\text{omax}} \) and the minimum voltage \( V_{\text{omin}} \), can be specified in different ways, as shown in Table 5.1.

During the charging interval, under steady-state conditions the capacitor charges from \( V_{\text{omin}} \) to \( V_m \).

Let us assume that at an angle \( \alpha \) (rad/s), the positive input voltage is equal to the minimum capacitor voltage \( V_{\text{omin}} \) at the end of the capacitor discharge. As the input voltage rises sinusoidally from zero to \( V_m \), at the first cycle the angle \( \alpha \) can be determined from

\[
V_{\text{omin}} = V_m \sin(\alpha) \quad \text{or} \quad \alpha = \sin^{-1}\left( \frac{V_{\text{omin}}}{V_m} \right)
\]  

(5.53)

By redefining the time origin (\( \omega t = 0 \)) at \( \pi/2 \), as the beginning of interval 1, we can deduce the discharging current from

\[
\frac{1}{C} \int i_0 \, dt - v_C(t = 0) + R_L i_O = 0
\]

which, with an initial condition of \( v_C(\omega t = 0) = V_m \), gives

\[
i_O = \frac{V_m}{R_L} e^{-t/R_L C} \quad \text{for} \ 0 \leq t \leq t_d
\]

The instantaneous output (or capacitor) voltage \( v_O \) during the discharging period can be found from

\[
v_O(t) = R_L i_O = V_m e^{-t/R_L C}
\]  

(5.54)

From Fig. 5.15(b), we can find the discharging time \( t_d \) or the discharging angle \( \beta \) (rad/s) as

\[
\omega t_d = \beta = \frac{\pi}{2} + \alpha \quad \text{for a full-wave rectifier}
\]  

(5.55a)

\[
= \frac{3\pi}{2} + \alpha \quad \text{for a half-wave rectifier}
\]  

(5.55b)

At \( t = t_d \), \( v_O(t) \) in Eq. (5.54) becomes equal to \( V_{\text{omin}} \), and we can relate \( t_d \) to \( V_{\text{omin}} \) by

\[
v_O(t = t_d) = V_{\text{omin}} = V_m e^{-t_d/R_L C}
\]  

(5.56)
which gives the discharging time \( t_d \) as

\[
t_d = R_L C \ln \left( \frac{V_m}{V_{o(min)}} \right) \quad (5.57)
\]

Equating \( t_d \) in Eq. (5.57) to \( t_d \) in Eq. (5.55), we get

\[
\omega R_L C \ln \left( \frac{V_m}{V_{o(min)}} \right) = \frac{\pi}{2} + \alpha = \frac{\pi}{2} + \sin^{-1} \left( \frac{V_{o(min)}}{V_m} \right) \quad \text{for a full-wave rectifier} \quad (5.58a)
\]

\[
= \frac{3\pi}{2} + \alpha = \frac{3\pi}{2} + \sin^{-1} \left( \frac{V_{o(min)}}{V_m} \right) \quad \text{for a half-wave rectifier} \quad (5.58b)
\]

Therefore, the filter capacitor \( C \) can be found from

\[
C = \frac{\pi/2 + \sin^{-1} (V_{o(min)}/V_m)}{\omega R_L \ln (V_m/V_{o(min)})} \quad \text{for a full-wave rectifier} \quad (5.59a)
\]

\[
= \frac{3\pi/2 + \sin^{-1} (V_{o(min)}/V_m)}{\omega R_L \ln (V_m/V_{o(min)})} \quad \text{for a half-wave rectifier} \quad (5.59b)
\]

Redefining the time origin \((\omega t = 0)\) at \( \pi/2 \) when the discharging interval begins, we can find the average output voltage \( V_{o(av)} \) from

\[
V_{o(av)} = \frac{V_m}{\pi} \left[ \int_0^\beta e^{-\omega t/R_L C} d(\omega t) + \int_0^{\beta} \cos (\omega t) d(\omega t) \right]
\]

\[
= \frac{V_m}{\pi} \left[ \omega R_L C (1 - e^{-\beta/\omega R_L C}) + \sin \beta \right] \quad \text{for a full-wave rectifier} \quad (5.60)
\]

\[
V_{o(av)} = \frac{V_m}{2\pi} \left[ \int_0^\beta e^{-\omega t/R_L C} d(\omega t) + \int_{\beta}^{2\pi} \cos (\omega t) d(\omega t) \right]
\]

\[
= \frac{V_m}{2\pi} \left[ \omega R_L C (1 - e^{-\beta/\omega R_L C}) + \sin \beta \right] \quad \text{for a half-wave rectifier} \quad (5.61)
\]

The equations just given for \( C \) in Eq. (5.59) and \( V_{o(av)} \) in Eq. (5.60) are nonlinear. We can derive simple explicit expressions for the ripple voltage in terms of the capacitor value if we make the following assumptions:

- \( t_c \) is the charging time of the capacitor \( C \).
- \( t_d \) is the discharging time of the capacitor \( C \).
If we assume that the charging time $t_c$ is small compared to the discharging time $t_d$ (i.e., $t_d >> t_c$, which is generally the case), we can relate $t_c$ and $t_d$ to the period $T$ of the input supply as

$$t_d = \frac{T}{2} - t_c \approx \frac{T}{2} = \frac{1}{2} f$$ for a full-wave rectifier

$$= T - t_c \approx T = \frac{1}{f}$$ for a half-wave rectifier

(5.62a)

(5.62b)

Using Taylor series expansion of $e^{-x} = 1 - x$ for small values of $x << 1$, we can simplify Eq. (5.56) to

$$V_{o(min)} = V_m e^{-t_d/R_l C} = V_m \left( 1 - \frac{t_d}{R_l C} \right)$$

(5.63)

This gives the peak-to-peak ripple voltage $V_{r(pp)}$ as

$$V_{r(pp)} = V_m - V_{o(min)} = V_m \frac{t_d}{R_l C} = \frac{V_m}{2 f R_l C}$$ for a full-wave rectifier

$$= \frac{V_m}{f R_l C}$$ for a half-wave rectifier

(5.64a)

(5.64b)

Equations (5.64a and 5.64b) can be used to find the value of capacitor $C$ with reasonable accuracy for most practical purposes as long as the ripple factor is within 10%. We can observe from Eq. [5.64(a) and (b)] that the ripple voltage depends inversely on the supply frequency $f$, the filter capacitance $C$, and the load resistance $R_l$. For the same amount of voltage ripple, the full-wave rectifier will require half the capacitance $C$ due to having double the output ripple frequency $2f$ as compared to the half-wave rectifier.

If we assume that the output voltage decreases linearly from $V_{o(max)}(=V_m)$ to $V_{o(min)}$ during the discharging interval, the average output voltage can be found approximately from

$$V_{o(av)} = \frac{V_m + V_{o(min)}}{2} = \frac{1}{2} \left[ V_m + V_m \left( 1 - \frac{t_d}{R_l C} \right) \right]$$

(5.65)

After we substitute for $t_d$ in Eq. (5.65), this becomes

$$V_{o(av)} = \frac{1}{2} \left[ V_m + V_m \left( 1 - \frac{1}{2 R_l f C} \right) \right] = \frac{V_m}{2} \left[ 2 - \frac{1}{2 R_l f C} \right]$$ for a full-wave rectifier

$$= \frac{1}{2} \left[ V_m + V_m \left( 1 - \frac{1}{2 R_l f C} \right) \right] = \frac{V_m}{2} \left[ 2 - \frac{1}{R_l f C} \right]$$ for a half-wave rectifier

(5.66a)

(5.66b)

The ripple factor $RF$ can be found from

$$RF = \frac{V_{r(pp)}/2}{V_{o(av)}} = \frac{1}{4 R_l f C - 1}$$ for a full-wave rectifier

$$= \frac{1}{2 R_l f C - 1}$$ for a half-wave rectifier

(5.67a)

(5.67b)

The peak input voltage $V_m$ is generally fixed by the supply, whereas we can vary the minimum voltage $V_{o(min)}$ from almost zero to $V_m$ by varying the values of $C$, $f$, and $R_l$. Therefore, it is possible to design for an average output voltage $V_{o(av)}$ in the range from $V_m/2$ to $V_m$. We can find the value of capacitor $C$ to meet either a specific value of the minimum voltage $V_{o(min)}$ or the average output voltage $V_{o(av)}$ so that $V_{o(min)} = (2 V_{o(av)} - V_m)$. 
EXAMPLE 5.10

**Designing an output C filter** The single-phase full-wave bridge rectifier of Fig. 5.15(a) is supplied directly from a 120-V, 60-Hz source without any input transformer. The load resistance is $R_L = 500 \Omega$.

(a) Design a $C$ filter so that the peak-to-peak ripple voltage $V_{r(pp)}$ is within 10% of $V_m$.

(b) With the value of $C$ found in part (a) calculate the actual output voltage $V_{o(\text{av})}$, and the capacitor voltage if the load resistance $R_L$ is disconnected.

(c) Use PSpice/SPICE to check the design by plotting the instantaneous output voltage $v_o$. Use default diode parameters of 1N4148 type.

**SOLUTION**

(a) $V_m = \sqrt{2} V_s = \sqrt{2} \times 120 = 169.7$ V

The peak-to-peak ripple voltage is

$$V_{r(pp)} = 10\% \text{ of } V_m = 0.1 \times 169.7 = 16.97$$

The minimum output voltage is

$$V_{o(\text{min})} = V_m - V_{r(pp)} = 170 - 16.97 = 152.74$$

From Eq. (5.53), we get the angle $\alpha$ as

$$\alpha = \sin^{-1}\left(\frac{V_{o(\text{min})}}{V_m}\right) = \sin^{-1}\left(\frac{152.74}{1169.7}\right) = 1.12 \text{ rad}, \text{ or } 64.16^\circ$$

From Eq. (5.55a), we get the discharge $\beta$ as

$$\beta = \frac{\pi}{2} + \alpha = \frac{\pi}{2} + 1.12 = 2.698 \text{ rad}, \text{ or } 154.16^\circ$$

From Eq. (5.59a), we get the filter capacitor $C$:

$$C = \frac{\pi/2 + \alpha}{2\pi f R_L \ln \left(\frac{V_m}{V_{o(\text{min})}}\right)} = \frac{\pi/2 + 1.12}{2\pi \times 86 \times 500 \times \ln (169.7/152.74)} = 135.48 \mu\text{F}$$

(b) From Eq. (5.60), we get the average output voltage:

$$V_{o(\text{av})} = \frac{V_m}{\pi} \left[2\pi f R_L C \left(1 - e^{-\beta/\omega R_L C}\right) + \sin \beta \right] = 161.49$$

The approximate Eq. (5.64a) gives $C = 166.7 \mu\text{F}$, and Eq. (5.65) gives $V_{o(\text{av})} = \frac{(V_m + V_{o(\text{min})})}{2} = 161.2$ V. If the load resistance $R_L$ is disconnected, the capacitor will charge to the peak input voltage $V_m$. Therefore, the average output voltage with no load is

$$V_{o(\text{no-load})} = V_m = 169.7$$

The average output voltage $V_{o(\text{av})}$ will change from 169.7 V to 158.49 V if the load is connected. This change in voltage is normally specified by a factor known as the *voltage regulation*, which is defined as

$$\text{Voltage regulation} = \frac{V_{o(\text{no-load})} - V_{o(\text{load})}}{V_{o(\text{load})}} = \frac{V_{o(\text{load})} - V_{o(\text{av})}}{V_{o(\text{av})}}$$

$$= \frac{169.7 - 161.49}{161.49} = 5.08\%$$
5.3.3 LC Filters

An LC filter, which opposes any change in either the voltage or the current, reduces the harmonics more effectively than an L filter or a C filter. A rectifier with an LC filter is shown in Fig. 5.18(a). The equivalent circuit for harmonics is shown in Fig. 5.18(b) where \( V_{\text{in}} \) is the nth harmonic component of the rms ripple voltage.

To make it easier for the nth harmonic ripple current to pass through the filter capacitor \( C \) rather than through the load resistance \( R_L \), the load impedance \( Z_L (= R_L) \) must be greater than that of the capacitor. That is,

\[
R_L \gg \frac{1}{n \omega C}
\]
This condition is generally satisfied by choosing a ratio of 1:10. That is,

\[ R_L = \frac{10}{\omega C} \quad \text{(5.69)} \]

Under this condition, \( R_L \) can be neglected and the effect of the load resistance \( R_L \) will be negligible. Thus, Fig. 5.18(b) is reduced to Fig. 5.18(c). Using the voltage divider rule, we can find the rms value of the \( n \)th harmonic voltage component appearing after filtering on the output from

\[ V_{\text{in(rms)}} = \left| \frac{-j/(n\omega C)}{j\omega L - j/(n\omega C)} \right| V_{\text{on(rms)}} = \frac{1}{\left| 1 - (n\omega)^2 LC \right|} V_{\text{on(rms)}} \quad \text{(5.70)} \]

where \( V_{\text{on(rms)}} \) is the rms \( n \)th harmonic voltage of Eq. (5.19) or Eq. (5.37). If the higher-order harmonics are neglected and the second harmonic becomes the dominant one, \( V_{\text{o2(rms)}} \) becomes the output ripple voltage of the rectifier, and Eq. (5.70) can be written as

\[ V_{\text{r(rms)}} = V_{\text{o2(rms)}} = \frac{1}{\left| 1 - (n\omega)^2 LC \right|} V_{\text{o2(rms)}} \quad \text{(5.71)} \]

With the value of \( C \) from Eq. (5.69), the value of \( L \) can be computed for a specified value of \( V_{\text{r(rms)}} \).

**EXAMPLE 5.11**

**Designing an output LC filter**  The single-phase bridge rectifier of Fig. 5.18(a) is supplied directly from a 120-V, 60-Hz source without any input transformer. The load resistance is \( R_L = 500 \) Ω.

(a) Design an LC filter so that the rms ripple voltage \( V_{\text{r(rms)}} \) is within 5% of \( V_{\text{o(avg)}} \).

(b) Use PSpice/SPICE to check your design by plotting the instantaneous output voltage \( v_O \). Use default diode parameters.

**SOLUTION**

(a) \( f = 60 \) Hz, \( \omega = 2\pi f = 377 \) rad/s, \( R_L = 500 \) Ω, and RF = 5% = 0.05.

\[ V_m = \sqrt{2} V_s = \sqrt{2} \times 120 = 169.7 \text{ V} \]

\[ V_{\text{o(avg)}} = \frac{2V_m}{\pi} = \frac{2 \times 169.7}{\pi} = 108.03 \text{ V} \]

\[ V_{\text{r(rms)}} = 5\% \text{ of } V_{\text{o(avg)}} = 0.05 \times 108.03 = 5.4 \text{ V} \]
Assume that the ripple voltage is approximately sinusoidal. Then, the peak ripple voltage is given by
\[ \frac{V_{r(\text{peak})}}{H} \]
The peak-to-peak ripple voltage \( V_{r(pp)} \) is
\[ V_{r(pp)} = 2 \times V_{r(\text{peak})} = 2 \times 7.64 = 15.28 \text{ V} \]
Let us consider only the dominant harmonic—that is, the second harmonic. From the second term in Eq. (5.37), the rms value of the second harmonic is
\[ V_{o2(rms)} = \frac{4V_m}{3\sqrt{2\pi}} \]

**NOTE:** \( \sqrt{2} \) converts the peak value to a rms value.

For \( n = 2 \), the value of \( C \) can be found from Eq. (5.69) as follows:
\[ C = \frac{10}{n\omega R_L} = \frac{10}{2 \times 377 \times 500} = 26.53 \text{ } \mu \text{F} \]

Using Eqs. (5.71) and (5.25), we can find the ripple factor RF of the output voltage from
\[ RF = \frac{V_{o(\text{rms})}}{V_{o(\text{av})}} = \frac{V_{o2(rms)}}{|1 - (n\omega)^2LC|} \times \frac{\pi}{2V_m} = \frac{1}{|1 - (n\omega)^2LC|} \times \frac{4V_m}{3\sqrt{2\pi}} \times \frac{\pi}{2V_m} \]
\[ RF = \frac{\sqrt{2}/3}{|1 - (n\omega)^2LC|} \]
which can be solved for \( L \):
\[ L = \frac{1}{(n\omega)^2C} \left[ \frac{\sqrt{2}}{3RF} - 1 \right] = \frac{1}{(2 \times 377)^2 \times 26.53 \times 10^{-6} \left[ \frac{\sqrt{2}}{3 \times 0.05} - 1 \right]} = 0.56 \text{ H} \]

(b) The single-phase bridge rectifier circuit with an \( LC \) filter for PSpice simulation is shown in Fig. 5.19. The PSpice plot of \( v_O \), shown in Fig. 5.20, gives the peak-to-peak ripple voltage as \( V_{r(pp)} = 14.99 \text{ V} \), compared to the calculated value of 15.28 V. There is an error of 1.39 V, which can arise from various factors such as neglecting the higher-order harmonics, not considering the loading effect of \( R_L \), and assuming an ideal diode with zero voltage drop. Thus, the design values should be revised until the desired specifications are satisfied.
5.4 Diode Peak Detectors and Demodulators

The half-wave rectifier shown in Fig. 5.21(a) can be employed as a peak signal detector. Let us consider a sinusoidal input voltage, \( v_S = V_m \sin \omega t \). During the first quarter-cycle, the input voltage will rise, the capacitor \( C \) will be charged almost instantaneously to the input voltage, and the capacitor (or output) voltage \( v_O \) will follow the input voltage \( v_S \) until the instantaneous \( v_S \) reaches \( V_m \) at time \( t = \pi / 2 \omega \). When the input voltage \( v_S \) tries to decrease, diode \( D_1 \) will be reverse biased and the capacitor \( C \) will discharge through resistance \( R \). If we define the time \( t = t_1 \) when \( C \) is charged to \( V_m \), the output (or capacitor) voltage \( v_O \), which falls exponentially, takes the form

\[
v_O(t) = V_m e^{-(t-t_1)/RC} \quad \text{for} \quad t_1 \leq t \leq (t_1 + t_2)
\]  

(5.72)

The waveform of the output voltage is shown in Fig. 5.21(b). If the time constant \( \tau = RC \) is too small, the capacitor will discharge its voltage very quickly and will not maintain its voltage close to \( V_m \). The output voltage will be discontinuous and will not be a true representation of the peak input signal. On the other hand, if the time constant \( \tau \) is too large, the output voltage will not change rapidly with a change in the peak value \( V_m \) of the input voltage. If the time constant \( \tau \) is properly selected, the output voltage should approximately represent the peak input signal, within a reasonable error.
A peak detector can be used as a demodulator to detect the audio signal in an amplitude modulated (AM) radio signal. Amplitude modulation is a method of translating a low-frequency signal into a high-frequency one. The AM waveform can be described by

$$v_s(t) = V_m[1 + M \sin(2\pi f_m t)] \sin(2\pi f_c t)$$  \hspace{1cm} (5.73)

where  
\begin{align*}
f_c & \text{ = carrier frequency, in Hz} \\
f_m & \text{ = modulating frequency, in Hz} \\
M & \text{ = modulation index, whose value varies between 0 and 1} \\
V_m & \text{ = peak modulating voltage}
\end{align*}

The term $V_m[1 + M \sin(2\pi f_m t)]$ represents the envelope of the modulated waveform. Its slope (or rate of change) $S$ is given by

$$S = \frac{d}{dt}[V_m + M V_m \sin(2\pi f_m t)] = M 2\pi f_m V_m \cos(2\pi f_m t)$$  \hspace{1cm} (5.74)

The waveform of a modulated signal is shown in Fig. 5.22(a). Since the demodulator gives the peak value, the corresponding output of the peak detector is shown in Fig. 5.22(b). A low-pass filter can be used to smooth the demodulated signals. With a proper choice of time constant $\tau = RC$, the output will trace each peak of the modulating signal. If the time constant is too large, the output will not be able to change fast enough and the audio signal will be distorted. If the time constant is too small, there will be too much “ripple” superimposed on the modulating signal.

![Amplitude modulated waveform](image.png)

**FIGURE 5.22** Amplitude modulated waveform
The slope $S$ in Eq. (5.74) will be maximum at $\theta = 2\pi f_m t = 0$ or $\pi$. Therefore, the peak slope (or rate of change) $S_m$ is given by

$$S_m = \pm M 2\pi f_m V_m$$  \hspace{1cm} (5.75)

From Eq. (5.72), we can find the peak slope $S_D$ of the detector as

$$S_D = \frac{dv_O}{dt} \bigg|_{t=t_1} = -V_m \frac{1}{RC} e^{-(t-t_1)/RC} = -\frac{V_m}{RC}$$ \hspace{1cm} (5.76)

For the detector to cope with a rapid change in the peak input voltage, the magnitude of the slope $S_D$ of the detector must be greater than that of the modulating signal. That is,

$$|S_D| \geq |S_m|$$

Substituting $S_m = -2\pi M f_m V_m$ from Eq. (5.75) under the falling slope condition and $S_D$ from Eq. (5.76), we get

$$\left| -\frac{V_m}{RC} \right| \geq \left| -2\pi M f_m V_m \right|$$  \hspace{1cm} (5.77)

which gives the desired value of capacitance $C$ as

$$C \geq \frac{1}{2\pi f_m MR}$$  \hspace{1cm} (5.78)

The peak slope $S_m$ of the modulating signal in Eq. (5.75) will have a maximum value if $M = 1$. Therefore, the value of capacitance $C$ should be determined for $M = 1$. Thus, Eq. (5.78) gives the limiting value of $C$ as

$$C \geq \frac{1}{2\pi f_m R}$$  \hspace{1cm} (5.79)

The design value of $C$ should be higher than the limiting value in order to follow the peaks.

**EXAMPLE 5.12**

**Designing a demodulator circuit**  The carrier frequency $f_c$ of a radio signal is 100 kHz, and the modulating frequency $f_m$ is 10 kHz. The load resistance $R$ of the detector is 5 k$\Omega$.

(a) Design a demodulator for the waveform of Fig. 5.22(a) by determining the value of capacitance $C$.

(b) Use PSpice/SPICE to plot the output voltage $v_O$ for a modulation index of $M = 0.5$ and 1.0. The peak modulating voltage is $V_m = 20$ V. Use diode parameters of diode D1N4148:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$I_S$</td>
<td>2.682 nA</td>
</tr>
<tr>
<td>$C_J0$</td>
<td>4 pF</td>
</tr>
<tr>
<td>$M$</td>
<td>0.3333</td>
</tr>
<tr>
<td>$V_J$</td>
<td>0.5 V</td>
</tr>
<tr>
<td>$B_V$</td>
<td>100 V</td>
</tr>
<tr>
<td>$I_BV$</td>
<td>100 uA</td>
</tr>
<tr>
<td>$T_T$</td>
<td>11.54 N</td>
</tr>
</tbody>
</table>

**SOLUTION**

(a) $f_c = 100$ kHz, $f_m = 10$ kHz, and $R = 5$ k$\Omega$.

From Eq. (5.79),

$$C = \frac{1}{2\pi f_m R} = \frac{1}{2\pi \times 10 \times 10^3 \times 5 \times 10^3} = 1605 \text{ nF}$$
(b) PSpice allows sine functions only, so we need to convert the cosine term into a sine term. Using the trigonometric relationship

\[
\sin A \sin B = \frac{1}{2} \left[ \cos (A - B) - \cos (A + B) \right]
\]

we can expand Eq. (5.73) to

\[
v_s(t) = V_m \sin (2\pi f_c t) + M V_m \sin (2\pi f_m t) \sin (2\pi f_c t)
\]

\[
= V_m \sin (2\pi f_c t) + \frac{M V_m}{2} \cos (2\pi (f_c - f_m) t) - \frac{M V_m}{2} \cos [2\pi (f_c + f_m) t]
\]

\[
= V_m \sin (2\pi f_c t) + \frac{M V_m}{2} \sin [2\pi (f_c - f_m) t + 90^\circ] - \frac{M V_m}{2} \sin [2\pi (f_c + f_m) t + 90^\circ]
\]  
\[(5.80)\]  

For \( M = 0.5, \frac{M V_m}{2} = 0.5 \times 20 = 5 \) V

\( f_1 = f_c - f_m = 100 \) kHz - 10 kHz = 90 kHz

\( f_2 = f_c + f_m = 100 \) kHz + 10 kHz = 110 kHz

The demodulator circuit for PSpice simulation is shown in Fig. 5.23. The PSpice plot of \( v_o \), shown in Fig. 5.24, gives the peak value of output voltage as \( V_{o(peak)} = 29.1 \) V, compared to the calculated value of \((1 + m)V_m = (1 + 0.5) \times 20 = 30 \) V.

\[\text{FIGURE 5.23} \quad \text{Demodulator circuit for PSpice simulation} \]

\[\text{FIGURE 5.24} \quad \text{PSpice plot for Example 5.12} \]

**KEY POINTS OF SECTION 5.4**

- A diode can charge a capacitor to the peak value of the input voltage and thus can be used as a peak detector.
- A peak detector can be used as a demodulator to detect the audio signal in an amplitude modulated (AM) radio signal.
5.5 Diode Clippers

A clipper is a limiting circuit; it is basically an extension of the half-wave rectifier. The output of a clipper circuit looks as if a portion of the output signal was cut off (clipped). Although the input voltage can have any waveform, we will assume that the input voltage is sinusoidal, \( v_S = V_m \sin \omega t \), in order to describe the output voltage. Clippers can be classified into two types: parallel clippers and series clippers. The diode can be connected either in series or in parallel with the load.

5.5.1 Parallel Clippers

A clipper in which the diode is connected across the output terminals is known as a parallel clipper because the diode will be in parallel (or shunt) with the load. In a shunt connection, elements are connected in parallel such that each element carries a different current. Some examples of parallel clipper circuits and their corresponding output waveforms are shown in Fig. 5.25. The resistance \( R \) limits the diode current when the diode conducts. In determining the output waveform of a clipper, it is important to keep in mind that a diode will conduct only if the anode voltage is higher than the cathode voltage.

FIGURE 5.25 Diode parallel clipper circuits
When diode $D_1$ in Fig. 5.25(a) is off, the instantaneous output voltage $v_O$ equals the instantaneous input voltage $v_S$. Diode $D_1$ will conduct for the portion of the positive half-cycle during which the instantaneous input voltage $v_S$ is higher than the battery voltage $E_1$. On the other hand, diode $D_2$ in Fig. 5.25(b) will conduct when the input voltage is less than the battery voltage $E_1$. Although the output waveforms of these two circuits are identical, as shown in Fig. 5.25(c), diode $D_2$ in Fig. 5.25(b) remains on for a longer time than diode $D_1$ in Fig. 5.25(a). For this reason, the clipper of Fig. 5.25(a) is preferable to that of Fig. 5.25(b).

Diode $D_1$ in Fig. 5.25(d) will conduct most of the time and be off for the portion of the positive half-cycle during which the instantaneous input voltage $v_S$ is higher than the battery voltage $E_1$ whereas diode $D_2$ will remain on for a short time. The output waveforms for the clippers of Fig. 5.25[(d) and (e)] are identical, as shown in Fig. 5.25(f).

The circuits of Fig. 5.25[(a) and (d)] (with $E_1$ reversed and renamed as $E_2$) can be combined to form a two-level clipper, as shown in Fig. 5.25(g). The positive and negative voltages are limited to $E_1$ and $E_2$, respectively, as shown in Fig. 5.25(h). One battery terminal of the clippers in Fig. 5.25 is common to the ground.

### 5.5.2 Series Clippers

A clipper in which the diode forms a series circuit with the output terminals is known as a *series clipper*. The current-limiting resistance $R$ can be used as a load, as shown in Fig. 5.26(a). If the direction of the battery is reversed, the negative part of the sine wave is clipped as shown in Fig. 5.26(b). If the direction of
the diode is reversed, the clipping becomes the opposite of that in Fig. 5.26(a); this situation is shown in Fig. 5.26(c). The potential difference between terminals A and B of the battery must be $E_1$. But terminal B cannot be at zero or ground potential. Therefore, these circuits require an isolated DC voltage (or battery) of $E_1$. Note that the zero level of the output voltage $v_O$ is different from that of the input voltage $v_S$ and is shifted by an amount equal to $E_1$.

**EXAMPLE 5.13**

**Designing a clipper circuit** The clipper circuit shown in Fig. 5.27(a) is supplied from the input voltage shown in Fig. 5.27(b). The battery voltage is $E_1 = 10$ V. The peak diode current $I_{D(peak)}$ is to be limited to 30 mA. Determine (a) the value of resistance $R$, (b) the average diode current $I_{D(avg)}$ and the rms diode current $I_{D(rms)}$, and (c) the power rating $P_R$ of the resistance $R$.

**SOLUTION**

$I_{D(peak)} = 30$ mA, and $E_1 = 10$ V. Imagine a line at $E_1 = 10$ V on the plot of $v_S$ in Fig. 5.27(b).

(a) During the period $0 \leq t \leq t_1$, the input voltage $v_S$ is 20 V. Diode D1 is reverse biased, and the output voltage $v_O$ becomes the same as the input voltage $v_S$. That is, $v_O = v_S = 20$ V. During the period $t_1 \leq t \leq (t_1 + t_2)$, diode D1 is forward biased and it will conduct. The output voltage $v_O$ is clamped to $E_1 = 10$ V. The equivalent conducting circuit is shown in Fig. 5.28(a); the waveform for the output voltage is shown in Fig. 5.28(b). The peak diode current $I_{D(peak)}$ is given by

$$I_{D(peak)} = \frac{V_m + E_1}{R} = \frac{20 + 10}{R}$$

For $I_{D(peak)} = 30$ mA, $R = (20 + 10) \text{ V}/30 \text{ mA} = 1 \text{ k}\Omega$.

(b) The average diode current $I_{D(avg)}$ can be found from

$$I_{D(avg)} = \frac{1}{t_1 + t_2} \int_{t_1}^{t_1+t_2} I_{D(peak)} dt = \frac{I_{D(peak)} t_2}{t_1 + t_2} = \frac{30 \text{ mA} \times 6 \text{ ms}}{4 \text{ ms} + 6 \text{ ms}} = 18 \text{ mA}$$

**FIGURE 5.27** Clipper circuit
5.6 Diode Clamping Circuits

A clamping circuit simply shifts the output waveform to a different DC level. Thus, it is often known as a level shifter. The shapes of the input and output waveforms are identical; only the DC level is shifted. The input voltage can have any shape. However, we will assume that the input voltage is sinusoidal, \( v_S = V_m \sin \omega t \).

Clampers can be classified into two types: fixed-shift clampers and variable-shift clampers.

---

**FIGURE 5.28** Equivalent circuit and waveforms for Example 5.13

(a) Equivalent conducting circuit

(b) Output voltage

---

Then

\[
I_{D(rms)} = 30 \text{ mA} \sqrt{\frac{6 \text{ mA}}{10 \text{ mA}}} = 23.24 \text{ mA}
\]

(c) Then

\[
P_R = I_{D(rms)}^2 R = (23.24 \times 10^{-3} \text{ A})^2 \times 1 \text{ k} \Omega = 0.54 \text{ W}
\]

---

**KEY POINTS OF SECTION 5.5**

- A diode clipper can cut off a portion of its output voltage.
- If the diode forms a series circuit with the load, it is called a series clipper. If the diode forms a parallel circuit with the load, it is called a parallel clipper.
- The output voltage of a clipper can be determined as follows:
  - **Step 1.** Draw a clockwise loop to determine the polarity of the battery. If the positive terminal of the battery is encountered first, then \( E_1 \) is positive. If the negative terminal is encountered first, then \( E_1 \) is negative.
  - **Step 2.** Draw a line at \( \pm E_1 \) on the plot of the input voltage.
  - **Step 3.** Find out when the diode will conduct. Then clip the appropriate portion of the input voltage, depending on the state of the diode (on or off), in order to obtain the output voltage \( v_O \).
  - **Step 4.** Draw the final output voltage.

---

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
5.6.1 Fixed-Shift Clampers

As shown in Fig. 5.29, a fixed-shift clamper shifts the output voltage by an amount ±V_m with respect to the zero level. Let us consider the clamping circuit in Fig. 5.29(a). As soon as the input voltage v_S is switched on, diode D_1 will conduct during the first positive quarter-cycle of the input voltage, and the capacitor C will be charged almost instantaneously to the peak input voltage V_m. But the output voltage will be zero, v_O = 0. The circuit will reach a steady-state condition with a voltage of V_m across the capacitor C, as depicted in Fig. 5.29(a). Therefore, after the first quarter-cycle, the capacitor voltage will be v_C = V_m, and the output voltage v_O will become

\[ v_O = v_S - v_C = v_S - V_m = V_m \sin \omega t - V_m = V_m (\sin \omega t - 1) \text{ for } \omega t \geq \frac{\pi}{2} \]

as shown in Fig. 5.29(b).

Let us assume that the input voltage v_S falls below the initial peak voltage of V_m (say, 20 V) to a new peak value of V_m1 (say, 10 V). This situation is shown in Fig. 5.29(c). The diode voltage is now v_O = v_S - v_C = 10 \sin \omega t - 20, which is negative for all \omega t, and the diode becomes reverse biased.

![Fixed-shift clamping circuit](image)
The capacitor voltage cannot adjust to the new value \( V_{m1} \) because diode \( D_1 \) is now reverse biased and there is no discharge path for the capacitor. The output voltage will be \( v_O = V_m - V_{m1} \sin \omega t \), instead of \( v_O = V_{m1} \sin(\omega t - 1) \) as expected. To allow the capacitor voltage to adjust to the change in the peak input voltage, a resistance \( R \) is connected across diode \( D_1 \), as shown in Fig. 5.29(d). If the input voltage then falls to a new peak, the capacitor \( C \) can discharge slowly through the resistance \( R \). Similarly, if the input voltage is increased to a new peak, the capacitor \( C \) can charge through the resistance \( R \). However, the voltage across the capacitor must remain fairly constant during the whole period. The values of \( R \) and \( C \) must be chosen such that the time constant \( \tau = RC \) is large enough to ensure that the capacitor voltage does not change significantly within one period \( T \) of the input voltage. This condition is generally satisfied by making the time constant \( \tau \) equal to 10 times the period \( T \). That is, \( \tau = 10T \).

If the direction of diode \( D_1 \) is reversed, as shown in Fig. 5.29(e), the diode will be reverse biased during the first positive half-cycle of the input voltage, and the output voltage will be equal to the input voltage, \( v_O = V_S \). Diode \( D_1 \) will conduct during the first negative half-cycle of the input voltage. The capacitor \( C \) will be charged almost instantaneously to the negative peak input voltage \(-V_m\), and the output voltage becomes zero, \( v_O = 0 \). This process is completed during the first cycle, and the circuit reaches a steady-state condition with an input voltage of \(-V_m\) across the capacitor \( C \). After the first cycle, the capacitor voltage remains constant at \( v_C = -V_m \). The output voltage \( v_O \) under steady-state conditions becomes

\[
v_O = v_S - v_C = v_S - (-V_m) = V_m \sin \omega t + V_m = V_m (\sin \omega t + 1) \quad \text{for} \quad \omega t \geq \frac{3\pi}{2}
\]

as shown in Fig. 5.29(f). Therefore, switching the direction of the diode makes the output inverted with a phase shift of \( \pi \).

**NOTE** If we ignore the initial transient interval, which is required to charge the capacitor for normal operation, the output waveform of the clamping circuit in Fig. 5.29(f) becomes positive with respect to that in Fig. 5.29(b). That is, one shifts the input signal in the positive direction and the other shifts it in the negative direction.

### 5.6.2 Variable-Shift Clampsers

The output voltage \( v_O \) can be shifted to a predefined value by introducing a battery voltage \( E_1 \). This type of clamp shown in Fig. 5.30 shifts the output voltage by an amount \( \pm V_m \pm E_1 \) with respect to the zero level. Consider the clamping circuit in Fig. 5.30(a). The capacitor \( C \) will be charged to \( v_C = V_m - E_1 \) during the first positive quarter-cycle of the input voltage, and the instantaneous output voltage \( v_O \) under steady-state conditions becomes

\[
v_O = v_S - v_C = V_m \sin \omega t - (V_m - E_1) = V_m \sin \omega t - V_m + E_1 \quad \text{for} \quad \omega t \geq \frac{\pi}{2}
\]

The capacitor \( C \) in Fig. 5.30(b) will be charged to \( v_C = -(V_m + E_1) \) during the first negative quarter-cycle of the input voltage. There will be an instantaneous charging to \(-E_1\) at \( t = 0 \). Thus, the instantaneous output voltage \( v_O \) under steady-state conditions becomes

\[
v_O = v_S - v_C = V_m \sin \omega t + (V_m + E_1) = V_m \sin \omega t + V_m + E_1 \quad \text{for} \quad \omega t \geq \frac{3\pi}{2}
\]
The capacitor C in Fig. 5.30(c) will be charged to \( v_C = (V_m + E_1) \) during the first positive quarter-cycle of the input voltage. The instantaneous output voltage \( v_O \) under steady-state conditions becomes

\[
v_O = v_S - v_C = V_m \sin \omega t - (V_m + E_1) = V_m \sin \omega t - V_m - E_1 \quad \text{for} \quad \omega t \geq \frac{\pi}{2}
\]

The capacitor C in Fig. 5.30(d) is charged to \( v_C = -(V_m - E_1) \) during the first negative quarter-cycle of the input voltage. The instantaneous output voltage \( v_O \) under steady-state conditions becomes

\[
v_O = v_S - v_C = V_m \sin \omega t + (V_m - E_1) = V_m \sin \omega t + V_m - E_1 \quad \text{for} \quad \omega t \geq \frac{3\pi}{2}
\]
EXAMPLE 5.14

**Designing a clamping circuit** The input voltage \( v_S \) to the clamping circuit of Fig. 5.31(a) is a rectangular wave, as shown in Fig. 5.31(b). The peak diode current \( I_{D(\text{peak})} \) is to be limited to 0.5 A.

(a) Design the clamping circuit by determining the peak inverse voltage (PIV) of the diode and the values of \( R_S, R, \) and \( C \).

(b) Use PSpice/SPICE to plot the output voltage \( v_O \). Use diode parameters of diode D1N4148:

\[
\begin{align*}
I_S &= 2.682 N \\
C_{JO} &= 4 P \\
M &= 0.3333 \\
V_J &= 0.5 \\
B_V &= 100 \\
I_{BV} &= 100 U \\
TT &= 11.54 N
\end{align*}
\]

**SOLUTION**

(a) \( I_{D(\text{peak})} = 0.5 \) A. The period \( T \) of the input waveform is

\[
T = t_1 + t_2 = 6 \text{ ms} + 8 \text{ ms} = 14 \text{ ms}
\]

\[
\text{PIV} = -v_S + v_C - E_1 = 10 + 25 - 5 = 30 \text{ V}
\]

for \( 6 \text{ ms} \leq t \leq 14 \text{ ms} \)

For \( 0 \leq t \leq 6 \text{ ms} \), \( v_O = -E_1 = -5 \text{ V} \) and for \( 6 \text{ ms} \leq t \leq 14 \text{ ms} \), \( v_O = -25 - 10 = -35 \text{ V} \). The waveform of the output voltage \( v_o \) is shown in Fig. 5.31(c).

The peak diode current \( I_{D(\text{peak})} \) is given by

\[
I_{D(\text{peak})} = \frac{20 + E_1}{R_S}
\]

or

\[
R_S = \frac{20 + E_1}{I_{D(\text{peak})}} = \frac{20 + 5}{0.5} = 50 \Omega
\]

Let \( \tau = (R + R_S)C = 10T = 10 \times 14 \text{ ms} = 140 \text{ ms} \). Choose a suitable value of \( C \). Let \( C = 0.1 \mu F \). Then

\[
R + R_S = \frac{\tau}{C} = \frac{140 \times 10^{-3}}{0.1 \times 10^{-6}} = 1.4 \text{ M}\Omega
\]

which gives \( R = 1.4 \text{ M}\Omega - R_S = 1.4 \text{ M}\Omega - 50 \Omega = 1.4 \text{ M}\Omega \).

(b) The clamping circuit for PSpice simulation is shown in Fig. 5.32. The PSpice plot of \( v_O \), shown in Fig. 5.33, gives the peak-to-peak output voltage as \( V_o(\text{pp}) = 29.99 \text{ V} \), compared to the calculated value of 30 V.
5.7 Diode Voltage Multipliers

A diode clamping circuit followed by a peak voltage detector can be used as a building block for stepping up the peak input voltage $V_m$ by a factor of 2, 3, 4, or more.

5.7.1 Voltage Doublers

A half-wave voltage doubler circuit, shown in Fig. 5.34(a), uses a clamping circuit and a peak detector. Let us consider a sinusoidal input voltage of $v_S = V_m \sin \omega t$. The circuit operation can be divided into four intervals: interval 1, interval 2, interval 3, and interval 4.
FIGURE 5.34  Half-wave voltage doubler circuit

Interval 1 is the interval $0 \leq \omega t \leq \pi/2$. As soon as the input voltage is switched on, diode D1 will conduct, but diode D2 will be reverse biased. The output voltage is $v_O = 0$. The capacitor C1 will be charged during the first quarter-cycle to $V_m$ (at $\omega t = \pi/2$) with the polarities shown.

Interval 2 is the interval $\pi/2 \leq \omega t \leq \pi$. D1 will be off, and D2 will be on. If the value of $R$ is large enough that $RC > 1/f$, where $f$ = supply frequency, then capacitor C1 will not have time to discharge through $R$ and the voltage on capacitor C1 will remain approximately at $V_m$.

Interval 3 is the interval $\pi \leq \omega t \leq 3\pi/2$. The polarity of the input voltage is negative. Diode D1 will be off, and diode D2 will conduct. The output voltage $v_O$, which will be the same as the voltage across capacitor C2, will become $v_O = v_{C1} - v_S = V_m - V_m \sin \omega t$. At $\omega t = 3\pi/2$, the output voltage will become $2V_m$ and the capacitor C2 will be charged to $2V_m$.

Interval 4 is the interval $3\pi/2 \leq \omega t \leq 2\pi$. Diodes D1 and D2 will be off. The voltage on capacitor C1 will be $v_{C1} = V_m$, and that on capacitor C2 will be $v_{C2} = 2V_m$. However, we have assumed that capacitor C1 acts as the voltage source of $V_m$ and contributes to charging C2. In fact, C1 and C2 form a series circuit and share $2V_m$, so the voltage on capacitor C2 will be less than $2V_m$. It will take a couple of cycles before the steady-state condition is reached.

The waveforms for instantaneous input and output voltages are shown in Fig. 5.34(b). If the directions of the diodes are reversed, as shown in Fig. 5.34(c), the polarities of the output voltage will also be reversed. If a load resistance $R_L$ is connected across capacitor C2, the output voltage will fall when D1 is off and will rise when D2 is on. More time will be required to reach the steady-state condition.

Figure 5.35 shows a full-wave voltage doubler circuit. During the first quarter-cycle, $v_S$ is positive, diode D1 will conduct, and diode D2 will be reverse biased, thereby causing the capacitor C1 to be charged to $v_{C1} = V_m$ with polarities as shown. During the third quarter-cycle, $v_S$ is negative, diode D1 is reverse biased, and diode D2 will conduct. Thus capacitor C2 will be charged to $v_{C2} = V_m$ with polarities as shown. The steady-state output voltage after a complete cycle will be $v_O = 2V_m$. If a load resistance $R_L$ is connected across the output, the effective capacitance seen by the load is $C = (C_1 || C_2)$, which will be less than $C_2$ for the half-wave doubler circuit of Fig. 5.34(a). A lower value of effective capacitance indicates poorer filtering than that provided by a single capacitor filter. The peak inverse voltage PIV of the diodes in Figs. 5.34 and 5.35 will be $2V_m$. 
5.7.2 Voltage Triplers and Quadruplers

Two half-wave voltage doublers can be cascaded to develop three or four times the peak input voltage $V_m$, as shown in Fig. 5.36(a). Note that resistances, which are not shown across diodes $D_1$, $D_2$, and $D_3$, should be connected so that the circuit can cope with a changing peak in the input voltage. During the first quarter-cycle ($0 \leq \omega t \leq \pi/2$) of input voltage $v_s$, capacitor $C_1$ will be charged to $V_m$ through $D_1$. During the third quarter-cycle ($\pi \leq \omega t \leq 3\pi/2$), capacitor $C_2$ will be charged to $2V_m$ through $C_1$ and $D_2$. During the fifth quarter-cycle ($2\pi \leq \omega t \leq 5\pi/2$), capacitor $C_3$ will be charged to $2V_m$ through $C_1$, $C_2$, and $D_3$. During the seventh quarter-cycle ($3\pi \leq \omega t \leq 7\pi/2$), capacitor $C_4$ will be charged to $2V_m$ through $C_1$, $C_2$, $C_3$, and $D_4$. Depending on the output connections, the steady-state output voltage can be $V_m$, $2V_m$, $3V_m$, or $4V_m$. The instantaneous output voltages across various terminals are shown in Fig. 5.36(b) (e.g., $v_{O1} = v_{C1}$, $v_{O2} = v_{C2}$, $v_{O3} = v_{C3}$, and $v_{O4} = v_{C4}$). If additional sections of diode and capacitor are used, each capacitor will be charged to $2V_m$. The peak inverse voltage $PIV$ of each diode is $2V_m$. It will take a couple of cycles before the steady-state conditions are reached.
5.8 Diode Function Generators

Diodes can be employed to generate and synthesize driving-point functions, which refer to the \( v-i \) relations of two-port circuits. Some diode circuits for generating functions are shown in Fig. 5.39 [2, 5]. In deriving the transfer functions, it is important to keep in mind that a diode will conduct only when it is forward biased; it is off under reverse-biased conditions. The following guidelines will be helpful in analyzing the characteristics of diode function generators:

**Step 1.** To determine whether the diode is forward biased or reverse biased, assume that the diode is reverse biased and determine the anode-to-cathode voltage \( V_{AK} \) of the open diode.

**Example 5.15**

**Voltage quadrupler circuit** Use PSpice/SPICE to plot the output voltages \( v_{O2} \) and \( v_{O4} (=v_{C4}) \) for the voltage quadrupler in Fig. 5.36(a). Assume \( v_S = 20 \sin 2000\pi t \) and \( C_1 = C_2 = C_3 = C_4 = 0.1 \) \( \mu F \). Assume parameters of diode D1N4148:

\[
\begin{align*}
I_S & = 2.682 \text{nA} \quad C_{JO} = 4 \text{pF} \quad M = 0.3333 \quad V_J = 0.5 \quad BV = 100 \quad IBV = 100 \text{mV} \quad TT = 11.54 \text{N}
\end{align*}
\]

**Solution**

The voltage quadrupler circuit for PSpice simulation is shown in Fig. 5.37. The PSpice plots of \( v_{O4} \), shown in Fig. 5.38, give the peak output voltage as \( V_{o4(peak)} = 76.59 \) V, compared to the calculated value of \( 4V_m = 4 \times 20 = 80 \) V. It takes a couple of cycles before steady-state conditions are reached.

**Key Points of Section 5.7**

- A diode clamping circuit followed by a peak voltage detector can be used to multiply the peak input voltage \( V_m \) by a factor of 2, 3, or more.
- Each peak detector adds \( 2V_m \).
FIGURE 5.39  Diode circuits for function generation
Step 2. If $V_{AK}$ is negative, the assumption that the diode is reverse biased is correct; proceed with the analysis.

Step 3. If $V_{AK}$ is positive, the assumption that the diode is reverse biased is wrong. Replace the diode with a short circuit and reanalyze.

**EXAMPLE 5.16**

**Finding the transfer function of a diode circuit** A diode circuit is shown in Fig. 5.40(a). The circuit parameters are $R_1 = 5 \, \text{k}\Omega$, $R_2 = 1.25 \, \text{k}\Omega$, $R_3 = 1 \, \text{k}\Omega$, $V_1 = 5 \, \text{V}$, and $V_2 = 8 \, \text{V}$.

(a) Plot the $v-i$ relationship of the circuit.

(b) Use PSpice/SPICE to plot the transfer characteristic for $v_S = 0$ to $10 \, \text{V}$. Assume parameters of diode D1N4148:

- $I_S = 2.682 \, \text{N A}$
- $C_J = 4 \, \text{pF}$
- $M = 0.3333$
- $V_J = 0.5 \, \text{V}$
- $B_V = 100 \, \text{V}$
- $I_{BV} = 100 \, \text{uA}$
- $T_T = 11.54 \, \text{N}$

**SOLUTION**

(a) If $v_S < V_1 = 5 \, \text{V}$, diodes D1 and D2 will be reverse biased. The input current $i_S$ is described by

$$i_S = \frac{v_S}{R_1} = \frac{v_S}{5} \, \text{mA}$$

If $5 < v_S < 8 \, \text{V}$, diode D1 conducts and diode D2 is reverse biased. The input current $i_S$ can be found from

$$i_S = i_1 + i_2 = \frac{v_S}{R_1} + \frac{v_S - V_1}{R_2} = v_S \left( \frac{1}{R_1} + \frac{1}{R_2} \right) - \frac{V_1}{R_2} = (v_S - 4) \, \text{mA}$$

If $v_S > 8$, both diodes D1 and D2 will conduct. The input current can be found from

$$i_S = \frac{v_S}{R_1} + \frac{v_S - V_1}{R_2} + \frac{v_S - V_2}{R_3} = v_S \left( \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} \right) - \frac{V_1}{R_2} - \frac{V_2}{R_3}$$

$$= 2v_S - 4 - 8 = (2v_S - 12) \, \text{mA}$$

The $v-i$ plot of the relationship is shown in Fig. 5.40(b).

**FIGURE 5.40** Diode circuit for function generation
Diodes are used in many electronic circuits, including those of rectifiers, battery chargers, clippers, clammers, peak demodulators, voltage multipliers, function generators, logic gates, and voltage regulators. The analysis of diode circuits can be simplified by assuming an ideal diode model in which the resistance in the forward-biased condition is zero and the resistance in the reverse direction is very large, tending to infinity.

(b) The function generator for PSpice simulation is shown in Fig. 5.41. The PSpice plot of $i_S$ against $v_S$ is shown in Fig. 5.42. The break voltages (8.23 V and 5.44 V) at which the diodes are switched into the circuits are higher than the estimated values because the diode drops were neglected in hand calculations, whereas PSpice uses real diodes.

**Summary**

Diodes are used in many electronic circuits, including those of rectifiers, battery chargers, clippers, clammers, peak demodulators, voltage multipliers, function generators, logic gates, and voltage regulators. The analysis of diode circuits can be simplified by assuming an ideal diode model in which the resistance in the forward-biased condition is zero and the resistance in the reverse direction is very large, tending to infinity.
References


Review Questions

1. What is a rectifier?
2. What is an AC–DC converter?
3. What is the efficiency of rectification?
4. What are the differences between half-wave and full-wave rectifiers?
5. What is the lowest frequency of harmonics in a half-wave rectifier?
6. What is the lowest frequency of harmonics in a full-wave rectifier?
7. What are the advantages of full-wave rectifiers?
8. What are the purposes of filters in rectifiers?
9. What is a DC filter?
10. What is an AC filter?
11. What is a clamper?
12. What is a clipper circuit?
13. What is a demodulator?
14. What is a voltage multiplier?
15. How is voltage multiplication accomplished?
16. What is the transfer characteristic of a diode circuit?

Problems

The symbol indicates that a problem is a design problem.

5.2 Diode Rectifiers

5.1 The single-phase half-wave rectifier of Fig. 5.2(a) is supplied directly from a 120-V (rms), 60-Hz source through a step-down transformer with turns ratio \( n = 10:1 \). The load resistance \( R_L \) is 10 \( \Omega \). Determine (a) the average output voltage \( V_{o(\text{av})} \), (b) the average load current \( I_{o(\text{av})} \), (c) the rms load voltage \( V_{o(\text{rms})} \), (d) the rms load current \( I_{o(\text{rms})} \), (e) the ripple factor \( RF \) of the output voltage, (f) the rms ripple voltage \( V_{r(\text{rms})} \), (g) the average diode current \( I_{D(\text{av})} \), (h) the rms diode current \( I_{D(\text{rms})} \), (i) the peak inverse voltage \( PIV \) of the diode, (j) the average output power \( P_{o(\text{ac})} \), (k) the DC output power \( P_{o(\text{dc})} \), and (l) the frequency \( f_r \) of the output ripple voltage.

5.2 The single-phase half-wave rectifier of Fig. 5.1(a) is connected to a sinusoidal source of \( V_s = 220 \) V (rms), 50 Hz. Express the instantaneous output voltage \( v_O(t) \) by a Fourier series.
5.3 The single-phase rectifier shown in Fig. 5.3(a) is employed as a battery charger. The battery capacity is 100 Wh, and the battery voltage is $E = 24$ V. The average charging current should be $I_{o(\text{av})} = 5$ A. The primary AC input voltage is $V_p = 120$ V (rms), 60 Hz, and the transformer has a turns ratio of $n = 2:1$.

a. Calculate the conduction angle $\delta$ of the diode, the current-limiting resistance $R$, the power rating $P_R$ of $R$, the charging time $h$ in hours, the rectification efficiency $\eta_R$, and the peak inverse voltage PIV of the diode.

b. Use PSpice/SPICE to plot $P_{O(\text{ac})}$ and $P_{O(\text{dc})}$ as a function of time. Use default model parameters.

5.4 The input voltage to the single-phase bridge rectifier of Fig. 5.8(a) is shown in Fig. P5.4. Determine (a) the average voltage $V_{o(\text{av})}$, (b) the rms output voltage $V_{o(\text{rms})}$, and (c) the ripple factor RF of the output voltage. Assume a transformer with turns ratio $n = 1:1$.

5.5 The single-phase full-wave center-tapped rectifier shown in Fig. 5.6(a) is supplied from a 220-V (rms), 50-Hz source through a step-down center-tapped transformer with turns ratio $n = 10:2$. The load resistance $R_L$ is 10 $\Omega$. Determine (a) the average output voltage $V_{o(\text{av})}$, (b) the average load current $I_{o(\text{av})}$, (c) the rms load voltage $V_{o(\text{rms})}$, (d) the rms load current $I_{o(\text{rms})}$, (e) the ripple factor RF of the output voltage, (f) the rms ripple voltage $V_{r(\text{rms})}$, (g) the average diode current $I_{D(\text{av})}$, (h) the rms diode current $I_{D(\text{rms})}$, (i) the peak inverse voltage PIV of the diodes, (j) the average output power $P_{O(\text{ac})}$, (k) the DC output power $P_{O(\text{dc})}$, and (l) the frequency $f_c$ of the output ripple voltage.

5.6 The single-phase full-wave rectifier of Fig. 5.6(a) is supplied from a 220-V (rms), 50-Hz source through a step-down center-tapped transformer with turns ratio $n = 10:2$.

a. Express the instantaneous output voltage $v_o(t)$ by a Fourier series.

b. Use PSpice/SPICE to calculate the harmonic components of the output voltage, up to and including the ninth harmonic. Use default model parameters of 1N4148 diodes.

5.7 The single-phase full-wave bridge rectifier of Fig. 5.8(a) is supplied directly from a 220-V (rms), 50-Hz source through a transformer with turns ratio $n = 10:1$. The load resistance $R_L$ is 100 $\Omega$. Determine (a) the average output voltage $V_{o(\text{av})}$, (b) the average load current $I_{o(\text{av})}$, (c) the rms load voltage $V_{o(\text{rms})}$, (d) the rms load current $I_{o(\text{rms})}$, (e) the ripple factor RF of the output voltage, (f) the rms ripple voltage $V_{r(\text{rms})}$, (g) the average diode current $I_{D(\text{av})}$, (h) the rms diode current $I_{D(\text{rms})}$, (i) the peak inverse voltage PIV of the diode, (j) the average output power $P_{O(\text{ac})}$, (k) the DC output power $P_{O(\text{dc})}$, and (l) the frequency $f_c$ of the output ripple voltage.

5.8 An AC voltmeter is constructed by using a DC meter and a bridge rectifier, as shown in Fig. 5.11(a). The meter has an internal resistance of $R_m = 50$ $\Omega$, and its average current is $I_m = 200$ mA for a full-scale deflection. The current-limiting resistance is $R_s = 2.5$ k$\Omega$.

a. Determine the rms value of the AC input voltage $V_S$ that will give a full-scale deflection if the input voltage $v_S$ is sinusoidal.

b. If this meter is used to measure the rms value of an input voltage with a triangular waveform, as shown in Fig. 5.11(b), calculate the necessary correction factor $K$ to be applied to the meter reading.
5.9 A DC meter has an internal resistance of $R_m = 50 \, \Omega$, and its full-scale deflection current is $I_m = 200 \, mA$.

The meter should read an rms input voltage of $V_s = 250$ at the full-scale deflection.

a. Design an AC voltmeter that uses the DC meter and a bridge rectifier, as shown in Fig. 5.11(a).

b. Use PSpice/SPICE to check your results by plotting the average meter current. Use default model parameters of 1N4148 diodes.

5.10 An AC voltmeter is constructed by using a DC meter and a bridge rectifier as shown in Fig. P5.10. The DC meter has an internal resistance of $R_m = 250 \, \Omega$, and the average meter current is $I_m = 1 \, mA$ for full-scale deflection.

a. Determine the rms input $V_s$ for full-scale deflection.

b. Use PSpice/SPICE to check your results by plotting the average meter current. Use default model parameters of 1N4148 diodes.

FIGURE P5.10

5.11 A single-phase bridge rectifier is shown in Fig. 5.9. The load resistance $R_L$ is 2.5 k$\Omega$, and the source resistance $R_s$ is 1 k$\Omega$.

a. Determine the transfer characteristic ($v_o$ versus $v_s$) of the rectifier.

b. Use PSpice/SPICE to plot the transfer characteristic for $v_s = -10 \, V$ to 10 V. Use default model parameters of 1N4148 diodes.

5.12 Repeat Prob. 5.11 for the half-wave rectifier of Fig. 5.1(a).

5.13 The single-phase bridge rectifier shown in Fig. 5.12(a) is supplied directly from a 220-V (rms), 50-Hz source without any input transformer. The load resistance is $R_L = 1 \, k\Omega$.

a. Design an $L$ filter so that the rms ripple current $I_{r\text{rms}}$ is limited to less than 5% of $I_{o\text{av}}$. Assume that the second harmonic $I_{o2\text{rms}}$ is the dominant one and that the effects of higher-order harmonics are negligible.

b. Use PSpice/SPICE to check your design by plotting the output current. Use default model parameters of 1N4148 diodes.

5.14 Repeat Prob. 5.13 for the half-wave rectifier of Fig. 5.2(a). Assume that the first harmonic is the dominant one. Also assume a turns ratio of $n = 1:1$.

5.15 The single-phase full-wave bridge rectifier of Fig. 5.15(a) is supplied directly from a 120-V (rms), 60-Hz source without any input transformer. The load resistance is $R_L = 1 \, k\Omega$. Assume that the second harmonic is the dominant one.

a. Design a $C$ filter so that the rms ripple voltage $V_{r\text{rms}}$ is limited to less than 5% of $V_{o\text{av}}$.

b. With the value of $C$ found in part (a), calculate the average output voltage $V_{o\text{av}}$, and the capacitor voltage if the load resistance $R_L$ is disconnected.

c. Use PSpice/SPICE to check your design by plotting the instantaneous output voltage $v_o$. Use default model parameters of 1N4148 diodes.
5.16 Repeat Prob. 5.15 for the half-wave rectifier of Fig. 5.2(a). Assume that the first harmonic is the dominant one.

5.17 Measurements of the output of a full-wave bridge rectifier give \( V_{o\text{(av)}} = 150 \text{ V} \), \( I_{o\text{(av)}} = 120 \text{ mA} \), and \( V_{o\text{(rms)}} = 155 \text{ V} \). The rectifier uses a \( C \) filter across the load resistance. The supply frequency \( f \) is 60 Hz.

a. Determine the ripple factor \( RF \) of the output voltage and the value of the filter capacitance \( C \).

b. Use PSpice/SPICE to check your results by plotting the average output voltage. Use default model parameters of 1N4148 diodes.

5.18 The single-phase bridge rectifier of Fig. 5.18(a) is supplied from a 120-V (rms), 60-Hz source without any input transformer. The load resistance is \( R_L = 2 \text{ k}\Omega \). Assume that the second harmonic is the dominant one.

a. Design an \( LC \) filter so that the rms ripple voltage \( V_{r\text{(rms)}} \) is limited to less than 5% of \( V_{o\text{(av)}} \).

b. Use PSpice/SPICE to check your design by plotting the instantaneous output voltage \( v_O \). Use default model parameters of 1N4148 diodes.

5.19 Repeat Prob. 5.18 for the half-wave rectifier of Fig. 5.2(a). Assume that the first harmonic is the dominant one. Also assume a turns ratio of \( n = 1:1 \).

5.20 The single-phase bridge rectifier shown in Fig. P5.20 is used as a power supply.

a. Determine the DC output voltage for a load current of \( I_{o\text{(av)}} = 104 \text{ mA} \) and the ripple factor of the output voltage for \( R_L = 1 \text{ k}\Omega \).

b. Use PSpice/SPICE to check your results by plotting the average output voltage. Use default model parameters of 1N4148 diodes.

5.21 Repeat Prob. 5.20 for the load of Fig. P5.21 with \( C = 100 \mu\text{F} \) and \( R_L = 1 \text{ k}\Omega \).

5.22 Repeat Prob. 5.20 for the load of Fig. P5.22 with \( L = 5 \text{ mH} \) and \( R_L = 1.5 \text{ k}\Omega \).
5.4 Diode Peak Detectors and Demodulators

5.23 The carrier frequency $f_c$ of a radio signal is 250 kHz, and the modulating frequency $f_m$ is 10 kHz. The load resistance $R$ of the detector is 10 kΩ.

a. Design a demodulator for the waveform of Fig. 5.22(a) by determining the value of capacitance $C$ in Fig. 5.21(a).

b. Use PSpice/SPICE to plot the output voltage $v_O$ for a modulation index of $M = 0.5$ and a peak modulating voltage of $V_m = 20$ V. Use default model parameters.

5.24 Repeat Prob. 5.23(b) for a modulation index of $M = 1$.

5.5 Diode Clippers

5.25 The clipper circuit shown in Fig. 5.27(a) is supplied from the input voltage shown in Fig. 5.27(b). The battery voltage is $E_1 = 20$ V. The peak diode current $I_{D(peak)}$ is to be limited to 50 mA. Determine (a) the value of resistance $R$, (b) the average diode current $I_{D(avg)}$ and the rms diode current $I_{D(rms)}$, and (c) the power rating $P_R$ of the resistance $R$.

5.26 The clipper circuit shown in Fig. 5.25(a) is supplied from a sinusoidal input voltage of $v_S = 20 \sin(2000\pi t)$. The battery voltage is $E_1 = 5$ V. The peak diode current $I_{D(peak)}$ is to be limited to 10 mA.

a. Determine the value of resistance $R$, the average diode current $I_{D(avg)}$ and the rms diode current $I_{D(rms)}$, and the power rating $P_R$ of the resistance $R$.

b. Use PSpice/SPICE to plot the diode current. Use default model parameters of 1N4148 diodes.

5.27 The input voltage to the clipper circuit shown in Fig. P5.27(a) is $v_S = 5 \sin(2000\pi t)$. If $E_1 = 2$ V and $R = 10$ kΩ, plot (a) the output voltage $v_O(t)$ as a function of time, (b) the transfer characteristic of $v_O$ versus $v_S$, and (c) the peak diode current $I_{D(peak)}$. Assume a diode voltage drop of $V_D = 0.7$ V.

5.28 Repeat Prob. 5.27 for the circuit shown in Fig. P5.27(b).

5.29 Repeat Prob. 5.27 for the circuit shown in Fig. P5.27(c).

5.30 Repeat Prob. 5.27 for the circuit shown in Fig. P5.27(d).
5.6 **Diode Clamping Circuits**

5.31 The input voltage $v_S$ to the clamping circuit of Fig. 5.31(a) is a sinusoidal voltage $v_S = 30 \sin (2000\pi t)$. The peak diode current $I_{D(\text{peak})}$ is to be limited to 0.5 A. Assume that $E_1 = 5$ V and that a limiting resistance $R_s$ is connected in series with $C$.

- **a.** Design the clamping circuit by determining the peak inverse voltage PIV of the diode and the values of $R_s$, $R$, and $C$.
- **b.** Use PSpice/SPICE to plot the output voltage $v_O$. Use default model parameters of 1N4148 diodes.

5.32 The input voltage $v_S$ to the clamping circuit of Fig. 5.31(a) is $v_S = 20 \sin (2000\pi t)$. The peak diode current $I_{D(\text{peak})}$ is to be limited to 0.5 A. Assume that $E_1 = 5$ V and that a series resistance $R_s$ is connected in series with $C$ to limit the diode current.

- **a.** Design the clamping circuit by determining the peak inverse voltage PIV of the diode and the values of $R_s$, $R$, and $C$.
- **b.** Use PSpice/SPICE to plot the output voltage $v_O$. Use default model parameters of 1N4148 diodes.

5.33 The input voltage to the clamping circuit shown in Fig. P5.33(a) is $v_S = 5 \sin (2000\pi t)$. If $E_1 = 2$ V, $C = 0.1$ $\mu$F, and $R = 1$ M$\Omega$, (a) plot the output voltage $v_O(t)$ as a function of time, and (b) repeat part (a) for Fig. P5.33(b). Assume a diode voltage drop of $V_D = 0.7$ V and that the time constant $\tau = RC$ is much larger than the period of the signal, $T = 1/f_S$.

**FIGURE P5.33**

![](image)

5.34 The input voltage to the clamping circuit shown in Fig. P5.34(a) is $v_S = 10 \sin (2000\pi t)$. If $E_1 = 5$ V, $C = 0.1$ $\mu$F, and $R = 1$ M$\Omega$, (a) plot the output voltage $v_O(t)$ as a function of time, and (b) repeat part (a) for Fig. P5.34(b). Assume a diode voltage drop of $V_D = 0.7$ V and that the time constant $\tau = RC$ is much larger than the period of the signal, $T = 1/f_S$.

**FIGURE P5.34**

![](image)

5.7 **Diode Voltage Multipliers**

5.35 Use PSpice/SPICE to plot the output voltage $v_O$ for the voltage doubler in Fig. 5.34(c). Assume $v_S = 10 \sin 120\pi t$ and $C_1 = C_2 = C_3 = C_4 = 0.1$ $\mu$F. Use default model parameters of 1N4148 diodes.

5.36 Use PSpice/SPICE to plot the output voltage $v_{O4}$ ($=v_{C4}$) for the voltage quadrupler in Fig. 5.36(a). Assume $v_S = 10 \sin 120\pi t$ and $C_1 = C_2 = C_3 = C_4 = 0.01$ $\mu$F. The resistances that are connected across diodes $D_1$, $D_2$, and $D_3$ are $R_1 = R_2 = R_3 = 5$ M$\Omega$ (not shown). Use default model parameters of 1N4148 diodes.
5.8 Diode Function Generators

5.37 The circuit parameters of the diode circuit in Fig. 5.40(a) are $R_1 = 10 \, \Omega$, $R_2 = 5 \, \Omega$, $R_3 = 2.5 \, \Omega$, $E_1 = 4 \, V$, and $E_2 = 10 \, V$.
   a. Plot the $v-i$ relationship of the circuit.
   b. Use PSpice/SPICE to plot the transfer characteristic for $v_S = 0$ to $12 \, V$. Use default model parameters of 1N4148 diodes.

5.38 A diode circuit is shown in Fig. P5.38. The circuit parameters are $R = 1 \, \Omega$ and $E = 4 \, V$.
   a. Derive an expression for the $v-i$ characteristic of the circuit. Plot the $v-i$ characteristic.
   b. Use PSpice/SPICE to check your results by plotting the $v-i$ characteristic for $v_S = -5 \, V$ to $10 \, V$. Use default model parameters of 1N4148 diodes.

5.39 A $v-i$ characteristic representing a square law is shown in Fig. P5.39.
   a. Design a diode circuit to generate this characteristic.
   b. Use PSpice/SPICE to check your design by plotting the $v-i$ characteristic. Use default model parameters of 1N4148 diodes.

5.40 The input voltage to the diode clipper circuit shown in Fig. P5.40(a) is $v_S = 10 \sin(2000\pi t)$. If $R_1 = 10 \, \Omega$, $R_2 = 20 \, \Omega$, and $R_3 = 20 \, \Omega$, plot (a) the output voltage $v_O(t)$ as a function of time and (b) the transfer characteristic of $v_O$ versus $v_S$. Assume an ideal diode drop of $V_D = 0$.

5.41 Repeat Prob. 5.40 for the circuit shown in Fig. P5.40(b) if $R_1 = 10 \, \Omega$, $R_2 = 40 \, \Omega$, and, $E_1 = 5 \, V$. 
CHAPTER 6

SEMICONDUCTORS AND pn JUNCTION CHARACTERISTICS

Learning Outcomes

After completing this chapter, students should be able to do the following:

- Describe the physical structure and depletion region of the \( pn \) junction.
- Determine characteristics of the zero-biased, reverse-biased, and forward-biased \( pn \) junction.
- Determine the space charge widths, electric fields, and capacitances of a reverse-biased \( pn \) junction.
- Describe characteristics of the Schottky barrier junction.
- Determine the small-signal frequency model of a \( pn \) junction diode.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>( n_o, p_o )</td>
<td>Electron and hole concentrations of a semiconductor material</td>
</tr>
<tr>
<td>( n_a, n_d, n_i )</td>
<td>Acceptor (hole), donor (electron), and intrinsic carrier concentrations</td>
</tr>
<tr>
<td>( N_a, N_d )</td>
<td>Net acceptor (hole) and donor (electron) carrier concentrations</td>
</tr>
<tr>
<td>( V_{bi}, V_j )</td>
<td>Built-in potential and junction potential</td>
</tr>
<tr>
<td>( n_{po}, p_{no} )</td>
<td>Minority carrier electrons at the edge of the ( p )-region and of the ( n )-region</td>
</tr>
</tbody>
</table>
6.1 Introduction

We have seen in Chapters 4 and 5 that pn junction diodes can be used for signal conversion and processing. The characteristic of a practical diode that distinguishes it from an ideal one is that the practical diode experiences a finite voltage drop when it conducts and exhibits nonlinear characteristics. This drop is typically in the range of 0.5 V to 0.7 V. If the input voltage to a diode circuit is high enough, this small drop can be ignored. The voltage drop may, however, cause a significant error in electronic circuits, and the diode characteristic should be taken into account in evaluating the performance of diode circuits. To understand the characteristic of a practical diode, we need a clear understanding of its physical operation.

The pn junction is a basic building block in semiconductor devices, and the theory of the pn junction is still the fundamental concept in the physics of semiconductor devices. Most semiconductor devices contain at least one pn junction. A semiconductor diode, which has only one junction, is an example of pn devices. Other semiconductor devices are formed by combining two or more pn junctions in various configurations such as bipolar junction transistors, field-effect transistors, and silicon-controlled rectifiers. The characteristics of these devices depend on the pn characteristics under different biasing conditions: zero-biased, reverse-biased, and forward-biased. Semiconductor materials are the essential ingredients for pn junctions and semiconductor devices. The properties of high-purity, single-crystal materials are fundamental to the design of semiconductor devices.

6.2 Semiconductor Materials

Junction diodes are made of semiconductor materials [1]. A pure semiconductor is called an intrinsic material in which the concentrations of electrons and holes are equal. The currents induced in pure semiconductors are very small. The most commonly used semiconductors are silicon and germanium (Group IV in the periodic table as shown in Table 6.1), and gallium arsenide (Group V). Silicon materials cost less than germanium materials and allow diodes to operate at higher temperatures. For this reason, germanium diodes are rarely used. Gallium arsenide (GaAs) diodes can operate at higher switching speeds and higher frequencies than silicon diodes and hence are preferable. However, gallium arsenide materials are more expensive than silicon materials, and gallium arsenide diodes are more difficult to manufacture, so they are generally used only for high-frequency applications. GaAs devices are expected to become increasingly important in electronic circuits.

Semiconductors are a group of materials having conductivities between those of metals and insulators. One fundamental characteristic of semiconductor materials is that their conductivity can be varied over several orders of magnitude if we add controlled amounts of impurity atoms. To increase conductivity,
controlled quantities of materials known as impurities are introduced into pure semiconductors, creating free electrons or holes. The current through a diode is the result of the flow of electrons and holes in a semiconductor when forces are applied. These electrons and holes are referred to as carriers. Electrons are negatively charged particles. A hole is the absence of an electron in a covalent bond and is like an independent positive charge. The electrons and holes flow in opposite directions, and the direction of the holes is the direction of the conventional current flow.

The process of adding carefully controlled amounts of impurities to pure semiconductors is known as doping. A semiconductor to which impurities have been added is referred to as extrinsic. Two types of impurities are normally used: $n$-type from Group V, such as antimony, phosphorus, and arsenic, and $p$-type from Group III, such as boron, gallium, and indium.

### 6.2.1 $n$-type Materials

The $n$-type impurities are pentavalent materials, with five electrons in the outermost shell of each atom. The addition of a controlled amount of an $n$-type impurity (having five valence electrons) to silicon or germanium (having four valence electrons) causes one electron to be loosely attached to the parent atom because only four electrons are needed to form a covalent bond within a silicon or germanium atom. If a small amount of energy, such as thermal energy, is added to the donor electron, the electron can become free, leaving behind a positively charged ion of the donor atom. At room temperature, there is sufficient energy to cause the redundant electron to break away from its parent atom; thus, a free electron is generated. This electron is free to move randomly within the semiconductor crystal. Thus, an $n$-type impurity donates free electrons to the semiconductor; for this reason, it is often referred to as a donor impurity. The resulting material is referred to as an $n$-type semiconductor ($n$ for the negatively charged electron). An $n$-type semiconductor is shown in Fig. 6.1(a).
With a sufficient amount of energy, the donor electrons can be elevated to the conduction band, making them free to move within the crystal. The impurity atom was originally neutral, and the removal of the redundant electron will cause the impurity atom to exhibit a positive charge equal to $+e$ and to remain fixed in the crystal lattice of the structure as shown in Fig. 6.1(a). Figure 6.1(b) shows the energy band levels of $n$-type semiconductors in complete ionized states. $E_c$ is the conduction energy level, $E_d$ is the donor energy level, $E_v$ is the valence energy level, and $E_{Fi}$ is the intrinsic Fermi energy level. $E_{Fi}$ determines the statistical distribution of electrons, and its level is in the middle of $E_c$ and $E_v$.

The relative dielectric constants and the effective masses of the semiconductor materials and their impurities are different. As a result, they have different ionization energies. Table 6.2 lists the impurity ionization energies in silicon, germanium, and gallium arsenide semiconductors.

### 6.2.2 $p$-type Materials

The $p$-type impurities are trivalent materials (Group III) with three valence electrons in the outer shell of each atom. The addition of a $p$-type impurity to silicon or germanium (Group IV) causes a vacancy

<table>
<thead>
<tr>
<th>Acceptors</th>
<th>Donors</th>
<th>Ionization Energy (eV) of Materials</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Si</td>
</tr>
<tr>
<td>Boron</td>
<td>0.045</td>
<td>0.0104</td>
</tr>
<tr>
<td>Aluminum</td>
<td>0.06</td>
<td>0.0102</td>
</tr>
<tr>
<td>Phosphorus</td>
<td>0.045</td>
<td>0.012</td>
</tr>
<tr>
<td>Arsenic</td>
<td>0.05</td>
<td>0.0127</td>
</tr>
<tr>
<td>Beryllium</td>
<td></td>
<td>0.028</td>
</tr>
<tr>
<td>Zinc</td>
<td></td>
<td>0.0307</td>
</tr>
<tr>
<td>Cadmium</td>
<td></td>
<td>0.0347</td>
</tr>
<tr>
<td>Silicon</td>
<td></td>
<td>0.0345</td>
</tr>
<tr>
<td>Germanium</td>
<td></td>
<td>0.0404</td>
</tr>
<tr>
<td>Selenium</td>
<td></td>
<td>0.0059</td>
</tr>
<tr>
<td>Tellurium</td>
<td></td>
<td>0.0058</td>
</tr>
<tr>
<td>Silicon</td>
<td></td>
<td>0.0058</td>
</tr>
<tr>
<td>Germanium</td>
<td></td>
<td>0.0061</td>
</tr>
</tbody>
</table>

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
for one electron in the vicinity of the impurity atom because four electrons are necessary to complete covalent bonds. A vacancy for an electron is like a hole, which is equivalent to a positive charge $+e$. If an electron were to occupy this “empty” position, its energy would have to be greater than that of the valence electrons. If the valence electrons gain a small amount of thermal energy and move about in the crystal, the “empty” position becomes occupied and other valence electron positions become vacated, thereby creating holes in the semiconductor material. This type of semiconductor material is referred to as a $p$-type material ($p$ for the positively charged hole). A $p$-type semiconductor is shown in Fig. 6.2(a).

At room temperature, there is sufficient energy to cause a nearby electron to move into the existing vacancy, in turn causing a vacancy elsewhere. In this way, the hole moves randomly within the semiconductor crystal. Thus, a $p$-type impurity accepts free electrons and is referred to as an acceptor impurity. With the electron it gains, the impurity atom exhibits a charge of $-e$ and remains fixed in the crystal lattice of the structure. With a sufficient amount of energy, the acceptor atom can generate holes in the valence band without generating electrons in the conduction band. Figure 6.2(b) shows the energy band levels of $p$-type semiconductors in complete ionized states where $E_a$ is the acceptor energy level.

### 6.2.3 Majority and Minority Carriers

So far, we have assumed that materials are perfect; but practical materials are imperfect. The holes are also present in imperfect $n$-type semiconductor materials because of thermal agitation of electrons and holes within the materials. Therefore, in an $n$-type semiconductor, the electrons are the majority carriers and the holes are the minority carriers. Similarly, in a $p$-type semiconductor, the holes are the majority carriers and the electrons are the minority carriers. Doping and the application of energy can create electrons and holes (carriers). But within the semiconductor, there is also a recombination process by which electrons and holes (carriers) are annihilated. Any deviation from thermal equilibrium tends to change the electron and hole concentrations in a semiconductor. Any increase in energy (such as temperature or light) increases the rate at which electrons and holes are thermally generated; their concentrations change with time until new equilibrium values are reached. The simplified process of electron–hole generation and recombination is shown in Fig. 6.3.
6.2.4 The Fermi Function

The Fermi function $f(E)$ specifies how many of the existing states at the energy $E$ will be filled with an electron, or equivalently under equilibrium conditions. That is, it specifies, under equilibrium conditions, the probability that an available state at an energy $E$ will be occupied by an electron, and it is expressed mathematically \[2, 3\] as

$$f(E) = \frac{1}{1 + e^{(E-E_F)/kT}} \quad (6.1)$$

where $E_F$ = Fermi energy or Fermi level

$T$ = temperature in kelvin (K)

$k$ = Boltzmann constant ($k = 8.617 \times 10^{-5}$ eV/K)

As the temperature approaches absolute zero, $T \approx 0$ K, the exponent term of Eq. (6.1) tends to infinity: $f(E) \to 0$ for $E > E_F$ and $f(E) \to 1$ for $E < E_F$. There is a sharp cutoff at the Fermi energy $E_F$. Therefore, all states at energies below $E_F$ will be filled, and all states at energies above $E_F$ will be empty. This is shown in Fig. 6.4(a).

As the system temperature increases above zero, $T > 0$ K, the exponent term of Eq. (6.1) has a finite value, and the function goes through a transition from a filled state to an empty state. For $E > E_F$, the function $f(E)$ decays exponentially to zero with increasing energy, and most states will be empty at the
valence energy \( E_v \approx E = E_F + 3kT \) because the exponential term in Eq. (6.1) will be large and \( f(E) \to 0 \). For \( E < E_F \), the function \( f(E) \) increases exponentially with decreasing energy, and most states will be filled at the conduction energy \( E_c \approx E = E_F - 3kT \) because the exponential term in Eq. (6.1) will be small and \( f(E) \to 1 \). Figure 6.4(b) shows the plots of \( f(E) \) and \( 1 - f(E) \). Note that the Fermi function applies only under equilibrium conditions and is valid for all materials—insulators, semiconductors, and metals. It is simply a statistical function associated with electrons in general and does not depend on the characteristics and parameters of the semiconductors.

If \( f(E) \) is the probability of electrons occupying states at a given energy \( E \), then the probability that a state is empty (not filled) at a given energy \( E \) is equal to \( 1 - f(E) \). Thus, the probability that a state is filled at the conduction band edge \( (E_c) \) must be equal to the probability that a state is empty at the valence band edge \( (E_v) \). That is,

\[
f(E_c) = 1 - f(E_v) \quad (6.2)
\]

From Eq. (6.1) we get the probability function \( f(E_c) \) at \( E = E_c \) and \( 1 - f(E_v) \) at \( E = E_v \):

\[
f(E_c) = \frac{1}{1 + e^{(E_c - E_v)/kT}} \quad (6.3)
\]

\[
\approx e^{-(E_c - E_v)/kT} \quad \text{for } (E_c - E_F) \approx 3kT \quad \text{and } \quad e^{(E_c - E_v)\geq3kT} \gg 1
\]

\[
1 - f(E_v) = 1 - \frac{1}{1 + e^{(E_v - E_c)/kT}} = \frac{1}{1 + e^{(E_F - E_v)/kT}} \quad (6.5)
\]

\[
\approx e^{-(E_F - E_v)/kT} \quad \text{for } (E_F - E_v) \approx 3kT \quad \text{and } \quad e^{(E_F - E_v)\geq3kT} \gg 1
\]

Equating Eq. (6.3) to Eq. (6.5), we get

\[
\frac{1}{1 + e^{(E_c - E_v)/kT}} = \frac{1}{1 + e^{(E_F - E_v)/kT}} \quad (6.7)
\]

which can be solved for the Fermi energy \( E_F \):

\[
E_F = \frac{E_c + E_v}{2} \quad (6.8)
\]

Therefore, the Fermi energy level is positioned at the middle of the energy band.

### 6.2.5 Carrier Concentrations

The concentrations of electrons \( (n) \) and holes \( (p) \) depend on the amount of impurity doping and on the temperature. We can apply Eq. (6.4) to determine the thermal equilibrium electron concentration in the conduction band, which depends on the conduction band energy level and the temperature as given by

\[
n_o = N_c \ e^{-(E_c - E_F)/kT} \quad (6.9)
\]
where the parameter $N_c$ is called the \textit{effective density states function in the conduction band}. Its value depends on the effective mass values of the $n$-type semiconductor materials and the temperature as given by [2]

$$N_c = 2 \left( \frac{2\pi m_n^* kT}{\hbar^2} \right)^{3/2}$$  \hfill (6.10)

where $m_n^*$ = the effective mass of a free electron and $h$ = Planck’s constant. Assuming $m_n^* \approx m_o$ (mass of a free electron), $N_c = 2.5 \times 10^{19} \text{ cm}^{-3}$ at $T = 300$ K for most semiconductors. Similarly, we can apply Eq. (6.6) to determine the thermal equilibrium hole concentration in the valence band, which depends on the valence band energy level and the temperature as given by

$$p_o = N_v \ e^{-(E_F-E_v)/kT}$$  \hfill (6.11)

where the parameter $N_v$ is called the \textit{effective density states function in the valence band}. Its value depends on the effective mass values of the $p$-type semiconductor materials and the temperature as given by

$$N_v = 2 \left( \frac{2\pi m_p^* kT}{\hbar^2} \right)^{3/2}$$  \hfill (6.12)

where $m_p^* \approx m_o$ = the effective mass of a hole and $h$ = Planck’s constant. Assuming $m_p^* = m_o$ (mass of a free hole), $N_v = 1 \times 10^{19} \text{ cm}^{-3}$ at $T = 300$ K for most semiconductors. The calculated values of $N_c$ and $N_v$ at $T = 300$ K are listed in Table 6.3. An intrinsic semiconductor will have a Fermi energy level called the \textit{intrinsic Fermi energy}, $E_{Fi} = E_F$. From Eqs. (6.9) and (6.11), we can find the intrinsic concentrations of electrons and holes as

$$n_i = n_o = N_c \ e^{-(E_c-E_{Fi})/kT}$$  \hfill (6.13)

$$p_i = n_i = p_o = N_v \ e^{-(E_v-E_{Fi})/kT}$$  \hfill (6.14)

If we take the product of $n_i$ in Eq. (6.13) and $p_i$ in Eq. (6.14), we can find

$$n_i p_i = n_i^2 = N_c \ e^{-(E_c-E_{Fi})/kT} \times N_v \ e^{-(E_v-E_{Fi})/kT}$$

which can be simplified as follows:

$$n_i^2 = N_c N_v \ e^{-(E_c-E_{Fi})/kT} = N_c N_v \ e^{-E_g/kT}$$  \hfill (6.15)

Here $E_g$ is the conduction band energy, and $n_i$ refers to either the intrinsic electron or hole concentration in the semiconductor material. The value of $n_i$ is constant for a given semiconductor material at a constant temperature, and it is independent of the Fermi energy. The calculated values of $n_i$ from Eq. (6.15) for $E_g = 1.12$ eV and $T = 300$ K are also listed in Table 6.3.

Under thermal equilibrium conditions at a given temperature, the product of the amount of electron concentration $n_o$ and the amount of hole concentration $p_o$ is always constant for a given semiconductor material. That is,

$$n_o p_o = n_i^2$$  \hfill (6.16)
TABLE 6.3  Effective density of states function for conduction and valence bands

<table>
<thead>
<tr>
<th>Materials at $T = 300 \text{ K}$</th>
<th>$N_c (\text{cm}^{-3})$</th>
<th>$N_v (\text{cm}^{-3})$</th>
<th>$m^*_n/m_o$</th>
<th>$m^*_p/m_o$</th>
<th>$n_i (\text{cm}^{-3})$ at $E_g = 1.12 \text{ eV}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Silicon</td>
<td>$2.8 \times 10^{19}$</td>
<td>$1.04 \times 10^{19}$</td>
<td>1.08</td>
<td>0.56</td>
<td>$1.5 \times 10^{10}$</td>
</tr>
<tr>
<td>Germanium</td>
<td>$1.04 \times 10^{19}$</td>
<td>$6.0 \times 10^{18}$</td>
<td>0.55</td>
<td>0.37</td>
<td>$2.4 \times 10^{13}$</td>
</tr>
<tr>
<td>Gallium arsenide</td>
<td>$4.7 \times 10^{17}$</td>
<td>$7.0 \times 10^{18}$</td>
<td>0.067</td>
<td>0.48</td>
<td>$1.8 \times 10^{6}$</td>
</tr>
</tbody>
</table>

For example, if $n_o = 1 \times 10^{16} \text{ cm}^{-3}$ for silicon at $T = 300 \text{ K}$, then

$$p_o = \frac{n_i^2}{n_o} = \frac{(1.5 \times 10^{10})^2}{1 \times 10^{16}} = 2.25 \times 10^4 \text{ cm}^{-3}$$

### KEY POINTS OF SECTION 6.2

- Free electrons (in $n$-type material) and holes (in $p$-type material) are made available by adding a controlled amount of $n$-type impurities and $p$-type impurities to pure semiconductors, respectively.
- The Fermi function specifies the probability that an available state at a given energy will be occupied by an electron.
- The intrinsic concentration of a semiconductor material remains constant at a steady temperature, and it is independent of the Fermi energy.

### 6.3 Zero-Biased $pn$ Junction

To consider the operation principle of a $pn$ junction, we will assume that a $p$-type material is laid into one side of a single crystal of a semiconductor material and an $n$-type material is laid into the other side, as shown in Fig. 6.5(a). (This is not, however, the way to make a diode.) The doping profile of the impurity doping concentrations in the $p$-region ($N_a$) and $n$-region ($N_d$) is shown in Fig. 6.5(b) with the assumption that the doping concentration is uniform in each region.

At room temperature, the electrons, which are majority carriers in the $n$-region, diffuse from the $n$-type side to the $p$-type side; the holes, which are majority carriers in the $p$-region, diffuse from the $p$-type side to the $n$-type side. The electrons and holes will recombine near the junction and thus cancel each other out. There will be opposite charges on each side of the junction, creating a depletion region, or space charge region, as shown in Fig. 6.5(c). Under thermal equilibrium conditions at a given temperature, no more electrons or holes will cross the junction.

Because opposite charges are present on each side of the junction, an electric field is established across the junction. The resultant junction potential barrier $V_j$, which arises because the $n$-type side is at a higher potential than the $p$-type side, prevents any flow of majority carriers to the other side. The variation of the potential across the junction is shown in Fig. 6.5(d). $V_j$ is also called the built-in potential $V_{bi}$. 
Because of the potential barrier $V_j$, the electrons, which are minority carriers in the $p$-side, will be swept across the junction to the $n$-side; the holes, which are minority carriers in the $n$-side, will be swept across the junction to the $p$-side. Therefore, a current caused by the minority carriers (holes) will flow from the $n$-side to the $p$-side; it is known as the reverse drift current $I_{DR}$. Similarly, a current known as the forward diffusion current $I_{DF}$ will flow from the $p$-side to the $n$-side, caused by minority electrons. Under equilibrium conditions, the resultant current will be zero. Therefore, these two currents ($I_{DF}$ and $I_{DR}$) are equal and flow in opposite directions. That is,

$$I_{DF} = -I_{DR}$$

**6.3.1 Built-In Junction Potential**

The energy bands in the neutral $p$- and $n$-regions on either side of the space charge region must bend due to the potential barriers $e\phi_{FP}$ in the $p$-type and $e\phi_{FN}$ in the $n$-type. However, the Fermi energy level ($E_F$) is constant throughout the entire system at thermal equilibrium, as shown in Fig. 6.6.

The intrinsic energy level $E_{Fi}$ in the $p$- and $n$-regions is always equidistant from $E_c$ and $E_v$. $E_{Fi}$ determines the total junction potential $V_{bi}$, which is the difference between the intrinsic Fermi levels in the $p$- and $n$-regions as given by

$$V_{bi} = |\phi_{FP}| + |\phi_{FN}|$$
The hole concentration in the conduction band in the $p$-region decays exponentially and can be determined from

$$
n_p = n_i e^{-(E_F - E_{Fi})/kT} = n_i e^{-e\phi_{FP}/kT} \tag{6.19}
$$

Here $n_i$ and $E_{Fi}$ are the intrinsic carrier concentration and the intrinsic Fermi energy, respectively, in the $n$-region. $\phi_{FP}$ is the potential barrier in the $p$-region. If we take a natural log of both sides, Eq. (6.19) gives

$$
\phi_{FP} = -\frac{kT}{e} \ln \left( \frac{n_p}{n_i} \right) \tag{6.20}
$$

The electron concentration in the conduction band in the $n$-region can be determined from

$$
n_n = n_i e^{-(E_F - E_{Fi})/kT} = n_i e^{+e\phi_{FN}/kT} \tag{6.21}
$$

where $n_i$ and $E_{Fi}$ are the intrinsic carrier concentration and the intrinsic Fermi energy, respectively, in the $n$-region. $+\phi_{FN}$ is the potential barrier in the $n$-region. If we take a natural log of both sides, Eq. (6.21) gives

$$
\phi_{FN} = \frac{kT}{e} \ln \left( \frac{n_o}{n_i} \right) \tag{6.22}
$$

Substituting $\phi_{FP}$ from Eq. (6.20) and $\phi_{FN}$ from Eq. (6.22) into Eq. (6.18) gives

$$
V_{bi} = \frac{kT}{e} \ln \left( \frac{n_p}{n_i} \right) + \frac{kT}{e} \ln \left( \frac{n_n}{n_i} \right) = V_T \ln \left( \frac{n_p n_p}{n_i^2} \right) \tag{6.23}
$$

where $V_T = kT/e$ is defined as the thermal voltage. Assuming that $n_n$ equals the net donor concentration $N_d$ of the $n$-region and $n_p$ equals the net acceptor concentration $N_a$ in the $p$-region, we can write Eq. (6.23) as

$$
V_{bi} = \frac{kT}{e} \ln \left( \frac{N_d N_a}{n_i^2} \right) = V_T \ln \left( \frac{N_d N_a}{n_i^2} \right) \tag{6.24}
$$
Therefore, the junction potential depends on the donor and acceptor concentrations \(N_d, N_a\), the temperature \(K\), and the intrinsic concentration \(n_i\). For example, if \(T = 300\) K, \(V_T = 25.8\) mV, \(n_i = 1.5 \times 10^{10}\) cm\(^{-3}\), \(N_a = 2 \times 10^{16}\) cm\(^{-3}\), and \(N_d = 5 \times 10^{15}\) cm\(^{-3}\), Eq. (6.24) gives \(V_{bi} = 0.695\) V.

### 6.3.2 Electric Field Distribution

Assume that the doping concentration \((N_a\) or \(N_d\)) is uniform in each region and there is an abrupt change in doping at the junction. Initially, there is a step function gradient of the space charge density in both the electron and hole concentrations, as shown in Fig. 6.7(a). The space charge region extends from \(+x_n\) to \(-x_p\). The distribution abruptly ends in the \(n\)-region at \(+x_n\) and abruptly ends in the \(p\)-region at \(-x_p\).

We can determine the electric field by applying the one-dimensional Poisson’s equation as given by

\[
\frac{d^2\phi(x)}{dx^2} = \frac{\rho(x)}{\varepsilon_s} = \frac{de(x)}{dx}
\]  

(6.25)

where \(\phi(x) = \) electric potential
\(\varepsilon(x) = \) electric field
\(\rho(x) = \) volume charge density, and
\(\varepsilon_s = \) permittivity of the semiconductor

\(\varepsilon_s\) equals to the product of the relative permeability \(\varepsilon_r = 11.7\) and the permeability of the free air \(\varepsilon_o = 8.85 \times 10^{-14}\). That is, \(\varepsilon_s = \varepsilon_r\varepsilon_o\). Note that \(\varepsilon(x)\) is the variable electric field as a function of \(x\) while \(\varepsilon\) denotes a fixed value of electric field.

From Fig. 6.7(a), the charge densities are

\[
\rho(x) = \begin{cases} 
+eN_d & 0 < x < x_n \\
-eN_a & -x_p < x < 0
\end{cases}
\]  

(6.26)

We can find the electric field in the \(p\)-region by integrating Eq. (6.25) as

\[
\varepsilon_p(x) = \int \frac{\rho(x)}{\varepsilon_s} dx = \int \frac{-eN_a}{\varepsilon_s} dx = \frac{-eN_a}{\varepsilon_s}x + C_1
\]  

(6.27)

---

**FIGURE 6.7**  Space charge density and electric field
Here $C_1$ is a constant of integration that can be found from the final condition $\varepsilon = 0$ at $x = -x_p$; that is, if $\varepsilon_p(x = -x_p) = 0$, we find $C_1 = -eN_a x_p/e_s$. Substituting $C_1$ in Eq. (6.27) gives

$$\varepsilon_p(x) = \frac{-eN_a}{e_s} (x + x_p) \quad -x_p < x < 0$$  \hspace{1cm} (6.28)

which is a linear function of distance $x$ in the $p$-region. Similarly, we can find the electric field in the $n$-region from

$$\varepsilon_n(x) = \int \frac{\rho(x)}{e_s} dx = \int \frac{eN_d}{e_s} dx = \frac{eN_d}{e_s} x + C_2$$  \hspace{1cm} (6.29)

Here $C_2$ is a constant of integration that can be found from the final condition $\varepsilon = 0$ at $x = x_n$; that is, if $\varepsilon_n(x = x_n) = 0$, we find $C_2 = -eN_d x_n/e_s$. Substituting $C_2$ in Eq. (6.29) gives

$$\varepsilon_n(x) = \frac{-eN_d}{e_s} (x_n - x) \quad 0 < x < x_n$$  \hspace{1cm} (6.30)

The electric field, which becomes maximum at $x = 0$, is given by

$$\varepsilon_{\text{max}} = \frac{-eN_d x_n}{e_s} = \frac{-eN_d x_p}{e_s}$$  \hspace{1cm} (6.31)

since the electric field is continuous at the junction at $x = 0$. By setting the field in Eq. (6.28) equal to the field in Eq. (6.30) at $x = 0$, we get

$$N_d x_p = N_d x_n$$  \hspace{1cm} (6.32)

Thus the number of negative charges per unit area in the $p$-region is equal to the number of positive charges per unit area in the $n$-region. An electric field exists in the depletion region, and the plot of the electric field in the depletion region is shown in Fig. 6.7(b). The electric field is a linear function of the distance $x$, and it becomes maximum at $x = 0$.

### 6.3.3 Junction Potential Distribution

The junction potential in the $p$-region can be found by integrating the electric field in Eq. (6.28)

$$\phi_p(x) = -\int \varepsilon(x) \, dx = \int \frac{eN_a}{e_s} (x_p + x) \, dx = \frac{eN_a}{e_s} \left( x_p x + \frac{x^2}{2} \right) + C_3$$  \hspace{1cm} (6.33)

Here $C_3$ is a constant of integration that can be found from the final condition $\varepsilon = 0$ at $x = -x_p$; that is, if $\varepsilon(x = -x_p) = 0$, we find $C_3 = eN_a x_p^2/2e_s$. Substituting $C_3$ in Eq. (6.33) gives

$$\phi_p(x) = \frac{eN_a}{2e_s} (x_p + x_p)^2 \quad -x_p < x < 0$$  \hspace{1cm} (6.34)

Similarly, we can find the junction potential in the $n$-region from

$$\phi_n(x) = -\int \varepsilon(x) \, dx = \int \frac{eN_d}{e_s} (x_n - x) \, dx = \frac{eN_d}{e_s} \left( x_n x - \frac{x^2}{2} \right) + C_4$$  \hspace{1cm} (6.35)

Here $C_4$ is a constant of integration that can be found by equating $\phi_p(x)$ in Eq. (6.34) at $x = 0$ to $\phi_n(x)$ at $x = 0$ because the potential is a continuous function. Thus for $\phi_n(x = 0) = \phi_p(x = 0)$, we find $C_4 = \phi_p(x = 0) = eN_a x_p^2/2e_s$. Substituting $C_4$ into Eq. (6.35) gives

$$\phi_n(x) = \frac{eN_d}{e_s} \left( x_n x - \frac{x^2}{2} \right) + \frac{eN_a}{2e_s} x_p^2 \quad 0 < x < x_n$$  \hspace{1cm} (6.36)
The plot of the potential $\phi_n$ against the distance $x$ is shown in Fig. 6.8, which shows a quadratic dependence on distance $x$. The magnitude of the junction potential at $x = x_n$ is equal to the built-in potential barrier $V_{bi}$ or the junction potential $V_j$. Thus, for $x = x_n$ we can find the built-in potential from Eq. (6.36) as

$$V_{bi} = |\phi_n(x = x_n)| = \frac{e}{2e_s}(N_d x_n^2 + N_a x_p^2)$$  \hspace{1cm} (6.37)

Since the potential energy of an electron is related to the potential $\phi_n(x)$ by $E = -e\phi(x)$, the electron energy $e\phi_{Fn}$ (or the hole energy $e\phi_{Fp}$) also varies as a quadratic function of distance through the space charge region. The plot of the energy band diagram, which is a quadratic dependence on the distance $x$, is shown in Fig. 6.9. Thus, the conduction, valence, and intrinsic Fermi energy levels vary with distance in a semiconductor.

### 6.3.4 Space Charge Depletion Width

The distance of the space charge region that extends into the $p$-region can be found from Eq. (6.32) as

$$x_p = \frac{N_d x_n}{N_a}$$  \hspace{1cm} (6.38)

Substituting $x_p$ from Eq. (6.38) into Eq. (6.37) and solving for $x_n$, we get the space charge extension $x_n$ in the $n$-region as

$$x_n = \sqrt{\frac{2e_s V_{bi}}{e} \left( \frac{N_a}{N_d} \left( \frac{1}{N_a + N_d} \right) \right)}$$  \hspace{1cm} (6.39)
Similarly, substituting \( x_n = N_a x_p / N_d \) from Eq. (6.32) into Eq. (6.37), and then solving for \( x_p \), we get the space charge extension in the \( p \)-region as

\[
x_p = \sqrt{\frac{2eV_{bi}}{e} \left( \frac{N_d}{N_a} \right) \left( \frac{1}{N_a + N_d} \right)}
\]

(6.40)

Therefore, the total width \( W \) of the depletion or space charge region is the sum of \( x_n \) and \( x_p \):

\[
W = x_n + x_p = \sqrt{\frac{2eV_{bi}}{e} \left( \frac{N_a + N_d}{N_a N_d} \right)}
\]

(6.41)

The space charge width \( W \) depends on the doping concentrations \( N_a \) and \( N_d \). Once the built-in potential \( V_{bi} \) is determined from Eq. (6.24), the total space charge region width \( W \) can be determined from Eq. (6.41). Substituting for \( x_n \) or \( x_p \), the maximum field in Eq. (6.31) can be related to the impurity concentrations \( N_d \) and \( N_a \) by

\[
e_{\text{max}} = -\frac{eN_a x_n}{\varepsilon_s} = \frac{-eN_a x_p}{\varepsilon_s} = -\sqrt{\frac{2eV_{bi} \left( \frac{N_d}{N_a + N_d} \right)}{\varepsilon_s}}
\]

(6.42)

which can also be written as a function of \( W \):

\[
e_{\text{max}} = -\frac{2V_{bi}}{W}
\]

(6.43)

**EXAMPLE 6.1**

*Finding the space charge widths and the peak electric field in a pn junction*  The parameters of a uniformly doped pn junction for silicon semiconductors are \( V_T = 26 \) mV, \( T = 25^\circ \text{C} \), \( N_a = 1 \times 10^{16} \text{ cm}^{-3} \), \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \), and \( n_i = 1.5 \times 10^{16} \text{ cm}^{-3} \).

Find (a) the depletion width \( W \) and (b) the maximum field \( e_{\text{max}} \).

**SOLUTION**

\( T = 25^\circ \text{C} \), \( N_a = 1 \times 10^{16} \text{ cm}^{-3} \), and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \).

\( \varepsilon_r = 11.7 \), \( \varepsilon_0 = 8.85 \times 10^{-14} \), \( e = 1.6 \times 10^{-19} \), \( k = 1.3806 \times 10^{-23} \)

\( T_k = 273 + T = 273 + 25 = 298 \text{ K} \), \( \varepsilon_s = \varepsilon_r \times \varepsilon_0 = 11.7 \times 8.85 \times 10^{-14} = 1.035 \times 10^{-12} \)

(a) From Eq. (6.24),

\[
V_{bi} = 26 \times 10^{-3} \times \ln \left[ \frac{1 \times 10^{16} \text{ cm}^{-3} \times 2 \times 10^{15} \text{ cm}^{-3}}{(1.5 \times 10^{16})^2 \text{ cm}^2} \right] = 0.648 \text{ V}
\]

From Eq. (6.39),

\[
x_n = \sqrt{\frac{2 \times 1.035 \times 10^{-12} \times 0.648 \times (1 \times 10^{16} \text{ cm}^{-3})}{1.6 \times 10^{-19} \left( 2 \times 10^{15} \text{ cm}^{-3} + 1 \times 10^{16} \text{ cm}^{-3} + 2 \times 10^{15} \text{ cm}^{-3} \right)}} = 0.5913 \mu\text{m}
\]
KEY POINTS OF SECTION 6.3

- A semiconductor diode is formed by sandwiching a p-type material into one side and an n-type material in the other side of a single crystal.
- The built-in potential depends on the donor and acceptor concentrations, which are a strong function of temperature.
- An electric field exists in the depletion region. The width of the space charge region depends on the doping concentrations.

6.4 Reverse-Biased \textit{pn} Junction

A \textit{pn} junction is said to be reverse biased if the 	extit{n}-side is made positive with respect to the 	extit{p}-side, as depicted in Fig. 6.10(a). If the reverse voltage $V_R = V_D$ is increased, the potential barrier is increased from $V_{bi}$ to $V_{bi} + V_R$ as shown in Fig. 6.10(b). The holes from the 	extit{p}-side and the electrons from the 	extit{n}-side cannot cross the junction, and the diffusion current $I_{DR}$ due to the majority carriers will be negligible. Because of a higher potential barrier, however, the minority holes in the 	extit{n}-side will be swept easily across the junction to the 	extit{p}-side; the minority electrons in the 	extit{p}-side will be swept across the junction to the 	extit{n}-side. Thus, the current will flow solely due to the minority carriers. The reverse current flow will be due to the drift current $I_{DR}$, which is known as the reverse saturation (or leakage) current, denoted by $I_S$ as in Eq. (4.1).

The number of minority carriers available is very small, and consequently the resulting current is also very small, on the order of pico-amperes. The production of minority carriers is dependent on the temperature. Thus, if the reverse voltage $V_R$ is increased further, the diode current remains almost constant until a breakdown condition is reached. If the temperature increases, however, the reverse diode current also increases. The width of the depletion region grows with an increase in the reverse voltage.

Since there will not be an equilibrium condition in the 	extit{p}- and 	extit{n}-regions, the Fermi energy level will no longer be constant through the system. Figure 6.10(c) shows the energy band diagram of the \textit{pn} junction. $E_c$ and $E_v$ are shifted by the total voltage $V_{PB} = V_{bi} + V_R$. As $V_R$ pushes the energy levels, the Fermi level on the 	extit{n}-side $E_{Fn}$ is now below the Fermi level on the 	extit{p}-side $E_{FP}$. The difference between $E_{FP}$ and $E_{Fn}$ is equal to $eV_R$—that is, $E_{FP} - E_{Fn} = eV_R$.  

From Eq. (6.40),

$$x_n = \sqrt{\frac{2 \times 1.035 \times 10^{-12} \times 0.648 \left(\frac{2 \times 10^{15} \text{cm}^{-3}}{1 \times 10^{16} \text{cm}^{-3}}\right)}{1.6 \times 10^{-19}}} \frac{1}{1 \times 10^{16} \text{cm}^{-3} + 2 \times 10^{15} \text{cm}^{-3}}} = 0.1183 \text{ \mu m}$$

Therefore, $W = x_n + x_p = 0.5913 \text{ \mu m} + 0.1183 \text{ \mu m} = 0.7096 \text{ \mu m}$.

(b) From Eq. (6.42),

$$\varepsilon_{\text{max}} = \frac{-eN_d x_n}{\varepsilon_s} = \frac{-1.6 \times 10^{-19} \times 2 \times 10^{15} \text{cm}^{-3} \times 0.5913 \text{ \mu m}}{1.035 \times 10^{-12} \text{cm}^{-1}} = -1.827 \times 10^3 \text{ V/cm}$$
6.4.1 Breakdown Condition

If the reverse voltage is kept sufficiently high, the electric field in the depletion layer will be strong enough to break the covalent bonds of silicon (or germanium) atoms, producing a large number of electron–hole pairs throughout the semiconductor crystal. These electrons and holes give rise to a large reverse current flow. The depletion region (often called the space charge region) becomes so wide that collisions are less likely, but the even more intense electric field has the force to break the bonds directly. This phenomenon is called the tunneling effect or the zener effect. The mechanism is known as zener breakdown, in which case electrons and holes in turn cancel the negative and positive charges of the depletion region, and the junction potential barrier is virtually removed. The reverse current is then limited by the external circuit only, while the reverse terminal voltage remains almost constant at the zener voltage $V_z$ (see Sec. 4.7).

When the high electric field becomes strong enough, the electrons in the p-side will be accelerated through the crystal and will collide with the unbroken covalent bonds with a force sufficient to break them. The electrons generated by the collisions may gain enough kinetic energy to strike other unbroken bonds with sufficient force to break them as well. This cumulative effect, which will result in a large amount of uncontrolled current flow, is known as an avalanche breakdown.

In practice, the zener and avalanche effects are indistinguishable because both lead to a large reverse current. When a breakdown occurs at $V_z < 5$ V (as in heavily doped junctions), it is a zener breakdown.
When a breakdown occurs at $V_z > 7$ V (approximately), it is an avalanche breakdown. When a junction breaks down at a voltage between 5 V and 7 V, the breakdown can be either a zener or an avalanche breakdown or a combination of the two.

### 6.4.2 Depletion Region Width

With the reverse-biased voltage $V_R$, the total potential barrier will increase from $V_{bi}$ to $V_{bi} + V_R$. Thus Eq. (6.18) can be modified to obtain the total effective potential barrier as

$$V_{PB} = |\phi_{Fp}| + |\phi_{Fn}| + V_R = V_{bi} + V_R \quad (6.44)$$

Substituting for $V_{bi}$ with $V_{bi} + V_R$ in Eqs. (6.39), (6.40), and (6.41), we can obtain the space charge extension in the $n$- and $p$-regions as

$$x_n = \sqrt{\frac{2e_d(V_{bi} + V_R)}{e} \left( \frac{N_a}{N_d} \right) \left( \frac{1}{N_a + N_d} \right)} \quad (6.45)$$

$$x_p = \sqrt{\frac{2e_d(V_{bi} + V_R)}{e} \left( \frac{N_d}{N_a} \right) \left( \frac{1}{N_a + N_d} \right)} \quad (6.46)$$

Therefore, the total width $W$ of the depletion or space charge region is the sum of $x_n$ and $x_p$:

$$W = x_n + x_p = \sqrt{\frac{2e_d(V_{bi} + V_R)}{e} \left( \frac{N_a + N_d}{N_aN_d} \right)} \quad (6.47)$$

Thus, the depletion width $W$ increases with an increasing reverse-biased voltage $V_R$. Since $x_n$ in Eq. (6.45) and $x_p$ in Eq. (6.46) increase with reverse-biased voltage $V_R$, the magnitudes of the electric fields in Eq. (6.28) and Eq. (6.30) also increase. We can find the maximum field from Eq. (6.43) as a function of $V_R$ and $W$:

$$\varepsilon_{max} = \frac{-2 (V_{bi} + V_R)}{W} \quad (6.48)$$

Thus, the maximum field increases with $V_R$ and decreases with $W$.

#### EXAMPLE 6.2

**Finding the depletion width in a reverse-biased pn junction**  
The parameters of a uniformly doped $pn$ junction for silicon semiconductors are $V_R = 10$ V, $V_T = 26$ mV, $T = 25\degree C$, $N_a = 1 \times 10^{16}$ cm$^{-3}$, $N_d = 2 \times 10^{15}$ cm$^{-3}$, $e = 1.6 \times 10^{-19}$, and $k = 1.3806 \times 10^{-23}$.

Find (a) the depletion width $W$ and (b) the maximum field $\varepsilon_{max}$.

**SOLUTION**

$$T = 25\degree C, \quad N_a = 1 \times 10^{16}$ cm$^{-3}$, and $N_d = 2 \times 10^{15}$ cm$^{-3}$. $$

$$\varepsilon_r = 11.7 \quad \varepsilon_o = 8.85 \times 10^{-14} \quad e = 1.6 \times 10^{-19} \quad k = 1.3806 \times 10^{-23}$$

$$T_k = 273 + T = 273 + 25 = 298 \ K \quad \varepsilon_s = \varepsilon_r \times \varepsilon_o = 11.7 \times 8.85 \times 10^{-14} = 1.035 \times 10^{-12}$$
6.4.3 Junction Capacitance

Since a depletion region has positive charges in one side and negative charges in another side, there will be a capacitance associated with the \( pn \) junction. To find the junction capacitance, let us consider a small increase in the reverse voltage by \( dV_R \), which will add an incremental positive charge \( dQ = eN_d dx_n \) in the \( n \)-region and an incremental negative charge \( -dQ = -eN_d dx_p \) in the \( p \)-region. These incremental changes are shown in Fig. 6.11.

Thus, the junction capacitance per square area with the variation of the reverse voltage becomes

\[
C_j = \frac{dQ}{dV_R} = \frac{eN_d dx_n}{dV_R} = \frac{eN_d dx_p}{dV_R} \tag{6.49}
\]

\( x_n \), Charge density
\( p \)-side \( +eN_d \)
\( n \)-side
\( -dx_p \)
\( -dQ \)
\( -eN_d \)
\( (V_R - dV_R) \)

**FIGURE 6.11** Incremental changes in the space charge width with an incremental change in reverse-biased voltage
Substituting $x_n$ from Eq. (6.45), we get

$$C_j = eN_d \frac{dx_n}{dV_R} = eN_d \frac{d}{dV_R} \left[ \frac{2e_s(V_{bi} + V_R)}{e} \left( \frac{N_a}{N_d} \right) \left( \frac{1}{N_a + N_d} \right) \right]^{1/2}$$  \hspace{1cm} (6.50)

After we complete the differentiation, this gives

$$C_j = \left[ \frac{ee_sN_aN_d}{(V_{bi} + V_R)(N_a + N_d)} \right]^{1/2} = \frac{K_j}{\sqrt{V_{bi} + V_R}}$$  \hspace{1cm} (6.51)

where $K_j$ is a constant for a specific $pn$ junction. Equation (6.51) can be expressed as function of $W$ in Eq. (6.47):

$$C_j = \frac{e_s}{W}$$  \hspace{1cm} (6.52)

Therefore, the junction capacitance $C_j$ decreases with the reverse voltage $V_R$ and the depletion width $W$. $C_j$ is also referred to as the depletion layer capacitance. We can obtain the same expression for $C_j$ if we use $x_p$ from Eq. (6.46) in $dQ = eN_d dx_p$.

**EXAMPLE 6.3**

**Finding the junction capacitance of a reverse-biased $pn$ junction** The parameters of a reverse-biased $pn$ junction with uniform doping of silicon semiconductors are $V_T = 26$ mV, $V_R = 10$ V, $T = 25^\circ$C, $N_a = 1 \times 10^{16}$ cm$^{-3}$, and $N_d = 2 \times 10^{15}$ cm$^{-3}$. Use the parameters of Example 6.2.

Calculate the junction capacitance if the cross-sectional area of the $pn$ junction is $A_{pn} = 10^{-3}$ cm$^2$.

**SOLUTION**

$V_T = 26$ mV, $V_R = 10$ V, $T = 25^\circ$C, $N_a = 1 \times 10^{16}$ cm$^{-3}$, $N_d = 2 \times 10^{15}$ cm$^{-3}$, and $A_{pn} = 10^{-3}$ cm$^2$.

From Eq. (6.51),

$$C_x = \frac{1.6 \times 10^{-19} \times 1.035 \times 10^{-12} \times 1 \times 10^{16} \text{cm}^{-3} \times 2 \times 10^{15} \text{cm}^{-3}}{(0.648 + 10) \times (1 \times 10^{16} \text{cm}^{-3} + 2 \times 10^{15} \text{cm}^{-3})} = 3.601 \text{nF/cm}^2$$

Therefore, $C_j = C_x A_{pn} = 3.601 \text{nF} \times 10^{-3} \text{cm}^2 = 3.601 \text{pF}$.

**KEY POINTS OF SECTION 6.4**

- If a diode is reverse biased, the potential barrier is increased. The holes from the $p$-side and the electrons from the $n$-side cannot cross the junction. That is, the ohmic resistance of the diode becomes very high. A sufficiently high reverse voltage, however, may cause an avalanche breakdown.
- The width of the space charge region increases with the reverse voltage and decreases the junction capacitance.
6.5 Forward-Biased *pn* Junction

A *pn* junction is said to be forward biased if the *p*-side is made positive with respect to the *n*-side, as depicted in Fig. 6.12(a). If the forward voltage $V_D = V_F$ is increased, the potential barrier is reduced to $V_{bi} - V_F$, as shown in Fig. 6.12(b), and a large number of holes flow from the *p*-side to the *n*-side. Similarly, a large number of electrons flow from the *n*-side to the *p*-side. The resultant diode current becomes $I_D = I_{DF} - I_{DR}$. As the diode current $I_D$ increases, the ohmic resistances of the *p*-side and the *n*-side cause a significant series voltage drop. If $V_D$ is increased further, most of the increase in $I_D$ will be lost as a series voltage drop. Thus, the width of the depletion region is reduced with the increase in the forward voltage. The potential barrier will not be reduced proportionally, but it can become zero.

In this case, the barrier height between the two regions is reduced. Figure 6.12(c) shows the energy band diagram of the *pn* junction. $E_c$ and $E_v$ are shifted by the total voltage $V_{PB} = V_{bi} - V_F$. As $V_F$ pushes the energy levels, the Fermi level on the *n*-side ($E_{Fn}$) is now above the Fermi level on the *p*-side ($E_{Fp}$). The difference between $E_{Fp}$ and $E_{Fn}$ is equal to $eV_F$. That is, $E_{Fn} - E_{Fp} = eV_F$. 

![Diagram of Forward-biased pn Junction](image-url)
6.5.1 Depletion Region Width

With the forward-biased voltage \( V_F = V_T \), the total potential barrier will decrease from \( V_{bi} \) to \( V_{bi} - V_F \). Thus, Eq. (6.18) can be modified to obtain the total effective potential as

\[
V_{PB} = |\phi_{Fp}| + |\phi_{Fn}| - V_F = V_{bi} - V_F \tag{6.53}
\]

Substituting for \( V_{bi} \) with \( V_{bi} - V_F \) in Eqs. (6.39) through (6.41), we can obtain the space charge extension in the \( n \)- and \( p \)-regions as

\[
x_n = \sqrt{\frac{2\varepsilon_s(V_{bi} - V_F)}{e} \left(\frac{N_a}{N_d}\right) \left(\frac{1}{N_a + N_d}\right)} \tag{6.54}
\]

\[
x_p = \sqrt{\frac{2\varepsilon_s(V_{bi} - V_F)}{e} \left(\frac{N_d}{N_a}\right) \left(\frac{1}{N_a + N_d}\right)} \tag{6.55}
\]

Therefore, the total width \( W \) of the depletion or space charge region is the sum of \( x_n \) and \( x_p \):

\[
W = x_n + x_p = \sqrt{\frac{2\varepsilon_s(V_{bi} - V_F)}{e} \left(\frac{N_a + N_d}{N_a N_d}\right)} \tag{6.56}
\]

Thus, the depletion width \( W \) decreases with an increasing forward biased voltage \( V_F \). Since \( x_n \) in Eq. (6.54) and \( x_p \) in Eq. (6.55) decrease with forward bias voltage \( V_F \), the magnitudes of the electric fields in Eq. (6.28) and Eq. (6.30) also decrease. We can find the maximum field from Eq. (6.43) as a function of \( V_F \) and \( W \):

\[
\varepsilon_{\text{max}} = \frac{-2 (V_{bi} - V_F)}{W} \tag{6.57}
\]

Thus, the maximum field decreases with \( V_F \) and \( W \).

**EXAMPLE 6.4**

**Finding the depletion width in a forward-biased pn junction**  The parameters of a uniformly doped \( pn \) junction for silicon semiconductors are \( V_F = 0.60 \, \text{V}, V_T = 26 \, \text{mV}, T = 25^\circ \text{C}, N_a = 1 \times 10^{16} \, \text{cm}^{-3} \), and \( N_d = 2 \times 10^{15} \, \text{cm}^{-3} \).

Find (a) the depletion width \( W \) and (b) the maximum field \( \varepsilon_{\text{max}} \).

**SOLUTION**

\( V_F = 0.65 \, \text{V}, T = 25^\circ \text{C}, N_a = 1 \times 10^{16} \, \text{cm}^{-3}, \) and \( N_d = 2 \times 10^{15} \, \text{cm}^{-3} \).

\[
\varepsilon_r = 11.7 \quad \varepsilon_o = 8.85 \times 10^{-14} \quad e = 1.6 \times 10^{-19} \quad k = 1.3806 \times 10^{-23}
\]

\[
T_k = 273 + T = 273 + 25 = 298 \, \text{K} \quad \varepsilon_s = \varepsilon_r \times \varepsilon_o = 11.7 \times 8.85 \times 10^{-14} = 1.035 \times 10^{-12}
\]
6.5.2 Minority Carrier Charge Distribution

Since a large number of electrons diffuse from the n-side to the p-side and become minority carriers in the p-region, let us define that \( n_{no} = N_d \) is the concentration of majority carrier electrons in the n-region once thermal equilibrium is reached. Since the product of \( n_{no} \) and \( n_{po} \) is constant according to Eq. (6.16), we can find the concentration of minority carrier electrons in the p-region as

\[
(6.58)\]

Equation (6.24) gives the built-in potential as

\[
V_{bi} = kT \ln \left( \frac{N_d N_a}{n_i^2} \right)
\]

which, after we substitute \( N_a \) from Eq. (6.58) and \( n_{no} = N_d \), gives

\[
V_{bi} = kT \ln \left( \frac{N_d N_a}{n_i^2} \right) = kT \ln \left( \frac{n_{no}}{n_{po}} \right) (6.59)
\]

Taking a natural log on both sides of Eq. (6.59), we can find the minority carrier electron concentration on the p-side, \( n_{po} \), as a function of the majority carrier electron concentration on the n-side, \( n_{no} \):

\[
n_{po} = n_{no} e^{-V_{bi}/kT} (6.60)
\]

For a forward-biased junction, we can substitute for \( V_{bi} \) in Eq. (6.60) with the effective voltage \( V_{bi} - V_F \) to find the minority carrier electron concentration on the p-side, \( n_p \):

\[
n_p = n_{no} e^{-(V_{bi} - V_F)/kT} = n_{no} e^{-V_{bi}/kT} \times e^{V_F/kT} (6.61)
\]
This can be expressed as a function of $n_{po}$ as

$$n_p = n_{po} \times e^{V_F/KT} \quad (6.62)$$

Therefore, $n_p$ is greater than $n_{po}$ on the $p$-side, and a forward-biased $pn$ junction is no longer in thermal equilibrium. Similarly, we can find the minority carrier hole concentration on the $n$-side, $p_n$:

$$p_n = p_{no} \times e^{V_F/V_T} \quad (6.63)$$

This also shows that $p_n$ is greater than $p_{no}$ on the $n$-side. Therefore, a forward-biased $pn$ junction will create excess minority carriers at each edge of the space charge region of the $pn$ junction, as shown in Fig. 6.13. Due to the exponential relationship, a relatively small forward-biased voltage can cause a significant increase in the minority carrier concentration. It is important to note that $n_p$ and $p_n$ decay exponentially with distance away from the junction to their thermal equilibrium values $n_{po}$ and $p_{no}$, as also shown in Fig. 6.13.

**EXAMPLE 6.5**

**Finding the minority carrier concentration of a forward-biased junction** The parameters of a uniformly doped $pn$ junction for silicon semiconductors are $V_F = 0.60$ V, $V_T = 26$ mV, $T = 25^\circ$C, $N_a = 2 \times 10^{15}$ cm$^{-3}$, and $N_d = 1 \times 10^{16}$ cm$^{-3}$.

Find the minority carrier concentrations at the edge of the depletion region: (a) electrons in the $p$-side, $n_p$, and (b) holes in the $n$-side, $p_n$.

**SOLUTION**

$$V_F = 0.60 \text{ V, } V_T = 26 \text{ mV, } T = 25^\circ \text{C, } N_a = 2 \times 10^{15} \text{ cm}^{-3}, \text{ and } N_d = 1 \times 10^{16} \text{ cm}^{-3}.$$  

$$\varepsilon = 11.7, \quad \varepsilon_0 = 8.85 \times 10^{-14}, \quad e = 1.6 \times 10^{-19}, \quad k = 1.3806 \times 10^{-23}.$$  

$$T_k = 273^\circ + 273^\circ + 25^\circ = 298 \text{ K, } \quad \varepsilon_4 = \varepsilon_1 \times \varepsilon_0 = 11.7 \times 8.85 \times 10^{-14} = 1.035 \times 10^{-12}$$
KEY POINTS OF SECTION 6.5

- If a diode is forward biased, the potential barrier is reduced and a large number of holes will flow from the p-side to the n-side. Similarly, a large number of electrons will flow from the n-side to the p-side. That is, the ohmic resistance of the diode becomes very small under forward-biased conditions.
- The width of the space charge region decreases with the forward voltage.
- A large number of electrons or holes diffuse from one side to the other, and they become minority carriers on the other side. This creates excess minority carriers at each edge of the space charge region of the pn junction.

6.6 Junction Current Density

It can be shown that the electron current density due to the charge flow from the n-region to the p-region at the edge of the p-region at \( x = -x_p \) is given by [2–4]

\[
J_n(x = -x_p) = \frac{eD_p n_{po}}{L_n} \left( e^{V_F/V_T} - 1 \right)
\]

where \( D_n \) is the minority electron diffusion density, \( L_n \) is the minority electron diffusion length and is related to \( D_n \) and the minority carrier life \( \tau_{no} \) by \( L_n^2 = D_n \tau_{no} \). The minority carrier life is defined as the average time for a minority electron in the p-region to recombine with a majority hole in the p-region. Similarly, the hole current density due to the charge flow from the p-region to the n-region at the edge of the n-region at \( x = x_n \) is given by

\[
J_p(x = x_n) = \frac{eD_p p_{no}}{L_p} \left( e^{V_F/V_T} - 1 \right)
\]
where $D_p$ is the minority hole diffusion density, $L_p$ is the minority hole diffusion length and is related to $D_p$ and the minority hole carrier life $\tau_{po}$ by $L_p^2 = D_p \tau_{po}$. Therefore, the total current density in the $pn$ junction is given by

$$J = J_n(x = -x_n) + J_p(x = x_n) = \left(\frac{eD_n n_{po}}{L_n} + \frac{eD_p p_{no}}{L_p}\right) \left(e^{V/V_T} - 1\right) \quad (6.66)$$

This can be written in a more general form as

$$J = J_s(e^{V/V_T} - 1) \quad (6.67)$$

where the parameter $J_s$ is known as the reverse saturation current density and depends on the physical parameters of the $pn$ junction as given by

$$J_s = \frac{eD_n n_{po}}{L_n} + \frac{eD_p p_{no}}{L_p} \quad (6.68)$$

Equation (6.67) for the junction current is applicable for both positive values (forward-biased condition) and negative values (reverse-biased condition) such that $V_D = -V_F$. The diode current can be obtained by multiplying the current density in Eq. (6.67) by the cross-sectional area of the $pn$ junction $A_{pn}$ and be expressed in the general form describing the Schottkey equation in Eq. (4.1) as

$$i_D = J \times A_{pn} = I_s(e^{\eta V/V_T} - 1) \quad (6.69)$$

Here the parameter $I_s$ is known as the reverse saturation current, $V_D$ is the applied voltage, and $\eta$ is a constant whole value that varies from 1 to 2 depending on the manufacturing process of practical diodes. Equation (6.69) describes the characteristics of the Schottkey barrier junction.

**NOTE** $V_D > 0$ for forward-biased conditions and $V_D < 0$ for reverse-biased conditions.

**EXAMPLE 6.6**

**Finding the reverse saturation current** The parameters of a reverse-biased $pn$ junction with uniform doping of silicon semiconductors are $N_a = 1 \times 10^{16}$ cm$^{-3}$, $N_d = 1 \times 10^{16}$ cm$^{-3}$, $n_i = 1.5 \times 10^{10}$ cm$^{-3}$, $D_a = 20$ cm$^2$/s, $D_d = 10$ cm$^2$/s, $\tau_{po} = \tau_{no} = 8 \times 10^{-6}$ s, and $A_{pn} = 10^{-3}$ cm$^2$.

Find the reverse saturation current $I_s$. 

**SOLUTION**

$N_a = 1 \times 10^{16}$ cm$^{-3}$, $N_d = 1 \times 10^{16}$ cm$^{-3}$, $n_i = 1.5 \times 10^{10}$ cm$^{-3}$, $D_a = 20$ cm$^2$/s, $D_d = 10$ cm$^2$/s, $\tau_{po} = \tau_{no} = 8 \times 10^{-6}$ s, and $A_{pn} = 10^{-3}$ cm$^2$.

$\epsilon_r = 11.7 \quad \epsilon_0 = 8.85 \times 10^{-14} \quad e = 1.6 \times 10^{-19} \quad k = 1.3806 \times 10^{-23}$
6.7 Temperature Dependence

According to Eq. (6.67), the current density \( J \) is a direct function of the reverse saturation current density \( J_s \), which depends on the minority carrier concentrations \( n_{po} \) and \( p_{no} \)—which in turn are also proportional to \( n_1^2 \), which is a function of temperature. Therefore, we have

\[
J_s = \frac{eD_n n_{po}}{L_n} + \frac{eD_p p_{no}}{L_p}
\]

\[
= \frac{1.6 \times 10^{-19} \times 20 \times 2.25 \times 10^4 \text{ cm}^{-3}}{0.013 \text{ m}} + \frac{1.6 \times 10^{-19} \times 10 \times 2.25 \times 10^4 \text{ cm}^{-3}}{8.944 \times 10^{-3} \text{ cm}}
\]

\[
= 9.717 \times 10^{-12} \text{ A/cm}^2
\]

Therefore, the reverse saturation current is \( I_s = J_s A_{pn} = 9.717 \times 10^{-12} \text{ A/cm}^2 \times 10^{-3} \text{ cm}^2 = 9.717 \times 10^{-15} \text{ A} \).

6.7 Temperature Dependence

According to Eq. (6.67), the current density \( J \) is a direct function of the reverse saturation current density \( J_s \), which depends on the minority carrier concentrations \( n_{po} \) and \( p_{no} \)—which in turn are also proportional to \( n_1^2 \), which is a function of temperature. Therefore, we have

\[
J_s \propto n_1^2 \propto T^3 e^{-E_g/kT}
\]  \( \text{(6.70)} \)

where \( E_g \) is the electron energy, 1.12 eV. We can relate \( J_{s2} \) and \( J_{s1} \) corresponding to temperatures \( T_2 \) and \( T_1 \) by

\[
\frac{J_{s2}}{J_{s1}} = \frac{T_2^3 e^{-E_g/kT_2}}{T_1^3 e^{-E_g/kT_1}} = \left( \frac{T_2}{T_1} \right)^3 \left( \frac{e^{-E_g/kT_2}}{e^{-E_g/kT_1}} \right)
\]  \( \text{(6.71)} \)

Therefore, the reverse saturation current density \( J_s \) is sensitive to the temperature and increases rapidly with the temperature, as shown in Fig. 6.14(a) for a reverse-biased condition. The forward current \( J \), which is a function of \( J_s \) and \( (eV_F/kT) \), is a function of temperature. As the temperature increases, the voltage drop decreases for the same amount of forward current, as shown in Fig. 6.14(b). For a constant forward voltage, the forward current increases with the temperature.
In Sec. 4.7 we considered the static behavior of a \( pn \) junction diode. A practical diode, however, exhibits some capacitive effects that need to be incorporated into any high-frequency model in order to get the time-dependent response of a diode circuit. We have seen that a depletion layer exists in the reverse-biased \( pn \) junction of diodes. That is, there is a region depleted of carriers, separating two regions of relatively good conductivity. Thus we have in essence a parallel-plate capacitor, with silicon as the dielectric. Also, there is an injection of a large number of minority carriers under forward-biased conditions. Therefore, there are two types of capacitances: depletion and diffusion.

### 6.8.1 Depletion Capacitance

A positively charged layer is separated from a negatively charged layer by a very small but finite distance. As the voltage across the \( pn \) junction changes, the charge stored in the depletion layer changes accordingly. This is shown in Fig. 6.15 for a nonlinear \( q-v \) relationship. The depletion capacitance relates the change in the charge \( \Delta q \) in the depleted region to the change in the bias voltage \( \Delta v_D \), and it is given by

\[
C_j = \frac{dq_j}{dv_D} \bigg|_{\text{at estimated } Q\text{-point } v_D = -V_D^*}
\]

This is derived in Eq. (6.51), and it can be expressed in general form as

\[
C_j = \frac{C_{jo}}{(1 - v_D/V_j^{\text{m}})}
\]
where \( m \) = junction gradient coefficient, whose value is in the range of 0.33 to 0.5
\[ V_D = \text{anode-to-cathode bias voltage, which will be positive in the forward direction and negative in the reverse direction} \]
\[ V_j = \text{potential barrier with zero external voltage applied to the diode and is known as the built-in potential} \] (It is a function of the type of semiconductor material, the degree of doping, and the junction temperature. For a silicon diode \( V_j \approx 0.5 \text{ V to 0.9 V} \), and for a germanium diode \( V_j \approx 0.2 \text{ V to 0.6 V} \)).
\[ C_{jo} = \text{depletion capacitance when the external voltage across the diode is zero} \]

The depletion capacitance is also known as the transition capacitance. The value of \( C_j \) is directly proportional to the cross-section of the diode junction and is in the range of 0.1 pF to 100 pF. Notice from Eq. (6.72) that the depletion capacitance \( C_j \) can be varied by changing the reverse voltage \( v_R = -v_D \) across the diode. The capability to change a capacitance by varying a voltage can be exploited in some applications. Diodes designed for such applications are called varactors or varicaps, depending on the applications. This depletion capacitance may be used for tuning FM radios, television circuits, microwave oscillators, and any other circuits in which a small variation in capacitance can effect a significant change in frequency. In these applications, a reverse-biased diode can be connected in parallel with an external capacitor of a parallel circuit consisting of resistor (R), indicator (L), and capacitor (C) circuit so that the resonant frequency \( f_p \) is given by
\[ f_p = \frac{1}{2\pi \sqrt{L(C + C_j)}} \] (6.74)

where \( C_j \) = depletion capacitance varied by the reverse-biased voltage \((v_D)\) of the diode (Typical values of \( C_j \) are 10 pF to 100 pF at reverse voltages of 3 V to 25 V).
\[ L = \text{inductance of the parallel RLC circuit} \]
\[ C = \text{capacitance of the parallel RLC circuit} \]

### 6.8.2 Diffusion Capacitance

When the junction is forward biased, the depletion region becomes narrower and the depletion capacitance increases because the bias voltage \( v_D \) is positive. However, a large number of minority carriers are injected into the junction under the forward-biased condition. There will be an excess of minority charge carriers
near the depletion layer, and this will cause a great charge storage effect. The excess concentration will be highest near the edge of the depletion layer and will decrease exponentially toward zero with the distance from the junction. This is shown in Fig. 6.13, where \( p_n \) is the hole concentration in the \( n \)-region and \( n_p \) is the electron concentration in the \( p \)-region. If the voltage applied to the diode is changed, the minority carrier charges stored in the \( p \)- and \( n \)-regions will also change and reach a new steady-state condition. Therefore, a forward-biased \( pn \) junction will exhibit a capacitive effect as a result of the shortage of minority carrier charges. Since these charges will be proportional to the diode current, the current density \( J_p \) in Eq. (6.65) can be applied to relate the charge \( q_m \) to the forward voltage \( V_D \), given by

\[
q_m = q_o(e^{\eta V_T} - 1) \tag{6.75}
\]

where \( q_o \) is the constant charge proportional to the leakage (or reverse saturation) current density \( J_s \). Therefore, the \( q-V \) characteristic of a forward-biased diode will be nonlinear, and it can be modeled by a small-signal capacitance \( C_d \) known as the diffusion capacitance. That is,

\[
C_d = \left. \frac{dq_m}{dV_D} \right|_{\text{at estimated } Q\text{-point } V_D=V_D} \tag{6.76}
\]

which indicates that \( C_d \) is proportional to the value of \( q_m + q_o \). In the reverse-biased condition, \( C_d = 0 \). In the forward direction, however, the value of \( C_d \) is approximately proportional to the DC bias current \( I_D \) (at the \( Q \)-point). That is, \( C_d \) is given by

\[
C_d = K_d I_D \tag{6.77}
\]

where \( K_d \) is a constant and \( C_d \) is directly proportional to the cross-section of the diode junction and is typically in the range of 10 pF to 100 pF.

### 6.8.3 Forward-Biased Model

A forward-biased diode will exhibit two capacitances: diffusion capacitance \( C_d \) and depletion-layer capacitance \( C_j \), expressed by Eq. (6.52) for \( V_D \geq 0 \). These capacitances will affect the high-frequency applications of diodes. For the small-signal high-frequency model of a forward-biased diode, as shown in Fig. 6.16(a), the model parameters are given by

\[
\frac{1}{r_d} = \left. \frac{dI_D}{dV_D} \right|_{\text{at estimated } Q\text{-point } I_D=I_D} = \frac{\eta V_T}{\eta V_T} \tag{6.78}
\]

\[
C_j = \frac{C_{jo}}{(1 - V_D/V_j)^m} \quad \text{for } V_D \geq 0 \tag{6.79}
\]

\[
C_d = K_d I_D
\]

For example, if \( C_{jo} = 4 \) pF, \( V_j = 0.75 \) V, \( m = 0.333 \), and \( V_D = 0.7158 \) V, then \( C_j = 11.18 \) pF.
The high-frequency AC model represents the frequency response of the diode by including two junction (diffusion and depletion-layer) capacitances to the low-frequency AC model. The depletion-layer capacitance is dependent on the diode voltage. But the diffusion capacitance is directly proportional to the diode current and is present only in the forward direction.

### 6.8.4 Reverse-Biased Model

The small-signal AC resistance $r_d$ in the reverse direction is very high, on the order of several megohms, and may be assumed to be very large, tending to infinity. The diffusion capacitance $C_d$, which depends on the diode current, is negligible in the reverse direction because the reverse current is very small. For the high-frequency AC model of a reverse-biased diode having $r_r$ as the resistance in the reverse direction, as shown in Fig. 6.16(b), the model parameters are given by

\[
\begin{align*}
    r_r &= \infty \\
    C_j &= \frac{C_{jo}}{(1 - v_D/V_j)^m} \quad \text{for } v_D \leq 0 \\
    C_d &= 0
\end{align*}
\]  

For example, if $C_{jo} = 4 \ \text{pF}$, $V_j = 0.75 \ \text{V}$, $m = 0.333$, and $V_D = -20 \ \text{V}$, then $C_j = 1.32 \ \text{pF}$.

\[C_j = \frac{C_{jo}}{(1 - v_D/V_j)^m}\]

### KEY POINT OF SECTION 6.8

- The high-frequency AC model represents the frequency response of the diode by including two junction (diffusion and depletion-layer) capacitances to the low-frequency AC model. The depletion-layer capacitance is dependent on the diode voltage. But the diffusion capacitance is directly proportional to the diode current and is present only in the forward direction.

### Summary

A pn junction is formed by sandwiching a p-type material into one side and an n-type material in the other side of a single crystal. Free electrons (in n-type material) and holes (in p-type material) are made available by adding a controlled amount of n-type impurities and p-type impurities to pure semiconductors, respectively. The Fermi function specifies the probability that an available state will be occupied by an electron. The intrinsic concentration of a semiconductor material remains constant at a constant temperature.
A depletion or space charge region exists at the \textit{pn} junction, whose width depends on the doping concentrations and the external applied voltage. The junction capacitance depends on the reverse voltage. In the forward-biased condition, a large number of electrons or holes diffuse from one side to the other side and become minority carriers on the other side. This creates excess minority carriers at each edge of the space charge region of the \textit{pn} junction. The depletion-layer capacitance is dependent on the forward voltage. But the diffusion capacitance is directly proportional to the forward current and is present only in the forward direction. The high-frequency AC model represents the frequency response of the diode by including two junction (diffusion and depletion-layer) capacitances to the low-frequency AC model.

\textbf{References}


\textbf{Review Questions}

1. What is a donor impurity? What is an acceptor impurity?
2. What is doping?
3. What is the depletion region of a \textit{pn} junction?
4. What are the minority carriers in \textit{p}-type materials?
5. What are the majority carriers in \textit{p}-type materials?
6. What are the minority carriers in \textit{n}-type materials?
7. What are the majority carriers in \textit{n}-type materials?
8. What is the effect of junction temperature on the diode characteristic?
9. What is the Fermi function?
10. What is the effect of temperature on the Fermi function?
11. What is the intrinsic electron or hole concentration in a semiconductor material?
12. What are the effects of reverse voltage on a \textit{pn} junction?
13. What is the depletion region?
14. What is a built-in potential?
15. What is the effect of reverse voltage on the depletion region?
16. What is a breakdown condition of a \textit{pn} junction?
17. What causes the junction capacitance of a \textit{pn} junction?
18. What are the effects of forward voltage on a \textit{pn} junction?
19. What is the minority carrier life?
20. Why is the reverse saturation current density sensitive to temperature?
21. What is the high-frequency AC model of a diode?
22. What is the depletion capacitance of a \textit{pn} junction?
23. What is the diffusion capacitance of a \textit{pn} junction?
### Problems

#### 6.2 Semiconductor Materials

6.1 Calculate the intrinsic carrier concentration \( n_i \) at \( T = 200 \text{ K}, 400 \text{ K}, \) and \( 600 \text{ K} \) for (a) silicon, (b) germanium, and (c) gallium arsenide.

6.2 Two silicon semiconductor materials have the same properties but different gap band energies: \( E_{gA} = 1.12 \text{ eV} \) and \( E_{gB} = 1.25 \text{ eV} \). Determine their intrinsic concentrations \( n_{iA} \) and \( n_{iB} \) and the ratio \( n_{iB}/n_{iA} \).

6.3 If the maximum intrinsic concentration of silicon is to be limited to \( n_i = 1.5 \times 10^{18} \text{ cm}^{-3} \), what will be the maximum permissible temperature if the gap band energy is \( E_g = 1.15 \text{ eV} \)?

6.4 If the maximum intrinsic concentration of gallium arsenide is to be limited to \( n_i = 1.5 \times 10^{18} \text{ cm}^{-3} \), what will be the maximum permissible temperature if the gap band energy is \( E_g = 1.15 \text{ eV} \)?

6.5 Calculate the equilibrium electron concentration \( n_o \) of a silicon material if the gap band energy at \( T = 350 \text{ K} \) is (a) \( E_g = 0.75 \text{ eV} \), (b) \( E_g = 1.12 \text{ eV} \), and (c) \( E_g = 1.25 \text{ eV} \).

6.6 Calculate the equilibrium electron concentration \( n_o \) of a gallium arsenide material if the gap band energy at \( T = 350 \text{ K} \) is (a) \( E_g = 0.75 \text{ eV} \), (b) \( E_g = 1.12 \text{ eV} \), and (c) \( E_g = 1.25 \text{ eV} \).

6.7 Calculate the equilibrium hole concentration \( p_0 \) of a silicon material if the gap band energy at \( T = 350 \text{ K} \) is (a) \( E_g = 0.75 \text{ eV} \), (b) \( E_g = 1.12 \text{ eV} \), and (c) \( E_g = 1.25 \text{ eV} \).

6.8 Calculate the equilibrium hole concentration \( p_0 \) of a gallium arsenide material if the gap band energy at \( T = 350 \text{ K} \) is (a) \( E_g = 0.75 \text{ eV} \), (b) \( E_g = 1.12 \text{ eV} \), and (c) \( E_g = 1.25 \text{ eV} \).

6.9 The value of equilibrium electron concentration for a silicon material is \( n_o = 1.5 \times 10^{17} \text{ cm}^{-3} \) at \( T = 30^\circ \text{C} \). Determine the gap band energy \( E_g \).

6.10 The value of equilibrium hole concentration for a silicon material is \( n_o = 1.5 \times 10^{16} \text{ cm}^{-3} \) at \( T = 30^\circ \text{C} \). Determine the gap band energy \( E_g \).

6.11 Determine the intrinsic Fermi energy \( E_{Fi} \) for silicon if \( E_g = 1.1 \text{ eV} \) and \( T = 25^\circ \text{C} \).

6.12 If the Fermi energy is 0.25 eV below the conduction band energy \( E_c \) and \( N_c = 1.5 \times 10^{19} \text{ cm}^{-3} \) at \( T = 25^\circ \text{C} \), (a) calculate the probability that an energy state in the conduction band at \( (E_c + kT) \) is filled by an electron, and (b) calculate the thermal equilibrium electron concentration in silicon.

6.13 If the Fermi energy is 0.25 eV below the valence band energy \( E_v \) and \( N_v = 1.5 \times 10^{19} \text{ cm}^{-3} \) at \( T = 25^\circ \text{C} \), (a) calculate the probability that an energy state in the valence band at \( (E_v - kT) \) is empty of an electron, and (b) calculate the thermal equilibrium hole concentration in silicon.

#### 6.3 Zero-Biased pn Junction

6.14 The parameters of a uniformly doped \( pn \) junction for silicon semiconductors are \( V_T = 26 \text{ mV}, T = 25^\circ \text{C}, N_a = 1 \times 10^{16} \text{ cm}^{-3} \), and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \). Find (a) the depletion width \( W \) and (b) the maximum field \( E_{\text{max}} \).

6.15 The parameters of a uniformly doped \( pn \) junction for silicon semiconductors are \( V_T = 26 \text{ mV}, T = 25^\circ \text{C}, N_a = 1 \times 10^{12} \text{ cm}^{-3} \), and \( N_d = 10^{16} \text{ cm}^{-3} \). Find (a) \( V_{bi} \), (b) \( x_n \) and \( x_p \), (c) the depletion width \( W \), and (d) the maximum field \( E_{\text{max}} \). Plot the electric field against the distance \( x \) through the junction.

6.16 Calculate the built-in potential \( V_{bi} \) of a uniformly doped \( pn \) junction for silicon semiconductors if \( V_T = 26 \text{ mV}, T = 25^\circ \text{C}, N_a = 1 \times 10^{16} \text{ cm}^{-3} \), and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \).
6.17 Plot the built-in potential \( V_{bi} \) against \( N_d \) for a uniformly doped \( pn \) junction for silicon semiconductors if \( V_T = 26 \text{ mV}, T = 25^\circ \text{C}, N_a = 1 \times 10^{18} \text{ cm}^{-3}, \) and \( 1 \times 10^{14} \leq N_d \leq 1 \times 10^{19} \text{ cm}^{-3} \).

6.18 Plot the built-in potential \( V_{bi} \) against \( N_d \) for a symmetrical silicon \( pn \) junction if \( V_T = 26 \text{ mV}, T = 25^\circ \text{C}, \) and \( 1 \times 10^{14} \leq N_a = N_d \leq 1 \times 10^{19} \text{ cm}^{-3} \).

6.19 Plot the built-in potential \( V_{bi} \) against temperature for a uniformly doped \( pn \) junction for silicon semiconductors if \( V_T = 26 \text{ mV}, 25^\circ \text{C} \leq T \leq 250^\circ \text{C}, N_a = 1 \times 10^{18} \text{ cm}^{-3}, \) and \( N_d = 1 \times 10^{19} \text{ cm}^{-3} \).

6.20 The parameters of a uniformly doped silicon \( pn \) junction are \( V_T = 26 \text{ mV}, N_a = 1 \times 10^{17} \text{ cm}^{-3}, \) and \( N_d = 5 \times 10^{15} \text{ cm}^{-3} \). Determine the temperature if the built-in potential barrier is \( V_{bi} = 0.56 \text{ V} \).

6.21 The parameters of a uniformly doped silicon \( pn \) junction are \( T = 25^\circ \text{C}, N_a = 1 \times 10^{16} \text{ cm}^{-3}, \) and \( N_d = 5 \times 10^{15} \text{ cm}^{-3} \). If the temperature changes by 15\%, what will be the change in the built-in potential barrier \( V_{bi} \)?

6.22 The parameters of a uniformly doped silicon \( pn \) junction are \( T = 25^\circ \text{C}, N_a = 1 \times 10^{16} \text{ cm}^{-3}, \) and \( N_d = 5 \times 10^{15} \text{ cm}^{-3} \). If \( T, N_a, \) and \( N_d \) change by \( \pm 15\% \), what will be the minimum and maximum values of the built-in potential barrier \( V_{bi} \)?

6.4 Reverse-Biased \( pn \) Junction

6.23 The parameters of a uniformly doped \( pn \) junction for silicon semiconductors are \( V_R = 10 \text{ V}, V_T = 26 \text{ mV}, T = 25^\circ \text{C}, N_a = 1 \times 10^{16} \text{ cm}^{-3}, \) and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \). Find (a) the depletion width \( W \) and (b) the maximum field \( \varepsilon_{max} \).

6.24 The parameters of a reverse-biased \( pn \) junction with uniform doping of silicon semiconductors are \( V_T = 26 \text{ mV}, V_R = 15 \text{ V}, T = 25^\circ \text{C}, N_a = 1 \times 10^{16} \text{ cm}^{-3}, \) and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \). Calculate the junction capacitance if the cross-sectional area of the \( pn \) junction is \( A_{pn} = 10^{-3} \text{ cm}^2 \).

6.25 The parameters of a reverse-biased \( pn \) junction with uniform doping of silicon semiconductors are \( V_T = 26 \text{ mV}, V_R = 15 \text{ V}, T = 25^\circ \text{C}, N_a = 1 \times 10^{16} \text{ cm}^{-3}, \) and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \). Calculate the junction capacitance if the cross-sectional area of the \( pn \) junction is \( A_{pn} = 2 \times 10^{-3} \text{ cm}^2 \).

6.26 The parameters of a reverse-biased abrupt silicon \( pn \) junction are \( V_R = 12 \text{ V}, T = 25^\circ \text{C}, N_a = 10^{16} \text{ cm}^{-3}, \) and \( N_d = 10^{15} \text{ cm}^{-3} \). Calculate (a) \( V_{bi} \), (b) \( W \), (c) the maximum field \( \varepsilon_{max} \), and (d) the junction capacitance \( C_j \) if \( A_{pn} = 10^{-3} \text{ cm}^2 \).

6.27 The parameters of a reverse-biased abrupt silicon \( pn \) junction are \( V_R = 12 \text{ V}, T = 25^\circ \text{C}, N_a = 10^{16} \text{ cm}^{-3}, \) and \( N_d = 100 N_a \). Calculate (a) \( V_{bi} \), (b) \( W \), (c) the maximum field \( \varepsilon_{max} \), and (d) the junction capacitance \( C_j \) if \( A_{pn} = 2 \times 10^{-3} \text{ cm}^2 \).

6.28 The parameters of a reverse-biased abrupt silicon \( pn \) junction are \( V_R = 12 \text{ V}, T = 25^\circ \text{C}, N_a = 10^{16} \text{ cm}^{-3}, \) and \( N_d = 100 N_a \). Calculate (a) \( V_{bi} \), (b) \( W \), (c) the maximum field \( \varepsilon_{max} \), and (d) the junction capacitance \( C_j \). If \( V_R \) changes by \( \pm 20\% \), what will be the minimum and maximum values of the junction capacitance \( C_j \) if \( A_{pn} = 10^{-3} \text{ cm}^2 \)?

6.29 The parameters of a reverse-biased abrupt silicon \( pn \) junction are \( V_R = 12 \text{ V}, T = 25^\circ \text{C}, N_a = 10^{16} \text{ cm}^{-3}, \) and \( N_d = 100 N_a \). Calculate (a) \( V_{bi} \), (b) \( W \), (c) the maximum field \( \varepsilon_{max} \), and (d) the junction capacitance \( C_j \) if \( A_{pn} = 10^{-3} \text{ cm}^2 \). If \( N_a \) changes by \( \pm 15\% \), what will be the minimum and maximum values of the junction capacitance \( C_j \)?

6.30 The parameters of a reverse-biased abrupt silicon \( pn \) junction are \( V_R = 12 \text{ V}, T = 25^\circ \text{C}, N_a = 10^{16} \text{ cm}^{-3}, \) and \( N_d = 2 \times 10^{18} \text{ cm}^{-3} \). Calculate (a) \( V_{bi} \), (b) \( W \), (c) the maximum field \( \varepsilon_{max} \), and (d) the junction capacitance \( C_j \) if \( A_{pn} = 10^{-3} \text{ cm}^2 \). If \( N_a \) and \( N_d \) change by \( \pm 15\% \), what will be the minimum and maximum values of the maximum field \( \varepsilon_{max} \)?
6.31 The parameters of a reverse-biased pn junction with uniform doping of silicon semiconductors are \( V_T = 26 \text{ mV} \), \( V_R = 10 \text{ V} \), \( T = 25^\circ \text{C} \), \( N_a = 1 \times 10^{16} \text{ cm}^{-3} \), and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \). Calculate the reverse voltage \( V_R \) that will give a junction capacitance of \( C_j = 5 \text{ pF} \) if \( A_{pn} = 10^{-3} \text{ cm}^2 \).

6.32 A uniformly doped silicon pn junction operating at \( T = 25^\circ \text{C} \) is to be designed such that at a reverse-biased voltage of \( V_R = 12 \text{ V} \), the maximum field is limited to \( \varepsilon_{\text{max}} = 5 \times 10^5 \text{ V/cm} \). Determine the maximum doping concentration in the \( n \)-region.

6.33 A uniformly doped silicon pn junction operating at \( T = 25^\circ \text{C} \) has a reverse-biased voltage of \( V_R = 12 \text{ V} \). The charge in the \( n \)-region is to be limited to 15% of the total space charge, and the total junction capacitance is \( C_j = 4 \text{ pF} \) if \( A_{pn} = 6 \times 10^{-4} \text{ cm}^2 \). Determine (a) \( N_a \), (b) \( N_d \), and (c) \( V_{bi} \).

6.5 Forward-Biased pn Junction

6.34 The parameters of a uniformly doped pn junction for silicon semiconductors are \( V_F = 0.65 \text{ V} \), \( V_T = 26 \text{ mV} \), \( T = 25^\circ \text{C} \), \( N_a = 1 \times 10^{17} \text{ cm}^{-3} \), and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \). Find (a) the depletion width \( W \) and (b) the maximum field \( \varepsilon_{\text{max}} \).

6.35 The parameters of a uniformly doped pn junction for silicon semiconductors are \( V_F = 0.5 \text{ V} \), \( V_T = 26 \text{ mV} \), \( T = 25^\circ \text{C} \), \( N_a = 2 \times 10^{16} \text{ cm}^{-3} \), and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \). Find the minority carrier concentrations at the edge of the depletion region: (a) electrons in the \( p \)-side, \( n_p \), and (b) holes in the \( n \)-side, \( p_n \).

6.36 The parameters of a uniformly doped pn junction for silicon semiconductors are \( V_F = 0.5 \text{ V} \), \( V_T = 26 \text{ mV} \), \( T = 25^\circ \text{C} \), \( N_a = 2 \times 10^{16} \text{ cm}^{-3} \), and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \). If \( V_F \) changes by \( \pm 15\% \), calculate the minimum and maximum values of the minority carrier concentrations at the edge of the depletion region: (a) electrons in the \( p \)-side, \( n_p \), and (b) holes in the \( n \)-side, \( p_n \).

6.6 Junction Current Density

6.37 The parameters of a reverse-biased pn junction with uniform doping of silicon semiconductors are \( N_a = 1 \times 10^{16} \text{ cm}^{-3} \), \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \), \( n_i = 1.5 \times 10^{10} \text{ cm}^{-3} \), \( D_n = 20 \text{ cm}^2/\text{s} \), \( D_p = 10 \text{ cm}^2/\text{s} \), \( \tau_{po} = \tau_{no} = 8 \times 10^{-6} \text{ s} \), and \( A_{pn} = 10^{-3} \text{ cm}^2 \). Find the reverse saturation current \( I_S \).

6.38 The parameters of a reverse-biased pn junction with uniform doping of silicon semiconductors are \( N_a = 1 \times 10^{16} \text{ cm}^{-3} \), \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \), \( n_i = 1.5 \times 10^{10} \text{ cm}^{-3} \), \( D_n = 20 \text{ cm}^2/\text{s} \), \( D_p = 10 \text{ cm}^2/\text{s} \), \( \tau_{po} = \tau_{no} = 8 \times 10^{-6} \text{ s} \), and \( A_{pn} = 10^{-3} \text{ cm}^2 \). If \( N_a \) and \( N_d \) change by \( \pm 15\% \), calculate the minimum and maximum values of the reverse saturation current \( I_S \).

6.39 The parameters of a reverse-biased pn junction with uniform doping of silicon semiconductors are \( N_a = 1 \times 10^{16} \text{ cm}^{-3} \), \( N_d = 50 N_a \), \( n_i = 1.5 \times 10^{10} \text{ cm}^{-3} \), \( D_n = 20 \text{ cm}^2/\text{s} \), \( D_p = 10 \text{ cm}^2/\text{s} \), \( \tau_{po} = \tau_{no} = 8 \times 10^{-6} \text{ s} \), and \( A_{pn} = 10^{-3} \text{ cm}^2 \). Find the reverse saturation current \( I_S \).

6.40 The parameters of a reverse-biased abrupt silicon pn junction are \( V_F = 0.5 \text{ V} \), \( T = 25^\circ \text{C} \), \( N_a = 10^{16} \text{ cm}^{-3} \), and \( N_d = 2 \times 10^{18} \text{ cm}^{-3} \). Calculate (a) \( W \), (b) the maximum field \( \varepsilon_{\text{max}} \), and (c) the junction capacitance \( C_j \) if \( A_{pn} = 10^{-3} \text{ cm}^2 \). If \( N_a \) and \( N_d \) change by \( \pm 15\% \), what will be the minimum and maximum values of the maximum field \( \varepsilon_{\text{max}} \)?

6.41 The parameters of a reverse-biased pn junction with uniform doping of silicon semiconductors are \( V_T = 26 \text{ mV} \), \( V_R = 10 \text{ V} \), \( T = 25^\circ \text{C} \), \( N_a = 1 \times 10^{16} \text{ cm}^{-3} \), and \( N_d = 2 \times 10^{15} \text{ cm}^{-3} \). Calculate the reverse voltage \( V_R \) that will give a junction capacitance of \( C_j = 5 \text{ pF} \) if \( A_{pn} = 10^{-3} \text{ cm}^2 \).

6.42 A uniformly doped silicon pn junction operating at \( T = 25^\circ \text{C} \) is to be designed such that at a reverse-biased voltage of \( V_R = 15 \text{ V} \), the maximum field is limited to \( \varepsilon_{\text{max}} = 5 \times 10^5 \text{ V/cm} \). Determine the maximum doping concentration in the \( n \)-region.
6.43 A uniformly doped silicon pn junction operating at \( T = 25^\circ\text{C} \) has a reverse-biased voltage of \( V_R = 12 \, \text{V} \). The charge in the n-region is to be limited to 15% of the total space charge, and the total junction capacitance is \( C_j = 4 \, \text{pF} \) if \( A_{pn} = 6 \times 10^{-4} \, \text{cm}^2 \). Determine (a) \( N_a \), (b) \( N_d \), and (c) \( V_{bi} \).

6.44 The parameters of a reverse-biased pn junction with uniform doping of silicon semiconductors are \( N_a = 2 \times 10^{16} \, \text{cm}^{-3} \), \( N_d = 5 \times 10^{15} \, \text{cm}^{-3} \), \( n_i = 1.8 \times 10^{10} \, \text{cm}^{-3} \), \( D_n = 20 \, \text{cm}^2/\text{s} \), \( D_p = 10 \, \text{cm}^2/\text{s} \), \( \tau_{po} = \tau_{no} = 8 \times 10^{-6} \, \text{s} \), and \( A_{pn} = 10^{-3} \, \text{cm}^2 \). Find the reverse saturation current \( I_S \).

6.45 The reverse saturation current of a forward-biased silicon pn junction diode is \( I_S = 5 \times 10^{-14} \, \text{A} \) at \( T = 25^\circ\text{C} \). Determine the required diode voltage to induce a diode current of (a) \( I_D = 1 \, \text{mA} \) and (b) \( I_D = 10 \, \text{mA} \).

6.46 The reverse saturation current of a forward-biased silicon pn junction diode is \( I_S = 5 \times 10^{-14} \, \text{A} \) at \( T = 25^\circ\text{C} \). Determine the forward-biased diode current for (a) \( V_F = 0.75 \, \text{V} \), (b) \( V_F = 1.0 \, \text{V} \), and (c) \( V_F = 1.2 \, \text{V} \).

6.47 The forward-biased current of a pn diode is \( I_D = 10 \, \text{mA} \) at \( T = 25^\circ\text{C} \). The GaAs pn junction at \( T = 300 \, \text{K} \) is \( I_D = 15 \, \text{mA} \). The forward diode voltage is \( V_F = 1.1 \, \text{V} \). Determine the reverse saturation current \( I_S \).

6.7 Temperature Dependence

6.48 The saturation current is \( I_S = 9.972 \times 10^{-15} \, \text{A} \) at \( T = 25^\circ\text{C} \). Find the value of \( I_S \) at \( T = 50^\circ\text{C} \). Assume \( E_g = 1.15 \, \text{eV} \).

6.49 If the junction temperature changes by 5 times, what will be changes in the junction current density \( J_S \), and the saturation current \( I_S \)? Assume \( E_g = 1.15 \, \text{eV} \).

6.50 If the junction temperature changes by 10 times, what will be changes in the junction current density \( J_S \), and the saturation current \( I_S \)? Assume \( E_g = 1.15 \, \text{eV} \).

6.8 High-Frequency AC Model

6.51 If the parameters of a pn junction are \( C_{po} = 4 \, \text{pF} \), \( V_f = 0.75 \, \text{V} \), \( m = 0.333 \), and \( V_D = -50 \, \text{V} \), calculate the value of \( C \).

6.52 If the parameters of a pn junction are \( C_{po} = 4 \, \text{pF} \), \( V_f = 0.75 \, \text{V} \), \( m = 0.333 \), and \( V_D = 0.7158 \, \text{V} \), calculate the value of \( C \).
CHAPTER 7

METAL OXIDE SEMICONDUCTOR FIELD-EFFECT TRANSISTORS

Learning Outcomes

After completing this chapter, students should be able to do the following:

• Describe the operation of metal oxide semiconductor field-effect transistors (MOSFETs).
• List the types of MOSFETs and their characteristics.
• Analyze and design MOSFET biasing circuits.
• Determine the small-signal model parameters of MOSFETs.
• Analyze and design MOSFET amplifiers.
• List the circuit configurations of MOSFET amplifiers and their relative advantages and disadvantages.
• Determine the frequency model of MOSFETs.
• Determine the frequency responses of MOSFET amplifiers.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_{vo}$, $G_{mo}$</td>
<td>No-load voltage gain and transconductance of an amplifier</td>
</tr>
<tr>
<td>$g_m$, $G_m$</td>
<td>Transconductance of a MOSFET and an amplifier</td>
</tr>
</tbody>
</table>
Symbol | Meaning
---|---
g\textsubscript{ds}, r\textsubscript{ds} | Small-signal drain–source conductance and resistance of a MOSFET
i\textsubscript{d}, I\textsubscript{D}, i\textsubscript{D} | AC, quiescent DC, and instantaneous DC drain currents
K\textsubscript{m}, K\textsubscript{pp} | MOS and process technology constants
K\textsubscript{p}, K\textsubscript{n} | MOS constants for PMOS and NMOS
L, W | Length and width of a MOSFET
R\textsubscript{i}, R\textsubscript{o} | Input and output resistances of an amplifier
r\textsubscript{o} | Small-signal output resistance of a transistor
\lambda, V\textsubscript{M} | Channel modulation length and voltage of a MOSFET
v\textsubscript{o}(t), v\textsubscript{o}(t) | Instantaneous DC and AC output voltages
V\textsubscript{t}, V\textsubscript{IN}, V\textsubscript{IP} | Threshold voltages of any MOSFET, NMOS, or PMOS
v\textsubscript{ds}, V\textsubscript{DS}, v\textsubscript{DS} | Small-signal AC, quiescent DC, and instantaneous DC drain-to-source voltages
v\textsubscript{gs}, V\textsubscript{GS}, v\textsubscript{GS} | Small-signal, quiescent DC, and instantaneous DC gate-to-source voltages

### 7.1 Introduction

In Chapter 2 we looked at an amplifier’s characteristics from an input–output perspective and found the specifications of amplifiers that satisfied certain input and output requirements. Internally, amplifiers use one or more transistors as amplifying devices, and these transistors are biased from a single DC supply to operate properly at a desired (quiescent) \( Q \)-point. Using transistors, we can build amplifiers that give a voltage (or current) gain, a high input impedance, or a high (or low) output impedance. The terminal behavior of an amplifier depends on the types of devices used within the amplifier.

Transistors are active devices with highly nonlinear characteristics. Thus, to analyze and design a transistor circuit, we need models of transistors. Creating accurate models requires detailed knowledge of the physical operation of transistors and their parameters as well as a powerful analytical technique. A circuit can be analyzed easily using simple models, but there is generally a trade-off between accuracy and complexity. A simple model, however, is always useful to obtain the approximate values of circuit elements for use in a design exercise and the approximate performance of the elements for circuit evaluation. The details of transistor operation, characteristics, biasing, and modeling are outside the scope of this text [1–3]. In this chapter, we will consider the operation and external characteristics of field-effect transistors using simple linear models.

### 7.2 Metal Oxide Field-Effect Transistors

The basic concept of field-effect transistors (FETs) has been known since the 1930s; however, FETs did not find practical applications until the early 1960s. Since the late 1970s, MOSFETs have become very popular; they are being used increasingly in integrated circuits (ICs). The manufacturing of MOSFETs is relatively simple. A MOSFET device can be made small, and it occupies a small silicon area in an IC chip. MOSFETs are currently used for very-large-scale integrated (VLSI) circuits such as microprocessors and memory chips.
A metal oxide semiconductor field-effect transistor (MOSFET) is a unipolar device. The current flow in a MOSFET depends on one type of majority carrier (electrons or holes). The output current of MOSFETs is controlled by an electric field that depends on a gate control voltage. There are two types of MOSFETs: enhancement MOSFETs and depletion MOSFETs.

### 7.3 Enhancement MOSFETs

There are two types of enhancement MOSFETs: $n$-channel and $p$-channel. An $n$-channel enhancement MOSFET is often referred to as an NMOS. The physical structure of an NMOS showing its terminal is illustrated in Fig. 7.1(a); a schematic appears in Fig. 7.1(b). Since the $p$-type substrate and the two $n^+$-type junctions are reverse biased, there will be a depletion region as shown in Fig. 7.1(b) by shaded lines. Two $n^+$-type regions act as low-resistance connections to the source and the drain. An insulating layer of silicon dioxide is formed on top of the $p$-type substrate by oxidizing the silicon. Ohmic contacts are provided to the $n^+$-regions for connection to the external circuit by leaving two windows on the silicon dioxide and depositing a layer of aluminum. The substrate B is normally connected to the source terminal. An $n$-channel is induced under the influence of an electric field; there is no physical $n$-channel between the drain and the source of an NMOS, as shown by the darker shade in Fig. 7.1(b). The symbol for an NMOS is shown in Fig. 7.1(c), where the arrow points from the $p$-type region to the $n$-type region. An NMOS is often represented by the abbreviated symbol shown in Fig. 7.1(d) in which the arrowhead indicates the direction of the current.

A $p$-channel enhancement-type MOSFET, often referred to as a PMOS, is formed by two $p^+$-type regions on top of the $n$-type substrate, as shown in Fig. 7.2(a) and (b). The $p$-regions offer low resistances. The symbol for a PMOS is the same as that for an NMOS, except that the direction of the arrow is reversed, as shown in Fig. 7.2(c). The abbreviated symbol is shown in Fig. 7.2(d).

![Structure and symbols of an n-channel enhancement MOSFET](image-url)
7.3.1 Operation

An NMOS is operated with positive gate and drain voltages relative to the source, as shown in Fig. 7.3(a), whereas a PMOS is operated with negative gate and drain voltages relative to the source, as shown in Fig. 7.3(b). Their substrates are connected to the source terminal.

An NMOS may be viewed as consisting of two diode junctions that are formed between the substrate and the source and between the substrate and the drain, as shown in Fig. 7.4(a). The hypothetical diodes are in series and back to back, as shown in Fig. 7.4(b). The NMOS can operate in any of the four operating regions: cutoff region, linear ohmic, nonlinear ohmic, and saturation.

Cutoff Region

The gate-to-source voltage $v_{GS}$ is greater than zero but less than the threshold voltage $V_t: 0 \leq v_{GS} \leq V_t$. A positive value of $v_{DS}$ will reverse-bias the right-hand diode, and the drain current $i_D$ will be approximately zero if the gate-to-source voltage $v_{GS}$ is zero.

![FIGURE 7.2 Structure and symbols of a $p$-channel enhancement MOSFET](image)

![FIGURE 7.3 Biasing of an NMOS and a PMOS](image)
Linear Ohmic Region

$v_{GS} \geq V_t$ and $0 < v_{DS} \ll (v_{GS} - V_t)$. A positive value of $v_{GS}$ will establish an electric field, which will attract negative carriers from the substrate and repel positive carriers. As a result, a layer of substrate near the oxide insulator becomes less $p$-type, and its conductivity is reduced. As $v_{GS}$ increases, the surface near the insulator will attract more electrons than holes and will behave like an $n$-type channel. The minimum value of $v_{GS}$ that is required to establish a channel is called the threshold voltage $V_t$. The drain current at $v_{GS} = V_t$ is very small. For $v_{GS} > V_t$, the drain current $i_D$ increases almost linearly with $v_{DS}$ for small values of $v_{DS}$, as shown in Fig. 7.4(c). If the drain-to-source voltage is low (usually less than 1 V), the drain current $i_D$ can be calculated from Ohm’s law ($i_D = v_{DS}/r_{DS}$). The conductance of the channel between the drain and the source can be found from

$$g_{DS} = \frac{1}{r_{DS}} = \frac{W}{L} \mu_n Q_n$$

where $\mu_n = \text{mobility of the electrons in the reverse-biased (also called the inversion) layer under the oxide layer}$

$Q_n = \text{magnitude of the reverse-biased layer charge per unit area}$

$W = \text{channel width}$

$L = \text{channel length}$

We can find $Q_n$ from the gate oxide capacitance $C_{ox}$ and the voltage difference ($v_{GS} - V_t$) as given by

$$Q_n = C_{ox} (v_{GS} - V_t)$$
Substituting $Q_n$ from Eq. (7.2) into Eq. (7.1), we can write the drain current as

$$i_D = g_{DS}V_{DS} = \frac{W}{L} \mu_n C_{ox} (v_{GS} - V_t) V_{DS}$$  \hspace{1cm} (7.3)

Therefore, the NMOS can be operated as a variable resistance $r_{DS}(= V_{DS}/i_D)$ by varying $v_{GS}$ and will vary linearly for a small value of $v_{DS}$.

**Nonlinear Ohmic Region**

$v_{GS} \geq V_t$ and $0 < v_{DS} < (v_{GS} - V_t)$. Increasing $v_{DS}$ does not change the depth of the channel at the source end. However, it increases the drain-to-gate voltage $V_{DG}$ or decreases the gate-to-drain voltage $V_{GD}$, and the channel width decreases at the drain end. As a result, the channel becomes narrower at the drain end with a tapered shape, as shown in Fig. 7.4(d). When $V_{DS}$ becomes sufficiently large and $V_{GD}$ is less than $V_t$ [i.e., when $V_{GD} = (v_{GS} - v_{DS}) \leq V_t$], pinch-down occurs at the drain end of the channel. The $i_D$-$V_{DS}$ characteristic will be nonlinear. Any further increase in $V_{DS}$ does not cause a large increase in $i_D$, and the transistor operates in the saturation region. If we consider a small incremental drain-to-source voltage $v$ along the channel, we can rewrite Eq. (7.2) as

$$Q_n(v) = C_{ox} (v_{GS} - V_t - v)$$

which can be applied to obtain the drain current as given by

$$i_D = \frac{W}{L} \mu_n C_{ox} \int_0^{V_{DS}} (v_{GS} - V_t - v) dv = \frac{W}{L} \mu_n C_{ox} \left[ (v_{GS} - V_t) V_{DS} - \frac{V_{DS}^2}{2} \right]$$  \hspace{1cm} (7.4)

Equation (7.4) can also be written [4] as

$$i_D = \frac{K_m}{2} [2(v_{GS} - V_t) V_{DS} - V_{DS}^2]$$  \hspace{1cm} (7.5)

where $K_m = (W/L) \mu_n C_{ox}$ is called the MOS constant whose value depends on the physical parameters. Equation (7.4) can be expressed in a more general form in terms of external voltages $v_{GS}$ and $v_{DS}$:

$$i_D = K_n [2(v_{GS} - V_t) V_{DS} - V_{DS}^2]$$  \hspace{1cm} (7.6)

Here $K_n$ is a MOS constant given by

$$K_n = \frac{K_m}{2} = \frac{W}{L} \frac{\mu_n C_{ox}}{2}$$  \hspace{1cm} (7.7)

where $L =$ channel length (typically 10 $\mu$m) in m

$W =$ channel width (typically 100 $\mu$m) in m

$\mu_n =$ surface mobility of electrons = 600 $\text{cm}^2/(\text{V-s})$
$\varepsilon_o = \text{permittivity of free space} = 8.85 \times 10^{-14} \text{ F/cm}$

$\varepsilon_{ox} = \text{dielectric constant of SiO}_2 = 4$

$t_{ox} = \text{thickness of the oxide}$

$C_{ox} = \text{MOSFET capacitance per unit area}$

For $t_{ox} = 0.10 \ \mu\text{m}$, $C_{ox}$ is $3.54 \times 10^{-8} \text{ F/cm}^2$. $K_{pp}$ is the product of $\mu_n$ and $C_{ox}$ (i.e., $K_{pp} = \mu_n C_{ox} = \mu_n \varepsilon_{ox} / t_{ox}$), which depends on the process parameters and will be constant for a given technology. By choosing $W = 2L$, we can make the two constants equal: $K_m = K_n$.

**Saturation Region**

$v_{GS} \geq V_t$ and $v_{DS} \geq (v_{GS} - V_t)$. Figure 7.5 shows the plot of Eq. (7.6) for three values of gate-to-source voltages. We can find the $v_{DS}$ for the peak drain current from the condition, $di_{D}/dv_{D} = 0$. That is,

$$
\frac{di_{D}}{dv_{DS}} = \frac{K_n d}{d v_{DS}} [2(v_{GS} - V_t) v_{DS} - v_{DS}^2] = 0
$$

which gives $v_{DS} = v_{GS} - V_t$ at which the saturation occurs—that is, $v_{DS(sat)} = v_{GS} - V_t$. Substituting $v_{DS} = v_{GS} - V_t$ in Eq. (7.6) gives the drain current in the saturation region:

$$
i_{D} = K_n(v_{GS} - V_t)^2 \quad \text{(7.8)}
$$

$V_t$ should be substituted for by $V_{TN}$, the threshold voltage of an NMOS, or $V_{TP}$, the threshold voltage of a PMOS. Substituting $v_{GS} = (v_{DS} + V_t)$ in Eq. (7.8) gives the peak (saturation) drain current as

$$
i_{D(sat)} = K_n(v_{DS} + V_t - V_t)^2 = K_n v_{DS}^2 \quad \text{(7.9)}
$$

The complete $i_{D}$-$v_{DS}$ characteristic for a constant $v_{GS}$ is shown in Fig. 7.6. In practice, there is a very slight increase in drain current $i_{D}$ as $v_{DS}$ increases, and the slope of the $i_{D}$-$v_{DS}$ characteristic has a finite value.
7.3.2 Output and Transfer Characteristics

The drain characteristics of an NMOS are shown in Fig. 7.7(a), and the complete transfer characteristics are shown in Fig. 7.7(b) for an NMOS and a PMOS.

Increasing $v_{DS}$ beyond the breakdown voltage, denoted by $V_{BD}$, causes an avalanche breakdown in the channel, and the drain current rises rapidly. This mode of operation must be avoided because a MOSFET can be destroyed by excessive power dissipation. Since the reverse voltage is highest at the drain end, the breakdown occurs at this end. The breakdown voltage specified by the manufacturer is typically in the range of 20 V to 100 V. Also, a large value of $v_{GS}$ will cause a dielectric breakdown in the oxide layer of the device.

Since the gate is insulated from the effective channel in an NMOS, no gate current can flow and consequently the resistance between the gate and the source terminals is theoretically infinite. In practice, the resistance is finite but very large, on the order of $10^8 \text{ M}\Omega$.

The output characteristics of an NMOS shown in Fig. 7.7(a) can be described by

$$i_D = K_n[2(v_{GS} - V_t)v_{DS} - v_{DS}^2] \quad \text{for } v_{GS} > V_t \text{ and } v_{DS} < (v_{GS} - V_t)$$

$$= K_n(v_{GS} - V_t)^2 \quad \text{for } v_{GS} > V_t \text{ and } v_{DS} \geq (v_{GS} - V_t)$$

The equations for the NMOS can be applied to a PMOS if we substitute $V_t = -V_{tP}$ and $v_{DS} = -v_{DS}$. $V_t = V_{tN}$ for NMOS.

![Figure 7.6](image)

**FIGURE 7.6** $i_D-v_{DS}$ characteristic for a constant $v_{GS} (> V_t)$

![Figure 7.7](image)

**FIGURE 7.7** Drain and transfer characteristics of enhancement MOSFETs
7.3.3 Channel Length Modulation

If we increase the drain voltage \( v_{DS} \), the voltage across the oxide layer decreases; therefore the inversion charge density decreases at the drain terminal as shown in Fig. 7.4(d). At \( v_{DS} = v_{DS(sat)} = v_{GS} - V_t \), the inversion charge density at the drain terminal becomes zero. As we increase \( v_{DS} \), the zero density point moves toward the source terminal as shown in Fig. 7.8. Increasing \( v_{DS} \) increases the biasing voltage of the \( pn \) junction and causes the depletion region at the drain terminal to extend laterally into the channel, thereby reducing the effective channel length. As a result, the effective channel length is modulated by the drain-to-source voltage \( v_{DS} \). The depletion width extending into the \( p \)-region of a \( pn \) junction with \( v_{DS} \) biasing can be found from Eq. (6.40) as

\[
x_p = \sqrt{\frac{2\varepsilon_s}{qN_a} (|\phi_{Fp}| + v_{DS})}
\]  
(7.11)

where \( |\phi_{Fp}| \) is the field potential due to the \( p \)-region given by

\[
\phi_{Fp} = -\frac{kT}{q} \ln \left( \frac{N_a}{n_i} \right) = -V_T \ln \left( \frac{N_a}{n_i} \right)
\]  
(7.12)

We can find the extension of the space charge region

\[
\Delta L = x_p (v_{DS(sat)} + \Delta v_{DS}) - x_p (v_{DS(sat)}
\]

which after substituting \( \Delta v_{DS} = v_{DS} - v_{DS(sat)} \) in Eq. (7.11) gives

\[
\Delta L = \sqrt{\frac{2\varepsilon_s}{qN_a} \left[ |\phi_{Fp}| + v_{DS(sat)} + \Delta v_{DS} \right]}
\]  
(7.13)

Since the drain \( i_D \) is inversely proportional to the effective channel length, we get

\[
i_D \propto \frac{1}{L - \Delta L} = \frac{1}{L (1 - \Delta L / L)} = \frac{1}{L} \left( 1 + \frac{\Delta L}{L} \right)
\]  
(7.14)
Since $\Delta L$ is a function of $v_{DS}$, the fractional change in the channel length is proportional to the drain-source biasing voltage. That is,

$$\frac{\Delta L}{L} = \lambda v_{DS} \quad (7.15)$$

Here $\lambda$ is called the channel length modulation. We can include the channel length modulation effect in Eq. (7.15) to Eq. (7.8) as follows:

$$i_D = K_n(v_{GS} - V_t)^2(1 + \lambda v_{DS}) \quad (7.16)$$

This gives the slope of the output characteristics shown in Fig. 7.7(a). The plot of the $i_D$-$v_{DS}$ characteristics for the saturation region is shown in Fig. 7.9. If we extrapolate the characteristics to the $v_{DS}$-axis, they intercept at a point $V_M$, which is known as the channel modulation voltage such that $V_M = 1/\lambda$.

### EXAMPLE 7.1

**Finding the channel modulation voltage** Determine the channel modulation voltage $V_M$. The NMOS parameters are these: substrate impurity doping concentration $N_a = 2 \times 10^{16}$ cm$^3$, threshold voltage $V_{IN} = 0.5$ V, channel length $L = 10$ $\mu$m, $v_{GS} = 1.5$ V, and $v_{DS} = 5$ V.

**SOLUTION**

$$V_T = 25.8 \text{ mV}, \quad N_a = 2 \times 10^{16} \text{ cm}^3, \quad n_i = 1.5 \times 10^{10} \text{ cm}^3, \quad V_{IN} = 0.5 \text{ V}, \quad L = 10 \text{ $\mu$m}, \quad v_{GS} = 1.5 \text{ V}, \quad \text{and} \quad v_{DS} = 5 \text{ V}.$$  

From Eq. (7.12),

$$\phi_{FP} = -V_T \ln \left( \frac{N_a}{n_i} \right) = -25.8 \times 10^{-3} \times \ln \left( \frac{2 \times 10^{16}}{1.5 \times 10^{10}} \right) = -0.364$$

$$V_{DS(sat)} = V_{GS} - V_{IN} = 1.5 - 0.5 = 1 \text{ V}$$

$$\Delta v_{DS} = V_{DS} - V_{DS(sat)} = 5 - 1 = 4 \text{ V}$$
7.3.4 Substrate Biasing Effects

The source-to-substrate \( p^n \) junction must always be zero or reverse biased, so \( v_{SB} \) must always be greater than or equal to zero; otherwise, electrons or holes will flow from the drain to the substrate rather than the source terminals. The body or the substrate of a MOSFET is often connected to the ground. In MOSFET circuits, the source and body may not be at the same potential as shown in Fig. 7.10, and applications of \( v_{SB} \) will increase the depletion region. In integrated circuits, however, the substrate is usually common to many MOS transistors. To maintain the cutoff condition for all the substrate-to-channel junctions, the substrate is usually connected to the most negative power supply in an NMOS circuit (the most positive in a PMOS circuit).

The reverse-biased voltage will widen the depletion region, thereby reducing the effective channel depth. Therefore, we need to apply more gate voltage to compensate for the channel reduction, and \( v_{SB} \) will affect the effective threshold voltage \( V_t \) of the MOSFET. It can be shown that increasing \( v_{SB} \) results in an increase in \( V_t \) as given by [5]

\[
V_t = V_{to} + \frac{2q\varepsilon_0 N_a}{C_{ox}} \left[ \sqrt{2|\phi_{Fp}| + v_{SB}} - \sqrt{2|\phi_{Fp}|} \right] \tag{7.17}
\]

Here \( v_{SB} \) is the source-to-substrate voltage, and \( V_{to} \) is the initial threshold voltage with \( v_{SB} = 0 \). Note that \( v_{SB} \) must always be positive for an NMOS and negative for a PMOS. Also, \( V_t \) must always be positive for an NMOS and negative for a PMOS.

### 7.3.5 Complementary MOS (CMOS)

The complementary metal oxide semiconductor (CMOS) consists of an \( n^- \) channel enhancement-mode device \((V_{tN} > 0)\) in series with a \( p^- \) channel enhancement-mode device \((V_{tP} < 0)\). The cross-section of a

![FIGURE 7.10](image-url)  
**FIGURE 7.10** Applying source-to-substrate voltage to an NMOS
CMOS is shown in Fig. 7.11. The NMOS transistor is implemented directly in the \( p \)-type substrate, while the PMOS transistor is fabricated in a specially created \( n \)-region, known as an \( n \)-well. The two devices are isolated from each other by a thick region of oxide that functions as an insulator. An external body terminal is also made from the \( p \)-type body and the \( n \)-well. Due to their unique advantages, such as very low power consumption, CMOS circuits are commonly used in integrated circuits. The CMOS inverter, which is the basis of CMOS digital electronics, is covered in detail in Sec. 15.7. CMOS technology has taken over many IC applications and continues to grow.

**KEY POINTS OF SECTION 7.3**

- An MOSFET is a voltage-controlled nonlinear device. A voltage between the gate and the source develops an electric field, which then controls the flow of drain current. Therefore, the drain current depends on the gate-to-source voltage, and an FET gives a transconductance gain.
- MOSFETs can be classified into two types: enhancement MOSFET and depletion MOSFETs. Each type can be either \( n \)-channel or \( p \)-channel.
- The output characteristic of a MOSFET can be divided into three regions: the cutoff region, in which the MOSFET is in the off state; the saturation region, in which the transistor exhibits a high output resistance and has a transconductance; and the ohmic region, in which the transistor offers a low resistance. A MOS is operated as an amplifier in the saturation region and as a switch in the ohmic region.

### 7.4 Depletion MOSFETs

The construction of an \( n \)-channel depletion MOSFET is very similar to that of an NMOS. An actual channel is formed by adding \( n \)-type impurity atoms to the \( p \)-type substrate, as shown in Fig. 7.12(a). The symbol for an \( n \)-channel depletion MOSFET is shown in Fig. 7.12(b); this symbol is often abbreviated to the one shown in Fig. 7.12(c). Note that the vertical line is bold or darker. An \( n \)-channel depletion MOSFET is normally operated with a positive voltage between the drain and the source terminals. However, the voltage between the gate and the source terminals can be positive, zero, or negative, whereas in an NMOS \( v_{GS} \) is positive.

#### 7.4.1 Operation

The operation of an \( n \)-channel depletion MOSFET is similar to that of an NMOS. A depletion NMOS is off when its gate-to-source voltage \( v_{GS} \) is less than \( -V_p \), whereas an NMOS is off when \( v_{GS} \leq V_{IN} \). The channel is fully established at \( v_{GS} = 0 \) for a depletion NMOS and at \( v_{GS} = V_{IN} \) for an NMOS. Let us assume that
the gate-to-source voltage is zero: $v_{GS} = 0$. If $v_{DS}$ is increased from zero to some small value ($\approx 1$ V), the drain current follows Ohm’s law ($i_D = v_{DS}/r_{DS}$) and is directly proportional to $v_{DS}$. Any increase in the value of $v_{DS}$ beyond $|V_p|$, known as the pinch-down voltage, does not increase the drain current significantly. The region beyond pinch-down is called the saturation region. The value of the drain current that occurs at $v_{DS} = |V_p|$ (with $v_{GS} = 0$) is termed the drain-to-source saturation current $I_{DSS}$. The complete $i_D$-$v_{DS}$ characteristic for $v_{GS} = 0$ is shown in Fig. 7.13. In practice, there is a very slight increase in drain current $i_D$ as $v_{DS}$ increases beyond $|V_p|$, and the slope of the $i_D$-$v_{DS}$ characteristic has a finite value. Saturation occurs at the value of $v_{DS}$ at which the gate-to-channel voltage at the drain end equals $V_p$. That is,

$$v_{GD} = v_{GS} - v_{DS} = V_p \quad \text{or} \quad v_{DS} = v_{GS} - V_p \quad (7.18)$$

If $v_{GS}$ is negative, some of the electrons in the $n$-channel area will be repelled from the channel and a depletion region will be created below the oxide layer, as shown in Fig. 7.14(a). This depletion region will result in a narrower channel. For $v_{GS} > 0$, a layer of substrate near the $n$-type channel becomes less $p$-type and its conductivity is enhanced as shown in Fig. 7.14(b). A positive value of $v_{GS}$ increases the
effective channel width in much the same way as in an NMOS. When the effective channel is increased, the transistor is said to be operating in the enhancement mode. The $i_D-v_{DS}$ characteristics for various values of $v_{GS}$ are shown in Fig. 7.15(a).

### 7.4.2 Output and Transfer Characteristics

The transfer characteristics are shown in Fig. 7.15(b) for an $n$-channel and a $p$-channel MOSFET. The output characteristics can be divided into three regions: ohmic, saturation, and cutoff.

---

**FIGURE 7.15** Drain and transfer characteristics of depletion MOSFETs
Ohmic Region

In the ohmic region, the drain-to-source voltage \( v_{DS} \) is low and the channel is not pinched down. The drain current \( i_D \) can be expressed as

\[
i_D = K_n[2(v_{GS} - V_p) - v_{DS}^2] \quad \text{for } 0 < v_{DS} \leq (v_{GS} - V_p)
\]

which, for a small value of \( v_{DS} \) (\( v_{DS} \ll |V_p| \)), can be reduced to

\[
i_D = K_n[2(v_{GS} - V_p)v_{DS}]
\]

where \( K_n = I_{DSS}/V_p^2 \).

Saturation Region

In the saturation region, \( v_{DS} \geq (v_{GS} - V_p) \). The drain-to-source voltage \( v_{DS} \) is greater than the pinch-down voltage, and the drain current \( i_D \) is almost independent of \( v_{DS} \). For operation in this region, \( v_{DS} \geq (v_{GS} - V_p) \). Substituting the limiting condition \( v_{DS} = (v_{GS} - V_p) \) in Eq. (7.19) gives the drain current \( i_D \) as

\[
i_D = K_n[2(v_{GS} - V_p)(v_{GS} - V_p) - (v_{GS} - V_p)^2]
\]

\[
= K_n(v_{GS} - V_p)^2
\]

Equation (7.21) represents the transfer characteristic, which is shown in Fig. 7.15(b) for both \( n \)- and \( p \)-channels. For a given value of \( i_D \), Eq. (7.21) gives two values of \( v_{GS} \), and only one value is the acceptable solution so that \( v_{GS} > V_p \) for the \( n \)-channel and \( v_{GS} < V_p \) for the \( p \)-channel. The pinch-down locus, which describes the boundary between the ohmic and saturation regions, can be obtained by substituting \( v_{GS} = v_{DS} + V_p \) into Eq. (7.21):

\[
i_D = K_n(v_{DS} + V_p - V_p)^2 = K_n v_{DS}^2
\]

This defines the pinch-down locus and forms a parabola similar to Eq. (7.9) and Fig. 7.5.

Cutoff Region

In the cutoff region, the gate-to-source voltage is less than the pinch-down voltage. That is, \( v_{GS} < V_p \) for the \( n \)-channel and \( v_{GS} > V_p \) for the \( p \)-channel, and the MOSFET is off. The drain current is zero: \( i_D = 0 \).

7.5 MOSFET Models and Amplifier

Since the drain currents of the enhancement and depletion MOSFETs depend on the gate–source voltage, they are known as voltage-dependent devices and exhibit similar output characteristics and, the same model can be applied to both of them with reasonable accuracy. An NMOS circuit with the transistor biased to operate in the saturation region is shown in Fig. 7.16(a). Using KVL around the drain-to-source loop gives

\[
V_{DD} = v_{DS} + R_D i_D
\]

\[
i_D = \frac{V_{DD} - v_{DS}}{R_D}
\]
which describes the load line, and intersects the \( i_D \)-axis at \( V_{DD}/R_D \) and the \( v_{DS} \)-axis at \( V_{DD} \) as shown in Fig. 7.16(b). The intersection of this load line with the \( i_D-v_{DS} \) characteristic gives the operating (or quiescent) point for a given value of \( V_{GS} \). Let us assume that the drain current, drain-to-source voltage, and gate-to-source voltage have initial quiescent values of \( I_D, V_{DS}, \) and \( V_{GS} \), respectively. In a MOSFET amplifier, an AC input signal is normally superimposed on the gate voltage. If a small AC signal \( v_{gs} \) is connected in series with \( V_{GS} \), it will produce a small variation in the drain-to-source voltage \( v_{DS} \) and the drain current \( i_D \). That is, if the gate-to-source voltage varies by a small amount, such that \( V_{GS} = V_{GS1} + v_{gs} \), there will be corresponding changes in the drain current and drain-to-source voltage such that \( v_{DS} = V_{DS} + v_{ds} \) and \( i_D = I_D + i_d \). This situation is shown in Fig. 7.16(b). The small variations of the drain current \( i_D \), as \( i_d \), and the drain-to-source voltage \( v_{DS} \), as \( v_{ds} \), around the operating point are shown in Fig. 7.16(b). The drain-to-source variation \( v_{ds} \) will equal the voltage gain times \( v_{gs} \). If the values of \( i_d, v_{gs}, \) and \( v_{ds} \) are small, Fig. 7.16(b) can be represented by the small-signal circuit shown in Fig. 7.16(c). Therefore, we need two types of models for MOSFETs: a DC model and a small-signal model.

### 7.5.1 DC Models

The large-signal (DC) models of MOSFETs are nonlinear. The drain characteristics as shown in Fig. 7.7(a) and 7.15(b) of \( i_D \) as a function of \( v_{DS} \) for different values of \( V_{GS} \) describe the large-signal model of a MOSFET.

Since the gate-channel has an oxide layer, the gate current will be negligibly small. Thus, MOSFETs can be represented by the simple DC model of Fig. 7.17.
7.5.2 Small-Signal AC Models

The small-signal behavior of the MOSFET in Fig. 7.17 can be represented by a small-signal AC equivalent circuit consisting of a voltage-dependent current source $g_m v_{gs}$ in parallel with an output resistance $r_o$ representing a finite slope of the $i_D$-$v_{DS}$ characteristic. This circuit is shown in Fig. 7.18(a). Since the gate current $i_g$ of MOSFETs is very small, tending to zero, the gate-to-source terminals are open circuits.

Applying the relations between Norton’s and Thevenin’s theorems, we can represent the current source in Fig. 7.18(a) by a voltage source, as shown in Fig. 7.18(b). We find $v_{ds}$ from

$$v_{ds} = i_d r_o - r_o g_m v_{gs} = i_d r_o - \mu_g v_{gs}$$

(7.24)

where $\mu_g$ is the open-circuit voltage gain of the MOSFETs and is given by

$$\mu_g = r_o g_m$$

(7.25)

The circuits of Fig. 7.18[(a) and (b)] are referred to as the Norton and Thevenin circuits, respectively, and they are equivalent. $r_o$ is the small-signal output resistance, and $g_m$ is the transconductance gain of the MOSFET. Their values are dependent on the operating point and are quoted at a specified operating point $(V_{DS}, I_D)$.

**Small-Signal Output Resistance $r_o$**

The small-signal output resistance is the inverse slope of the $i_D$-$v_{DS}$ characteristic in the pinch-down or saturation region. We can use Eq. (7.16) to find the value of the output resistance $r_o$ as given by

$$\frac{1}{r_o} = \frac{d i_D}{d v_{DS}} = \frac{I_D}{|V_M|} = \lambda I_D \quad \text{for all MOSFETs}$$

(7.26)
Here \( V_M \) is called the channel modulation voltage and \( \lambda = 1/|V_M| \) is called the channel modulation length (see Fig. 7.9). The parameter \( V_M \) is positive for a \( p \)-channel device and negative for an \( n \)-channel device. Its typical magnitude is 100 V. \( V_M \) is analogous to the Early voltage \( V_A \) of bipolar transistors (Sec. 8.3.3).

**Transconductance \( g_m \)**

The transconductance is the slope of the transfer characteristic \((i_D \text{ versus } v_{GS})\) and is defined as the change in the drain current corresponding to a change in the gate-to-source voltage. It is expressed by

\[
g_m = \frac{\delta i_D}{\delta v_{GS}} \bigg|_{v_{GS}=\text{constant}}
\]

Assuming \( i_D \approx I_D, v_{GS} \approx V_{GS}, \text{ and } v_{DS} \approx V_{DS} \), the small-signal transconductance of an NMOS can be derived from Eq. (7.10):

\[
g_m = \frac{\delta i_D}{\delta v_{GS}} = 2K_n(V_{GS} - V_t) \quad \text{for enhancement MOSFETs}
\]

\[
= g_m \left(1 - \frac{V_{GS}}{V_t}\right) \quad \text{for enhancement MOSFETs}
\]

where \( g_m = -2K_nV_t \). \hspace{1cm} (7.28)

The small-signal transconductance of a depletion MOSFET can be derived from Eq. (7.21):

\[
g_m = \frac{\delta i_D}{\delta v_{GS}} = 2K_n(v_{GS} - V_p) \quad \text{for depletion MOSFETs}
\]

\[
= g_m \left(1 - \frac{V_{GS}}{V_p}\right) \quad \text{for depletion MOSFETs}
\]

where \( g_m = -2K_nV_p = -2I_{DSS}/V_p \). \hspace{1cm} (7.31)

\( g_m \) is the transconductance corresponding to \( v_{GS} = 0 \), and it varies linearly with \( v_{GS} \), as shown in Fig. 7.19. For \( v_{GS} = 0 \), the device is cut off; thus it is never operated with a value of \( g_m \). The pinch-down voltage \( V_p \) can be determined experimentally by plotting \( g_m \) versus \( v_{GS} \) and then extrapolating to the \( v_{GS} \)-axis. This is a very useful method for determining \( V_p \) and \( V_t \) for a MOSFET.

**FIGURE 7.19** Variation of \( g_m \) with \( v_{GS} \) for MOSFETs
7.5.3 PSpice Models

The symbol for a MOSFET is M. The statements have the following general forms:

\[ M<\text{name}> \text{ ND } \text{ NG } \text{ NS } \text{ NB } \text{ MMOD} \]

where ND, NG, NS, and NB are the drain, gate, source, and bulk (or substrate) nodes, respectively. MMOD is the model name. The model statement has the following general forms:

\[ \text{.MODEL MMOD NMOS (P1=A1 P2=A2 P3=A3 \ldots P_N=AN)} \]

for \( n \)-channel MOSFETs

\[ \text{.MODEL MMOD PMOS (P1=A1 P2=A2 P3=A3 \ldots P_N=AN)} \]

for \( p \)-channel MOSFETs

Here NMOS and PMOS are the type symbols for \( n \)-channel and \( p \)-channel MOSFETs, respectively; and \( P_1, P_2, \ldots, P_N \) and \( A_1, A_2, \ldots, A_N \) are the parameters and their values, respectively.

Consider the NMOS of type 2N4351, whose parameters are \( V_t = 1 \text{ V to } 5 \text{ V} \), and \( g_m = 1 \text{ mA} / \text{V} \) at \( i_D = 2 \text{ mA} \) and at \( v_{GS} = 10 \text{ V} \). Taking the geometric mean value, we get \( V_t = \sqrt{1 \times 5} = 2.24 \text{ V} \), which is specified in PSpice/SPICE by \( V_{TO} = 2.24 \text{ V} \). The constant \( K_n \) can be found from Eqs. (7.10) and (7.27):

\[ i_D = K_n(v_{GS} - V_t)^2 \]

\[ g_m = 2K_n(v_{GS} - V_t) \]

These equations can be written in the form of a ratio as

\[ \frac{g_m^2}{i_D} = \frac{4K_n^2(v_{GS} - V_t)^2}{K_n(v_{GS} - V_t)^2} = \frac{4K_n}{1} \]

which, for \( g_m = 1 \text{ mA} / \text{V} \) and \( i_D = 2 \text{ mA} \), gives \( K_n = 125 \mu\text{A} / \text{V}^2 \). The ratio \( W/L \) can be found from Eq. (7.7):

\[ \frac{W}{L} = \frac{2K_n}{\mu_d C_{ox}} = \frac{2 \times 125 \times 10^{-6}}{600 \times 3.54 \times 10^{-8}} = 11.8 \]

Assume \( L = 10 \mu\text{m} \); then \( W = 118 \mu\text{m} \). Also assume \( |V_M| = 1/\lambda = 200 \text{ V} \) and \( \lambda = 5 \text{ mV}^{-1} \). Then NMOS 2N4351 can be specified in PSpice/SPICE by the following statements [6, 7]:

\[ M1 \text{ ND } \text{ NG } \text{ NS } \text{ NB } \text{ M2N4351} \]

\[ \text{.MODEL M2N4351 NMOS (KP=125U VTO=2.24 L=10U W=118U LAMBDA=5M)} \]

The MOS transistor has a length of \( 0.6 \mu\text{m} \) at minimum and can be expanded by integer increments of \( 0.3 \mu\text{m} \). The minimum width is \( 0.9 \mu\text{m} \) and can be expanded by integer increments of \( 0.3 \mu\text{m} \). Generally, attributes of an NMOS are \( L = 6 \text{ to } 10 \mu\text{m} \), \( W = 118 \mu\text{m} \), \( AD = 720 \text{ to } 283.2 \mu\text{m} \), \( AS = 720 \mu\text{m} \), \( PD = 302.4 \text{ to } 120.4 \mu\text{m} \), and \( PS = 302.4 \text{ to } 120.4 \mu\text{m} \). Generally, \( AS = AD = (2.4 \mu\text{m} \times W) \), and \( PS = PD = (2.4 \mu\text{m} \times W) \).

\[ \text{NOTE} \quad \text{The full data sheets for MOSFETs (e.g., NMOS of type 2N4351) can be found at the http://www.alldatasheet.com or by searching MOSFET data sheets at http://www.google.com.} \]

7.5.4 Small-Signal Analysis

Once the \( Q \)-point is established and the small-signal parameters are determined, we can find the small-signal parameters of the amplifier in Fig. 7.16(a) in response to a small-signal voltage \( v_{gs} \). For a small AC
signal, the DC supply offers zero impedance; $V_{DD}$ and $V_{GS}$ can be short-circuited. That is, one side of $R_D$ is connected to the ground. The small-signal AC equivalent circuit of the amplifier is shown in Fig. 7.20(a). Replacing the transistor $M_1$ by its transconductance model of Fig. 7.18(a), the small-signal AC equivalent circuit is shown in Fig. 7.20(b). The following steps are involved in analyzing an amplifier circuit:

1. DC biasing analysis of the transistor circuit
2. Determination of the small-signal parameters $g_m$ and $r_o$ of the transistor
3. Determination of the AC equivalent circuit of the amplifier
4. Performing the small-signal analysis for finding $R_i$, $A_{vo}$, and $R_o$

From Fig. 7.20(b), the small-signal input resistance can be found from

$$R_i = \frac{v_{gs}}{i_g} = \infty$$

Thevenin’s equivalent output resistance, looking from the output side for the condition $v_{gs} = 0$, can be found from

$$R_o = r_o \parallel R_D \quad (7.33)$$

The transconductance of the amplifier $G_{mo}$, which is the same as the transconductance of the transistor, is

$$G_{mo} = \frac{i_o}{v_{gs}} = -g_m \quad (7.34)$$

We can write the small-signal output voltage $v_o$ as

$$v_o = -g_m(r_o \parallel R_D)v_{gs} \quad (7.35)$$
which gives the small-signal voltage gain $A_{vo}$ as

$$A_{vo} = \frac{v_o}{v_{gs}} = -g_m(r_o \parallel R_D) \quad (7.36)$$

If we substitute $r_o = V_M/I_D$, Eq. (7.36) becomes

$$A_{vo} = -g_m\left(\frac{V_M R_D}{V_M + I_D R_D}\right) \quad (7.37)$$

Therefore, for obtaining a large voltage gain, the $g_m R_D$ product must be made large and the DC biasing drain current $I_D$ should be small. This will require both a large DC supply voltage $V_{DD}$ and a large value of resistance $R_D$. Figure 7.20[(c) and (d)] shows the equivalent voltage and transconductance amplifiers of the circuit in Fig. 7.20(a).

### EXAMPLE 7.2

**Finding the small-signal parameters of an NMOS amplifier** The amplifier in Fig. 7.16(a) has $V_{GS} = 2$ V, $V_{DD} = 15$ V, and $R_D = 3.5$ kΩ. The NMOS parameters are $V_t = 1$ V, $K_n = 3.25$ mA/V$^2$, and $V_M = 1/\lambda = 100$ V.

(a) Find the DC biasing point $V_{GS}$, $I_D$, and $V_{DS}$.

(b) Find the small-signal transistor model parameters $r_o$ and $g_m$.

(c) Find the small-signal amplifier parameters $R_v$, $A_{vo}$, and $A_{ro}$.

(d) Use PSpice to plot the small-signal AC output voltage for 1-mV sinusoidal input signal at 1 kHz. The NMOS parameters are $K_p = 6.5$ M, $V_{TO} = 1$ V, $L = 1$ U, $W = 1$ U, and LAMBDA = 0.01. Note: PSpice uses $K_p = K_m = 2K_n$.

**SOLUTION**

(a) $K_m = K_p = 6.5 \times 10^{-3}$ for $W = L$. From Eq. (7.8),

$$I_D = K_n(V_{GS} - V_t)^2 = 3.25 \times 10^{-3} \times 2 \times 1 = 3.25 \text{ mA}$$

$$V_{DS} = V_{DD} - R_D I_D = 15 - 3.5 \times 10^3 \times 3.25 \times 10^{-3} = 3.625 \text{ V}$$

(b) $r_o = V_M/I_D = 100/(3.25 \times 10^{-3}) = 30.77 \text{ KΩ}$

$$g_m = 2 \times K_n(V_{GS} - V_t) = 2 \times 3.25 \times 10^{-3} \times 2 \times 1 = 6.5 \text{ mA/V}$$

(c) $R_o = r_o \parallel R_D = 30.77 \text{ kΩ} \parallel 3.5 \text{ k} = 3.143 \text{ kΩ}$

$$G_{mo} = g_m = 6.5 \text{ mA/V}$$

$$A_{vo} = -g_m \times r_o = -6.5 \text{ mA/V} \times 3.143 \text{ kΩ} = -20.426 \text{ V/V}$$

(d) Figure 7.21 shows the PSpice schematic and the PSpice plot for small-signal output voltage is shown in Fig. 7.22. The capacitor $C_2$ blocks the DC and passes the small-signal output, which shows a voltage of $-19.84$; this is close to the calculated value of $-20.42$. 
A MOSFET can be operated as a voltage-controlled switch. Figure 7.23(a) shows the circuit arrangement. A switch should have the characteristic of a low on-state voltage at the maximum current so that the switch is subjected to the minimum power loss. These conditions require that the transistor is operated in the ohmic (or triode) region, as shown in Fig. 7.23(b).

To operate the MOSFET in the ohmic region, the gate-to-source voltage must be sufficient to maintain the drain current. Assuming \( v_I = v_{GS} > V_I \) and \( v_{DS} = v_{GS} - V_I \) at the boundary condition between the triode and saturation regions, we can find the output voltage as given by

\[
\begin{align*}
  v_o &= V_{DD} - R_D i_D = V_{DD} - R_D K_n (v_I - V_I)^2
\end{align*}
\]

(7.38)

**NOTE:** All PSpice results given here are from running the simulation with the schematic (.SCH) files. If you run the simulation with the netlist circuit (.CIR) files, you may get different results because the student’s version of PSpice has a limited number of active devices and models.
which describes the relationship between the input and output voltages as shown in Fig. 7.23(c). Substituting \( V_O = V_{DS} = v_1 - V_t \) in Eq. (7.38), we can determine the input voltage at the boundary condition:

\[
V_{DD} - R_D K_n (v_1 - V_t)^2 = v_1 - V_t
\]  
(7.39)

This can be solved for the input voltage at the boundary condition as given by

\[
v_{I(B)} = \left( \frac{-1 + \sqrt{4K_n R_D V_{DD}}}{2K_n R_D} + V_t \right)
\]  
(7.40)

Therefore, the MOSFET will operate in the ohmic region if \( v_1 > v_{I(B)} \) and in the saturation region if \( v_1 < v_{I(B)} \). For example, if \( V_{DD} = 15 \, \text{V}, R_D = 3.5 \, \text{kW}, V_t = 1 \, \text{V}, \) and \( K_n = 3.25 \, \text{mA/V}^2, \) Eq. (7.40) gives \( v_{I(B)} = 2.104 \, \text{V}. \) The slope of the \( V_O - v_1 \) described by Eq. (7.38) gives the voltage gain in the saturation region as

\[
A_{vo} = \frac{dV_O}{dv_1} = \frac{d}{dv_1} [V_{DD} - R_D K_n (v_1 - V_t)^2] = -2R_D K_n (v_1 - V_t)
\]  
(7.41)

This gives the same result as Eq. (7.36) if we neglect the MOSFET output resistor \( r_o \approx \infty. \) The maximum value of the drain current \( I_{D(max)} \) is specified by the manufacturer data sheet, which limits the minimum value of drain resistance \( R_D. \) Assuming \( V_{DS(sat)} \) is the drain-to-source saturation voltage, we can find the corresponding drain current \( I_{D(sat)} \) as given by

\[
I_{D(sat)} = \frac{V_{DD} - V_{DS(sat)}}{R_D}
\]  
(7.42)

which sets the limits of the drain current \( I_{D(sat)} \leq i_D \leq I_{D(max)} \) and the corresponding drain resistance

\[
R_{D(max)} \leq R_D \leq R_{D(min)}
\]  
(7.43)

### 7.7 DC Biasing of MOSFETs

It is necessary to bias a MOSFET at a stable operating point so that the biasing point does not change significantly with changes in the transistor parameters. Once the gate-to-source voltage \( V_{GS} \) has been set at a specified value, the MOS drain current \( i_D \) is then fixed. The drain-to-source voltage \( V_{DS} \) is dependent on \( i_D. \) Table 7.1 shows the parameters if their values are positive (+) or negative (−) quantities, and transfer characteristics for various types of MOSFETs.

Since the input gate is isolated electrically from the drain or source terminals, the gate voltage \( V_G \) can be set to any specified desired value independently of \( i_D, V_{DS}, \) and \( V_{SR}. \) The drain current depends on the gate-to-source voltage \( V_{GS}, \) which is the difference between the gate voltage \( V_G \) and the source (or substrate) voltage \( V_{SR}. \) That is, \( V_{GS} = V_G - V_{SR}. \) We can also write \( V_{GS} = V_G \) for \( V_{SR} = 0, \) and \( V_{GS} = -V_{SR} \) for \( V_G = 0. \) Therefore, we can bias a MOSFET at a specific \( V_{GS} \) by different biasing arrangements as shown in Fig. 7.24. Although there are many types of biasing circuits, we will consider the following types, which are most commonly used:

- Zero source resistance biasing
- Grounded gate terminal biasing
- Source resistance only biasing
- Source and drain resistance biasing
NOTE In the derivations of the drain currents for these biasing circuits, we will assume that MOSFETs operate in the saturation region and follow the relationship between $i_D$ and $v_{GS}$:

$$i_D = K_n(v_{GS} - V_t)^2$$

In the ohmic (or triode) region, $i_D = K_n[2(v_{GS} - V_t)v_{DS} - v_{DS}^2]$, where $v_{DS} < (v_{GS} - V_t)$ for n-channel and $v_{DS} > (v_{GS} - V_t)$ for p-channel.

In the saturation region, $i_D = K_n(v_{GS} - V_t)^2$, where $v_{DS} = (v_{GS} - V_t)$ for n-channel and $v_{DS} \leq (v_{GS} - V_t)$ for p-channel. Note: $V_p = V_t$.

### 7.7.1 MOSFET Biasing Circuit

The most common biasing circuit, which can implement the four arrangements in Fig. 7.24 if we select appropriate values of $R_1$, $R_2$, $R_D$, $R_{SR}$, $V_{DD}$ and $V_{SS}$, is shown in Fig. 7.25(a). The value of $v_{GS}$ can be adjusted by using a potential divider consisting of $R_1$ and $R_2$ as given by

$$v_{GS} = \frac{R_2V_{DD}}{R_1 + R_2}$$

Using KVL in the gate-to-source loop, and the same drain current $i_D$ flows through the source terminal, we get

$$v_{GS} = v_G - R_{SR}i_D$$

### TABLE 7.1 Biasing conditions of MOSFETs

<table>
<thead>
<tr>
<th>MOSFET</th>
<th>Enhancement MOSFET</th>
<th>Depletion MOSFET</th>
<th>Enhancement MOSFET</th>
<th>Depletion MOSFET</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_n$</td>
<td>$\frac{\mu_n C_{ox} W}{2L}$</td>
<td>$\frac{\mu_n C_{ox} W}{2L}$</td>
<td>$\frac{\mu_n C_{ox} W}{2L}$</td>
<td>$\frac{\mu_n C_{ox} W}{2L}$</td>
</tr>
<tr>
<td>$V_t$ or $V_p$</td>
<td>$+$</td>
<td>$-$</td>
<td>$-$</td>
<td>$+$</td>
</tr>
<tr>
<td>$v_{GS}$</td>
<td>$&gt; V_t$</td>
<td>$&gt; V_p$</td>
<td>$&lt; V_t$</td>
<td>$&lt; V_p$</td>
</tr>
<tr>
<td>$v_{DS}$</td>
<td>$+$</td>
<td>$+$</td>
<td>$-$</td>
<td>$-$</td>
</tr>
<tr>
<td>$i_D$</td>
<td>$+$</td>
<td>$+$</td>
<td>$-$</td>
<td>$-$</td>
</tr>
<tr>
<td>$V_{DD}$</td>
<td>$+$</td>
<td>$+$</td>
<td>$-$</td>
<td>$-$</td>
</tr>
<tr>
<td>$\lambda = 1/V_M$</td>
<td>$+$</td>
<td>$+$</td>
<td>$-$</td>
<td>$-$</td>
</tr>
</tbody>
</table>

In the ohmic (or triode) region, $i_D = K_n[2(v_{GS} - V_t)v_{DS} - v_{DS}^2]$, where $v_{DS} < (v_{GS} - V_t)$ for n-channel and $v_{DS} > (v_{GS} - V_t)$ for p-channel.

In the saturation region, $i_D = K_n(v_{GS} - V_t)^2$, where $v_{DS} = (v_{GS} - V_t)$ for n-channel and $v_{DS} \leq (v_{GS} - V_t)$ for p-channel. Note: $V_p = V_t$.

FIGURE 7.24 Arrangements for $v_{GS}$ bias
which gives the biasing load line as given by

\[ i_D = \frac{v_G}{R_{SR}} - \frac{v_{GS}}{R_{SR}} \quad (7.46) \]

The intersection of the biasing load line described by Eq. (7.46) with the transfer characteristic in Eq. (7.6) gives the operating point as shown in Fig. 7.25(b). Figure 7.25(b) also describes the input-output relationship, \( v_{GS} \) relates to \( i_D \) which in turn relates to \( v_{DS} \) (output voltage) of the MOSFET.

### 7.7.2 Design of MOSFET Biasing Circuit

Using KVL in the drain and the source loop in Fig. 7.25(a), we can write

\[ v_{DS} = V_{DD} - R_D i_D - R_{SR} i_D = V_{DD} - (R_D + R_{SR}) i_D \quad (7.47) \]

This gives the drain-to-source load line as

\[ i_D = \frac{V_{DD}}{R_D + R_{SR}} - \frac{v_{DS}}{R_D + R_{SR}} \quad (7.48) \]

This is the equation of a straight line and represents the load line, as shown in Fig. 7.26(a). The intersection of the drain-to-source load line described by Eq. (7.48) with the MOS characteristic gives the operating point, defined by \( (V_{GS}, V_{DS}, I_D) \).

The given design parameters are \( K_n, V_t, V_M, \) and \( I_{D\text{(max)}} \) of a MOSFET, along with \( V_{DD} \) and \( V_{SS} \). Select suitable values of the DC biasing drain current \( I_{DQ} \) and the drain-to-source voltage \( V_{DS} \) so that \( i_D \) and \( v_{DS} \) can have the maximum swings in both positive and negative directions: \( i_D = I_D \pm i_{d\text{(peak)}} \) and \( v_{DS} = V_{DS} \pm v_{d\text{(peak)}} \). Otherwise, the small-signal output will be distorted as shown in Fig. 7.26(b). To minimize distortion, \( I_D \) must therefore be less than \( I_{D\text{max}}/2 \), and the DC supply \( V_{DD} \) should be shared equally by all elements in the drain and the source loop. The guidelines for determining the biasing resistances in Fig. 7.25(a) for the different configurations shown in Fig. 7.24 are given in Table 7.2.
### TABLE 7.2  Guidelines for determining the biasing resistances

<table>
<thead>
<tr>
<th>Biasing Parameters</th>
<th>Figure 7.24(a)</th>
<th>Figure 7.24(b)</th>
<th>Figure 7.24(c)</th>
<th>Figure 7.24(d)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_{DS}$</td>
<td>$\frac{V_{DD}}{2}$</td>
<td>$\frac{V_{DD} + V_{SS}}{3}$</td>
<td>$\frac{V_{DD}}{2}$</td>
<td>$\frac{V_{DD}}{3}$</td>
</tr>
<tr>
<td>$V_{SR}$</td>
<td>0</td>
<td>$\frac{V_{DD} + V_{SS}}{2}$</td>
<td>$\frac{V_{DD}}{2}$</td>
<td>$\frac{V_{DD}}{3}$</td>
</tr>
<tr>
<td>$I_{D}$</td>
<td>$\frac{I_{D\text{max}}}{3}$</td>
<td>$\frac{I_{D\text{max}}}{3}$</td>
<td>$\frac{I_{D\text{max}}}{3}$</td>
<td>$\frac{I_{D\text{max}}}{3}$</td>
</tr>
<tr>
<td>$V_{GS}$</td>
<td>$\pm \sqrt{\frac{I_{D}}{K_n}} + V_t$</td>
<td>$\pm \sqrt{\frac{I_{D}}{K_n}} + V_t$</td>
<td>$\pm \sqrt{\frac{I_{D}}{K_n}} + V_t$</td>
<td>$\pm \sqrt{\frac{I_{D}}{K_n}} + V_t$</td>
</tr>
<tr>
<td>$V_G$</td>
<td>$V_{GS}$</td>
<td>$V_{SR} + V_{GS}$</td>
<td>$V_{SR} + V_{GS}$</td>
<td>$V_{SR} + V_{GS}$</td>
</tr>
<tr>
<td>$R_D$</td>
<td>$\frac{V_{DD}}{2I_D}$</td>
<td>$V_{DD} + V_{SS}$</td>
<td>$\frac{V_{DD}}{3I_D}$</td>
<td>$\frac{V_{DD}}{3I_C}$</td>
</tr>
<tr>
<td>$R_{SR}$</td>
<td>0</td>
<td>$\frac{V_{SS} - V_{GS}}{I_D}$</td>
<td>$\frac{V_{DD}}{2I_D}$</td>
<td>$\frac{V_{DD}}{3I_D}$</td>
</tr>
<tr>
<td>$R_1$</td>
<td>0</td>
<td>$\frac{V_{SS} - V_{GS}}{I_D}$</td>
<td>$\frac{V_{DD}}{2I_D}$</td>
<td>$\frac{V_{DD}}{3I_D}$</td>
</tr>
<tr>
<td>$R_2$</td>
<td>$\frac{V_{DD}}{V_G - 1}$</td>
<td>$\frac{V_{DD}}{3I_D}$</td>
<td>$\frac{V_{DD}}{3I_D}$</td>
<td>$\frac{V_{DD}}{3I_D}$</td>
</tr>
</tbody>
</table>

### EXAMPLE 7.3

Designing a biasing circuit for an NMOS amplifier

(a) Design the biasing circuit shown in Fig. 7.25(a) for an NMOS. The DC supply voltage is $V_{DD} = 15$ V. The NMOS parameters are $V_t = 1$ V, $K_n = 3.25$ mA/$V^2$, $I_{D\text{max}} = 10$ mA, and $|V_M| = 1/\lambda = 100$ V.
(b) Calculate the small-signal parameters of the NMOS: \( g_m \) and \( r_o \).
(c) Use SPICE to verify your design. Assume \( K_p = 2K_n \) for \( W = L = 1 \) \( \mu \)m.

**SOLUTION**

(a) Let us assume that \( v_{DS} = V_{DD}/3 = 15/5 = 5 \) V, and \( i_D = I_{D(\text{max})}/3 = 10 \) mA/3 = 3.33 mA. \( R_D = R_{SR} = v_{DS}/i_D = 5/(3.33 \times 10^{-3}) = 1.5 \) kΩ.

Substituting \( i_D = 3.33 \) mA and \( K_n = 3.25 \) mA/V² into Eq. (7.10), \( i_D = K_n (v_{GS} - V_t)^2 \), gives \( v_{GS} = 2.013 \) V or \(-0.013 \) V. For the NMOS, \( v_{GS} \) must be greater than 1 V. Thus, the acceptable value is \( v_{GS} = 2.013 \) V. Using KVL around the gate-to-source loop,

\[
r_G = v_{SR} + v_{GS} = 5 + 2.013 = 7.013 \text{ V}
\]

\[
R_1 \approx R_2 = \left( \frac{V_{DD}}{V_G} \right) - 1 = \frac{15}{7.013} - 1 = 1.139
\]

Letting \( R_2 = 100 \) kΩ, we get \( R_1 \approx 1.139R_2 = 1.139 \times 100 = 113.9 \) kΩ.

(b) From Eq. (7.27), \( g_m = 2 \times 3.25 \times 3 \times (2.013 - 1) = 6.583 \) mA/V. From Eq. (7.26), \( r_o = 1/\lambda_0 = 100/3.33 = 30 \) kΩ.

(c) The details of the DC bias calculations by the PSpice simulation are given here:

\[
\begin{align*}
ID & = 3.35E-03 \text{ (3.33 mA)} \quad VGS = 1.99E+00 \text{ (2 V)} \quad VDS = 4.96E+00 \text{ (5 V)} \\
GM & = 6.76E-03 \text{ (6.583 mA/V)} \quad GDS = 3.19E-05 \text{ (1/r_o = 1/30 k = 33.3 \( \mu \)A/V).}
\end{align*}
\]

**NOTE:** The values obtained by hand calculations are shown in parentheses.

**EXAMPLE 7.4**

**Designing a biasing circuit for a depletion NMOS amplifier**

(a) Design a biasing circuit as shown in Fig. 7.25(a) for an \( n \)-channel depletion MOSFET. The DC supply voltage is \( V_{DD} = 15 \) V. The parameters are \( I_{DSS} = 12.65 \) mA and \( V_p = -3.5 \) V. Assume operation in the saturation region.

(b) Calculate the small-signal parameters \( g_m \) and \( r_o \) of the transistor.

(c) Use PSpice/SPICE to verify your design.

**SOLUTION**

(a) To accommodate the maximum AC swing and the variations in depletion MOS parameters, the following conditions as listed in Table 7.2 are recommended for biasing for the \( Q \)-point \( (I_D, V_{DS}) \):

\[
i_D = \frac{I_{DSS}}{2}
\]

\[
v_{DS} = \frac{V_{DD}}{3}
\]
That is,

\[
i_D = \frac{I_{DSS}}{2} = \frac{12.65 \text{ mA}}{2} = 6.3 \text{ mA} \quad \text{and} \quad v_{DS} = \frac{V_{DD}}{3} = \frac{15}{3} = 5 \text{ V}
\]

Substituting \(K_n = I_{DSS}/V_p^2\) in Eq. (7.21),

\[
6.3 \text{ mA} = 12.65 \text{ mA} \times \left(1 + \frac{v_{GS}}{3.5}\right)^2 \quad \text{or} \quad \left(1 + \frac{v_{GS}}{3.5}\right)^2 = \pm 0.706
\]

which gives \(v_{GS} = -1.03 \text{ V or } -5.97 \text{ V. Since } v_{GS} > V_p (= -3.5 \text{ V}), the operational value of } v_{GS} \text{ is } -1.03 \text{ V.}

Since \(v_{GS}\) of a depletion MOSFET is negative \((-1.03 \text{ V}), we do not need the biasing resistance \(R_1(= \infty)\) because \(R_{SR}\) will cause a voltage drop of \(v_{GS} = -R_{SR}/i_D\). This arrangement is known as self-biasing of the depletion MOSFET as shown in Fig. 7.24(b).

We find that

\[
R_{SR} = \frac{-v_{GS}}{i_D} = \frac{-1.03 \text{ V}}{6.3 \text{ mA}} = 163.5 \Omega
\]

and its power rating is

\[
P_{R_{SR}} = (6.3 \times 10^{-3} \text{ A})^2 \times 163.5 \Omega = 6.49 \text{ mW}
\]

Since \(R_{D\text{SD}} = V_{DD} - R_{SR}/i_D - v_{DS} = 15 - 1.03 - 5 = 8.97 \Omega,

\[
R_D = \frac{8.97 \text{ V}}{6.3 \text{ mA}} = 1424 \Omega
\]

and its power rating is

\[
P_{R_D} = (6.3 \times 10^{-3} \text{ A})^2 \times 1424 \Omega = 56.52 \text{ mW}
\]

Since one side of \(R_2\) is connected to the ground and the gate–source junction is like a reverse-biased diode, the DC current flowing through \(R_2\) is very small, tending to zero. \(R_2\) provides continuity of the circuit for the gate–source biasing voltage. In selecting the value of \(R_2\), it is important to keep two things in mind: (1) \(R_2\) should match the reverse-bias resistance of the gate–source junction, and (2) \(R_2\) will carry current when an AC signal is applied to the gate terminal. A value of \(R_2\) between 50 k\(\Omega\) and 500 k\(\Omega\) is generally suitable. Let \(R_2 = 500 \text{ k}\(\Omega\).

(b) \(K_n = I_{DSS}/V_p^2 = 12.65 \text{ mA}/(3.5 \text{ V})^2 = 1.033 \text{ mA/V}^2\). From Eq. (7.30),

\[
g_m = 2K_n(v_{GS} - V_p) = 2 \times 1.033 \text{ m} \times (-1.03 + 3.5) = 5.10 \text{ mA/V}
\]

From Eq. (7.26),

\[
r_o = \frac{1}{\lambda i_D} = \frac{1}{5.929 \text{ m} \times 6.3 \text{ mA}} = 26.77 \text{ k}\(\Omega\)
\]

(c) The biasing circuit for PSpice simulation is shown in Fig. 7.27. For PSpice simulation, we use \(K_p = 2 \times K_n = 2 \times 1.033 = 2.066 \text{ mA/V}^2\) for \(W = L = 1 \mu\text{m}\) and \(V_{to} = V_p = -3.5 \text{ V}. The details of the DC bias calculations by the PSpice simulation are given here:

\[
\begin{align*}
\text{ID} &= 6.46E-03 (6.3 \text{ mA}) & \text{VGS} &= -1.06E+00 (-1.03 \text{ V}) & \text{VDS} &= 4.74E+00 (5 \text{ V}) \\
\text{GM} &= 5.29E-03 (5.10 \text{ mA/V}) & \text{GDS} &= 6.17E-05 (1/r_o = 1/15.81 \text{ k} = 63.25 \mu\text{A/V})
\end{align*}
\]
Design for limiting the drain current variation of an NMOS amplifier

Design a biasing circuit as shown in Fig. 7.25(a) for an NMOS for which $V_t$ varies from 1 V to 1.5 V and $K_n$ varies from $150 \mu A/V^2$ to $100 \mu A/V^2$. Limiting the variation in the drain current to $5 mA \pm 20\%$, calculate the values of $R_{SR}$, $R_1$, $R_2$, and $R_D$. Assume $V_{DD} = 15 V$.

**SOLUTION**

$V_{t1} = 1 V$, $V_{t2} = 1.5 V$, $K_{n1} = 150 \mu A/V^2$, and $K_{n2} = 100 \mu A/V^2$. The two possible transfer characteristics that can result from the variations in the parameters are shown in Fig. 7.28. Using Eq. (7.10), we can describe these characteristics as follows:

$I_{D1} = K_{n1}(V_{GS1} - V_{t1})^2 = 150 \times 10^{-6} \times (V_{GS1} - 1)^2$

$I_{D2} = K_{n2}(V_{GS2} - V_{t2})^2 = 100 \times 10^{-6} \times (V_{GS2} - 1.5)^2$

For a drain current variation of $I_{D1} = 5 mA + 20\% = 5 mA \times (1 + 0.2) = 6 mA$, we have

$6 mA = 150 \mu A \times (V_{GS1} - 1)^2$

which gives an operating value of $V_{GS1} = 7.32 V$.

For a drain current variation of $I_{D2} = 5 mA - 20\% = 5 mA \times (1 - 0.2) = 4 mA$, we have

$4 mA = 100 \mu A \times (V_{GS2} - 1.5)^2$
7.8 Common-Source (CS) Amplifiers

Figure 7.16(a) is an example of common-source (CS) amplifiers where the source terminal is common to both input and output terminals. Equation (7.36) gives the voltage gain as

$$A_v = -g_m R_D$$

for $r_o \gg R_D$. The resistive biasing design limits the value of $R_D$ and the voltage gain. Replacing $R_D$ with an active current source that has an inherent high output resistance can increase the voltage gain significantly. Any resistance in the source terminal reduces the effective small-signal voltage gain, and we will evaluate the effect in the voltage gain. We will consider CS amplifiers with four types of load: (a) active current source load, (b) enhancement MOSFET load, (c) depletion MOSFET load, and (d) resistive load.

![Figure 7.28](image)

**FIGURE 7.28** Two transfer characteristics

which gives an operating value of $V_{GS2} = 7.825$ V.

The slope of the biasing load line gives the value of $R_{SR}$:

$$R_{SR} = \frac{V_{GS2} - V_{GS1}}{I_{D1} - I_{D2}} = \frac{7.825 - 7.32}{6 - 4} \times 10^3 = 252.5 \, \Omega$$

Applying Eq. (7.45) at the $Q$-point characteristic with $V_{GS2}$ and $I_{D2}$ gives

$$V_G = V_{GS2} + I_{D2}R_{SR} = 7.825 + 4 \times 10^{-3} \times 252.5 = 8.835 \, V$$

The values of $R_1$ and $R_2$ can be found from

$$V_G = \frac{R_2V_{DD}}{R_1 + R_2} = \frac{R_2 \times 15}{R_1 + R_2} = 8.835 \, V$$

which gives $(1 + R_1/R_2) = 1.7$. Choose a suitable value of $R_2$, usually larger than 500 kΩ. Assuming $R_2 = 500 \, k\Omega$, $R_1 = 350 \, k\Omega$.

### KEY POINT OF SECTION 7.7

- A MOSFET should be biased properly in order to activate the device and also to establish a DC operating point such that a small variation in the gate-to-source voltage causes a variation in the drain current. Like any amplifier, a MOSFET amplifier can be used as a buffer stage to offer a low output resistance and a high input resistance.

**7.8 Common-Source (CS) Amplifiers**

Figure 7.16(a) is an example of common-source (CS) amplifiers where the source terminal is common to both input and output terminals. Equation (7.36) gives the voltage gain as $A_v = -g_m(r_o || R_D) \approx -g_mR_D$ for $r_o \gg R_D$. The resistive biasing design limits the value of $R_D$ and the voltage gain. Replacing $R_D$ with an active current source that has an inherent high output resistance can increase the voltage gain significantly. Any resistance in the source terminal reduces the effective small-signal voltage gain, and we will evaluate the effect in the voltage gain. We will consider CS amplifiers with four types of load: (a) active current source load, (b) enhancement MOSFET load, (c) depletion MOSFET load, and (d) resistive load.
A simple MOSFET amplifier with a current source is shown in Fig. 7.29(a). If \( r_{o1} \) and \( r_{o2} \) are the output resistances of the transistor \( M_1 \) and the current source \( I_O \), respectively, we can find the small-signal voltage gain from Eq. (7.36) as given by

\[
A_{vO} = \frac{v_o}{v_{gs}} = -g_m(r_{o1} \parallel r_{o2})
\]  

(7.51)

A basic MOSFET current source is shown in Fig. 7.29(b), which can be represented with a sinking current source \( I_O \) with an output resistance as shown in Fig. 7.29(b): \( r_{o2} = V_{M2}/I_{D2} \). Let us assume that the two transistors \( M_2 \) and \( M_3 \) are identical. Since their gate-to-source voltages are equal, their drain currents will be the same. That is, \( I_{D2} = I_{D3} \). Thus the output current \( I_O (=I_{D2}) \) will be the mirror of \( I_{D3} \). Since \( V_{DS3} = V_{GS3} \), which is greater than or equal to \( (V_{GS3} - V_{t3}) \), \( M_3 \) will be in saturation. Let \( V_{t2} \) and \( V_{t3} \) be the threshold voltages of \( M_2 \) and \( M_3 \), respectively. For \( M_2 \) also to be in saturation, \( V_{DS2} \) must be greater than \( (V_{GS2} - V_{t2}) \). This condition reduces the voltage compliance range of the MOSFET current source and prevents it from operating from a low power supply (say, 1 V for a battery source).

**NOTE**  
\( V_t \) is the threshold voltage of a MOSFET, whereas \( V_T \) is the thermal voltage.

The output current \( I_O \), which is equal to the drain current of \( M_2 \), is given by

\[
I_{D2} = I_O = K_n2(V_{GS2} - V_{t2})^2(1 + \lambda V_{DS2})
\]  

(7.52)

Drain current \( I_{D2} \), which is equal to the reference current \( I_R \), is given by

\[
I_{D3} = I_R = K_n3(V_{GS3} - V_{t3})^2(1 + \lambda V_{DS3})
\]  

(7.53)
In practice, all the components of the current source are processed on the same integrated circuit, and hence all of the physical parameters such as $K_n$ and $V_t$ are identical for both devices. Thus the ratio of $I_O$ to $I_R$ is given by

$$\frac{I_O}{I_R} = \frac{K_{n1}(1 + \lambda V_{DS2})}{K_{n2}(1 + \lambda V_{DS3})} = \frac{(W/L)_2}{(W/L)_3} \times \frac{(1 + \lambda V_{DS2})}{(1 + \lambda V_{DS3})}$$  \hspace{1cm} (7.54)

In practice, $\lambda V_{DS} << 1$. Thus Eq. (7.54) can be approximated by

$$\frac{I_O}{I_R} = \frac{(W/L)_2}{(W/L)_3}$$  \hspace{1cm} (7.55)

By controlling the ratio $(W/L)_m$ of $m$th MOSFET, therefore, we can change the output current $I_O$. The gate length $L$ is usually held fixed, and the gate width $W$ is varied from device to device to give the desired current ratio $I_O/I_R$. By choosing identical transistors with $W_2 = W_3$ and $L_2 = L_3$, a designer can ensure that the output current $I_O$ is almost equal to the reference current $I_R$.

Since $V_{GS3} = V_{DD} - R_{ref} I_R$ and $V_{DS3} = V_{GS3}$, the reference current $I_R$ can be found approximately from Eq. (7.53). That is,

$$I_R = I_{D3} = K_{n3}(V_{DD} - R_{ref} I_R - V_{(3)})^2$$  \hspace{1cm} (7.56)

can be solved for known values of $V_{(3)}$, $K_{n3}$, $V_{DD}$, and $R_{ref}$.

Replacing $I_O$ in Fig. 7.29(a) with the sourcing type of current source consisting of PMOS is shown in Fig. 7.29(c). This is accomplished by replacing $M_2$ and $M_3$ in Fig. 7.29(b) with PMOS. The small-signal equivalent circuit of Fig. 7.29(c) is shown in Fig. 7.29(d), from which we can find the small-signal voltage gain as given by Eq. (7.51). For identical transistors $r_o = r_{o1} = r_{o2}$, Eq. (7.51) becomes

$$A_{vo} = -g_m(r_{o1}/r_{o2}) = -g_m \frac{r_0}{2} = 2K_n(V_{GS} - V_t) \frac{1}{2\lambda I_D}$$

$$= \frac{2K_n(V_{GS} - V_t)}{2\lambda K_n(V_{GS} - V_t)^2} = \frac{1}{\lambda(V_{GS} - V_t)}$$  \hspace{1cm} (7.57)

Therefore, we can conclude that the voltage gain $A_{vo}$ is inversely proportional to $\lambda$, the biasing current $I_D$, and the gate-to-source voltage $V_{GS}$.

The reference resistance $R_1$ in Fig. 7.29(c) can be replaced by one or more PMOS, as shown in Fig. 7.29(e). Transistors $M_3$, ..., $M_n$ are used as voltage dividers to control the gate-to-source voltage of transistor $M_2$. If there are $n$ cascode-connected PMOS, and all have identical characteristics, the gate-to-source voltage of the PMOS $M_2$ is given by

$$v_{GS2} = v_{GS3} = \frac{-V_{DD}}{n}$$  \hspace{1cm} (7.58)

This gives the reference drain current $i_D$ as

$$i_D = K_{n2}(v_{GS2} - V_{(P)})^2 = K_{n2}\left(\frac{-V_{DD}}{n} - V_{(P)}\right)^2$$  \hspace{1cm} (7.59)

Therefore, we can find the integer number of MOSFETs to obtain a specific reference current $I_D$ or gate-to-source voltage $V_{GS}$. For example, if $V_{DD} = 12$ V, we need six PMOS MOSFETs to get $V_{SG} = 2$ V. We should note that $V_{(P)}$ in Eq. (7.59), which is the threshold voltage of a PMOS, has a negative value.
EXAMPLE 7.6

**Design of a CS amplifier with a MOS current source**

(a) Find the value of $R_1$ to obtain a biasing current of $I_D = 3.25$ mA and the small-signal voltage gain of the CS amplifier in Fig. 7.29(c). The DC supply voltage is $V_{DD} = 15$ V. The MOS parameters are $V_{IN} = -V_{IP} = 1$ V, $K_n = 3.25$ mA/V$^2$, $K_p = 6.5$ mA/V$^2$, for $W = L = 1$ $\mu$m and $|V_M| = 1/\lambda = 100$ V.

(b) Use SPICE to plot the small-signal output voltage $v_o$ for a sinusoidal input signal $v_s$ of 1 mV at 1 kHz.

**SOLUTION**

(a) For $I_D = 3.25$ mA, Eq. (7.8) gives $3.25$ mA $= 3.25$ m $\times (V_{GS3} - 1)^2$, which gives $V_{GS3} = -2$ V.

Applying KVL through the loop via $V_{DD}$, $R_1$, and $V_{GS3}$, we can find the values of

$$R_{ref} = \frac{V_{DD} + V_{GS3}}{I_D} = \frac{15 - 2}{3.25} = 4 \text{ k}\Omega$$

$$r_o = r_{o1} = r_{o2} = \frac{V_M}{I_D} = \frac{100}{3.25} = 30.77 \text{ k}\Omega$$

$$g_m = 2K_n(V_{GS3} - V_{IN}) = 2 \times 3.25 \times (2 - 1) = 6.5 \text{ mA/V}$$

From Eq. (7.57), the voltage gain $A_{vo} = -g_m r_o / 2 = -6.50 \times 30.7 \text{ k}/2 = -100 \text{ V/V}$.

The DC gate voltage of $M_1$ is $V_{G1} = 2$ V, for which we can use a voltage divider as shown in Fig. 7.25(a). Therefore, $R_1/R_2 = V_{DD}/V_{G1} - 1 = 15/2 - 1 = 6.5$.

Let $R_2 = 100 \text{ k}\Omega$; then $R_1 = 6.5 \times 100 \text{ k} = 650 \text{ k}\Omega$.

(b) The PSpice schematic is shown in Fig. 7.30. The plot of the output voltage is shown in Fig. 7.31, which gives a voltage gain of $-101 \text{ V/V}$; this is close to the calculated value of $-100$. Note that there is a phase shift of $180^\circ$. 

**FIGURE 7.30** PSpice schematic for Example 7.6

**FIGURE 7.31** PSpice plot of small-signal output voltage for Example 7.6
A CS amplifier with an NMOS driver and an NMOS active load, as shown in Fig. 7.32(a), is the simplest way of implementing an amplifier with NMOS technology. M2 is diode-connected, and it behaves as a nonlinear resistive load. If the input voltage \( v_G \) is less than the threshold voltage \( V_t \), then M1 is off and no current flows in the circuit. If the input voltage \( v_G \) exceeds the threshold voltage \( V_t \), then M1 is turned on. Both M1 and M2 operate in the saturation region, and the circuit provides amplification. Since \( v_{GS2} = V_{DD} - V_t - V_{DS1} \), the drain current \( i_D \) can be related to the output voltage \( v_O \) by

\[
i_D = K_n^2(v_{GS2} - V_t)^2 = K_n^2(V_{DD} - v_O - V_t)^2
\]

which gives \( i_D = 0 \) at \( v_O = V_{DD} - V_t \) and \( i_D = (V_{DD} - V_t)^2 \) at \( v_O = 0 \). The \( i_D \)-versus-\( v_O \) (=\( v_{DS1} \)) characteristic is superimposed on the output characteristics of M1 in Fig. 7.32(b), and the intersection of the two characteristics gives the operating point defined by \( I_D \) and \( V_{DS1} \).

Replacing the transistors in Fig. 7.32(a) with their small-signal models gives the AC equivalent circuit shown in Fig. 7.32(c). Summing currents at the output node, we get

\[
-g_{m2}v_o - \frac{v_o}{r_{o2}} - g_{m1}v_{gs1} - \frac{v_o}{r_{o1}} = 0
\]

which gives the open-circuit voltage gain as

\[
A_{vo} = \frac{v_o}{v_{gs1}} = \frac{-g_{m1}}{g_{m2} + 1/r_{o1} + 1/r_{o2}}
\]

The equivalent output resistance can easily be shown to be

\[
R_o = r_{o1} \parallel r_{o2} \left( \frac{1}{g_{m2}} \right)
\]

For \( g_{m2} \gg 1/r_{o1} \) and \( 1/r_{o2} \), which is generally true, Eq. (7.61) can be approximated by

\[
A_{vo} = -\frac{g_{m1}}{g_{m2}} = -\left[ \frac{K_n1}{K_n2} \right]^{1/2} = -\left[ \frac{W_1/L_1}{W_2/L_2} \right]^{1/2}
\]
Because of the practical limitations of device geometries, the maximum voltage gain is in the range of 10 to 20. However, the small-signal voltage gain is independent of the DC operating point, and this amplifier gives a linear amplification over a broad band. For example, if \( W_1 = 100 \ \mu m, L_1 = 5 \ \mu m, W_2 = 5 \ \mu m, \) and \( L_2 = 25 \ \mu m, \) Eq. (7.62) gives \(|A_{vo}| = 10\). It is worth noting that the load device \( M_2 \) will remain in the saturated mode of operation as long as the output voltage \( v_O \geq V_{DD} - V_t \). Otherwise, the transistor will be in the cutoff region and will carry no current.

### 7.8.3 CS Amplifier with Depletion MOSFET Load

A depletion MOSFET can behave as a current source when the gate and source are shorted together, and it can be fabricated on the same IC chip as an enhancement MOSFET. This load device exhibits a very high output resistance as long as the device is operated in the saturation region. Therefore, to provide the large resistance required of a load for high voltage gain, a depletion MOSFET must be operated in the saturation region. A CS amplifier with an NMOS driver and a depletion active load is shown in Fig. 7.33(a). \( M_2 \) is diode-connected, and it behaves as a nonlinear resistive load. If the input voltage \( v_G \) is less than the threshold voltage \( V_t \), then \( M_1 \) is off and no current flows in the circuit. If the input voltage \( v_G \) exceeds the threshold voltage \( V_t \), then \( M_1 \) is turned on. Both \( M_1 \) and \( M_2 \) operate in the saturation region, and the circuit provides amplification. Since \( v_{GS2} = 0 \) and \( v_O = V_{DD} - v_{DS2} \), the drain current \( i_D \) can be determined from

\[
i_D = K_n2(v_{GS2} - V_t)^2 = K_n2(-V_t)^2 = K_n2V_t^2
\]

The \( i_D \)-versus-\( v_O \) \((=V_{DD} - v_{DS2})\) characteristic is superimposed on the output characteristics of \( M_1 \) in Fig. 7.33(b), and the intersection of the two characteristics gives the operating point defined by \( I_D \) and \( V_{DS1} \).

The AC equivalent circuit of the amplifier in Fig. 7.33(a) is shown in Fig. 7.33(c), from which we can find the open-circuit voltage gain

\[
A_{vo} = -g_{m1}(r_o1 \parallel r_o2) = -g_{m1}R_o
\]

where \( R_o = (r_o1 \parallel r_o2) \).

![Diagram of CS amplifier with depletion load](image-url)
7.8.4 CS Amplifier with Resistive Load

A CS amplifier with resistive biasing is shown in Fig. 7.34(a). The amplifying device can be any type of MOSFET. The source resistance \( R_{SR} \) is split into two source resistors, \( R_{SR1} \) and \( R_{SR2} \). \( R_{SR2} \) is shunted by a large capacitance \( C_S \). Both \( R_{SR1} \) and \( R_{SR2} \) set the DC bias point, while \( R_{SR2} \) is effectively shorted for a small signal and gives the desired small-signal voltage gain. Let us assume that the coupling capacitors \( C_1 \), \( C_2 \), and \( C_S \) have high values so that they behave as short-circuited at the frequency of interest. Load resistance \( R_L \) is considered external to the amplifier and is not included. The DC biasing circuit is the same as the biasing circuit in Fig. 7.25(a).

The small-signal AC equivalent circuit of the amplifier is shown in Fig. 7.34(b). We could use either the small-signal Norton’s equivalent model in Fig. 7.18(a) or Thevenin’s equivalent model in Fig. 7.18(b). Due to the presence of \( R_{SR1} \) in the source branch, the analysis becomes simpler with the Thevenin’s equivalent model. If we ignore the output resistance \( r_o \), which we can generally do in most cases with reasonable accuracy, then the use of Norton’s model is recommended. To obtain accurate results, we will replace the MOSFET by its small-signal model of Fig. 7.18(b); the amplifier circuit is shown in Fig. 7.34(c), which can be represented by an equivalent voltage amplifier as shown in Fig. 7.35(a) or by an equivalent transconductance amplifier as shown in Fig. 7.35(b).

![Diagram of CS Amplifier with Resistive Load](image-url)
The DC analysis of a MOSFET amplifier must be performed prior to the small-signal analysis because the small-signal parameters depend on the DC operating point. The steps that are normally required to analyze a MOSFET amplifier are as follows:

**Step 1.** Draw the circuit diagram of the amplifier to be analyzed.

**Step 2.** Mark terminals G, D, and S for each MOSFET on the diagram. Locating these points is the beginning of drawing the equivalent circuit.

**Step 3.** Replace each MOSFET by its Thevenin (or Norton) model.

**Step 4.** Draw the other elements of the amplifier, keeping the original relative position of each element.

**Step 5.** Replace each DC voltage by its internal resistance. An ideal DC source should be replaced by a short circuit.

**Input Resistance** $R_i$ ($= v_g / i_g$)

The input resistance $R_i$ of the amplifier in Fig. 7.34(c) can be found from

$$R_i = \frac{v_g}{i_g} = R_G$$

(7.64)

The total input resistance $R_{in}$ seen by the input signal $v_s$ is

$$R_{in} = \frac{v_s}{i_s} = R_i + R_s$$

where $R_s$ is the input resistance of the input signal source.

**Output Resistance** $R_o$

The output resistance $R_o$ can be obtained by setting $v_s$ equal to zero and then applying a test voltage $v_x$ at the output side. This arrangement is shown in Fig. 7.36. Applying KVL around the gate, input, and source terminals (loop II) gives

$$v_{gs} = v_g - i_d R_{SR1} = -i_d R_{SR1}$$
Applying KVL around the drain, source, and test voltage source (loop I) gives

\[ v_x = i_d r_o - \mu_g v_{gs} + i_d R_{SR1} = i_d r_o + \mu_g i_d R_{SR1} + i_d R_{SR1} = i_d r_o + (1 + \mu_g) R_{SR1} i_d \]

which yields

\[ i_d = \frac{v_x}{r_o + (1 + \mu_g) R_{SR1}} \]

The test current \( i_x \) is given by

\[ i_x = i_d + \frac{v_x}{R_D} = \frac{v_x}{r_o + (1 + \mu_g) R_{SR1}} + \frac{v_x}{R_D} \]

which gives the output resistance \( R_o \) as

\[ R_o = \frac{v_x}{i_x} = \frac{v_x}{i_x} \left[ r_o + (1 + \mu_g) R_{SR1} \right] \| R_D \quad (7.65) \]

\[ R_{out} = R_o \| R_L \quad (7.66) \]

**Open-Circuit (or No-Load) Voltage Gain \( A_{vo} (= v_o/v_g) \)**

By applying KVL around the loop formed by \( r_o, R_D, \) and the voltage-controlled voltage source in Fig. 7.34(c), we get

\[ \mu_g v_{gs} = R_{SR1} i_d + R_D i_d + r_o i_d \quad (7.67) \]

Substituting \( v_{gs} = v_g - R_{SR1} i_d \) into Eq. (7.67) gives the drain current \( i_d \) as

\[ i_d = \frac{\mu_g v_g}{R_D + r_o + (1 + \mu_g) R_{SR1}} \quad (7.68) \]

The output voltage \( v_o \) can be found from

\[ v_o = -R_D i_d \quad (7.69) \]

Substituting \( i_d \) from Eq. (7.68) into Eq. (7.69) gives the open-circuit voltage gain \( A_{vo} \) as

\[ A_{vo} = \frac{v_o}{v_g} = \frac{-\mu_g R_D}{R_D + r_o + (1 + \mu_g) R_{SR1}} = -\frac{g_m r_o R_D}{R_D + r_o + (1 + g_m r_o) R_{SR1}} \quad (7.70) \]
which indicates that the resistance $R_{SR}$ of the source terminal has an effect $(1 + g_m r_o) R_{SR1}$ and reduces the open-circuit voltage gain $A_{vo}$ significantly. The voltage gain $A_{vo}$ can be made large (a) by making $R_{SR1} = 0$, (b) by using a MOSFET with a large value of $g_m$, and (c) by choosing a high value of $R_D$. For $R_{SR1} = 0$, Eq. (7.70) gives the maximum open-circuit voltage gain as

$$A_{vo(max)} = -\frac{-\mu_g R_D}{R_D + r_o} = \frac{-g_m r_o R_D}{R_D + r_o} = \frac{-g_m R_D}{1 + R_D/r_o} \quad (7.71)$$

For $r_o >> R_D$, which is generally the case with a resistive biasing circuit, Eq. (7.71) gives $A_{vo(max)} \approx -g_m R_D$.

**EXAMPLE 7.7**

**Designing an NMOS amplifier to give a specified voltage gain**

(a) Design an NMOS amplifier as shown in Fig. 7.34(a) to give a no-load voltage gain of $|A_{vo}| = v_o/v_i \geq 5$. The DC supply voltage is $V_{DD} = 15$ V. The NMOS parameters are $V_T = 1$ V, $K_n = 3.25$ mA/V$^2$, $K_p = 2K_n = 6.5$ mA/V$^2$ for $W = L = 1 \mu$m, $I_{D(max)} = 10$ mA, and $|V_M| = 1/\lambda = 100$ V.

(b) Use PSpice/SPICE to verify your results in part (a).

**SOLUTION**

(a) Step 1. Design the biasing circuit. The results of Example 7.3 give $R_D = 1.5$ k$\Omega$, $R_{SR} = 1.5$ k$\Omega$, $R_1 = 650$ k$\Omega$, and $R_2 = 100$ k$\Omega$.

Step 2. Find the small-signal parameters of the transistor. The results of Example 7.3 give $g_m = 6.583$ mA/V and $r_o = 30$ k$\Omega$.

Step 3. Find the values of $C_1$, $C_2$, $C_S$, $R_{SR1}$, and $R_{SR2}$. Let us choose $C_1 = C_2 = C_S = 10 \mu$F. The worst-case maximum possible gain that we can obtain from the transistor operating at $i_D = 6$ mA can be found from Eq. (7.71):

$$|A_{vo(max)}| = \frac{g_m R_D}{1 + R_D/r_o} = \frac{1.5 \times 6.583 \text{ mA}}{1 + 1.5 \times 30 \text{ k}$\Omega$} = 9.404 \text{ V/V}$$

The desired gain is less than the maximum possible value, and we can proceed with the design. Otherwise we would need to choose another transistor with a higher value of $g_m$. The value of unbypassed emitter resistance $R_{SR1}$ in Fig. 7.34(a) can be found from Eq. (7.70). That is,

$$R_D + r_o + (1 + \mu_g) R_{SR1} = \frac{\mu_g R_D}{|A_{vo}|} \quad (7.72)$$

which, for $|A_{vo}| = 5$, $R_D = 1.5$ k$\Omega$, $r_o = 30$ k$\Omega$, and $\mu_g = r_o g_m = 197.48$ V/V, gives $R_{SR1} = 139.79 \Omega$ and $R_{SR2} = R_{SR} - R_{SR1} = 1.5$ k$\Omega - 130.79 \Omega = 1.3$ k$\Omega$.

(b) The PSpice schematic is shown in Fig. 7.37. The plot of the output voltage is shown in Fig. 7.38, which gives a voltage gain of $-5.11$ V/V. This is close to the calculated value of $-5$, but it is much too low in comparison to $-100$ with an NMOS amplifier with current source biasing (in Example 7.6). Note that there is a phase shift of $180^\circ$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
FIGURE 7.37  PSpice simulation of a CS amplifier with a biasing resistive load for Example 7.7

FIGURE 7.38  PSpice plot of small-signal output voltage for Example 7.7

TABLE 7.3  Summary of expressions for MOSFET’s amplifiers

<table>
<thead>
<tr>
<th></th>
<th>CS Amplifier [Fig. 7.34(a)]</th>
<th>CD Amplifier [Fig. 7.42(a)]</th>
<th>CG Amplifier [Fig. 7.43(a)]</th>
<th>CS Amplifier with Active Load [Fig. 7.29(a)]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_1$ (Ω)</td>
<td>$R_G$</td>
<td>$R_G$</td>
<td>$R_{SR} \parallel \left( \frac{r_o + R_D}{1 + g_m r_o} \right)$</td>
<td>$\infty$</td>
</tr>
<tr>
<td>$R_o$ (Ω)</td>
<td>$R_D$</td>
<td>$\frac{r_o}{1 + g_m r_o} \parallel R_{SR}$</td>
<td>$R_D$</td>
<td>$r_{o2} \parallel r_{o1}$</td>
</tr>
<tr>
<td>$A_{vo}$ (V/V)</td>
<td>$\frac{-g_m r_o R_D}{R_D + r_o + (1 + g_m r_o) R_{SR1}}$</td>
<td>$\frac{g_m(r_o \parallel R_{SR})}{1 + g_m r_o \parallel R_{SR}}$</td>
<td>$\frac{R_D(1 + g_m r_o)}{r_o + R_D}$</td>
<td>$-g_m (r_{o2} \parallel r_{o1})$</td>
</tr>
</tbody>
</table>
7.9 Common-Drain Amplifiers

A general common-drain configuration is shown in Fig. 7.39(a). A common-drain amplifier has a very high input resistance and draws a very small gate current. It also offers a low output resistance and can be used as a buffer stage between a low resistance load (requiring a high current) and a signal source that can supply only a very small current. This configuration has a voltage gain approaching unity and is known as a source follower. We can derive an input and output relationship if we assume \( v_G \) is the input gate voltage and \( v_O \) is the output voltage at the source terminal. The gate-to-source \( v_{GS} \), which controls the drain current, is given by

\[
v_{GS} = v_G - v_O
\]

The corresponding drain current, which must also flow through the source resistance \( R_{SR} \), is as shown in biasing circuit in Fig. 7.24(b).

\[
i_D = K_n(v_{GS} - V_t)^2 = K_n(v_G - v_O - V_t)^2 = \frac{v_O}{R_{SR}}
\]

which we can solve to find the output voltage for a specific value of \( v_G \):

\[
v_O(v_G) = \frac{2K(v_G - V_t) + 1 - \sqrt{[2K(v_G - V_t) + 1]^2 - 4K^2(v_G - V_t)^2}}{2K}
\]

for \( v_{GS} = (v_G - v_O) > V_t \)

---

**FIGURE 7.39** Source follower with current source load

---

**KEY POINTS OF SECTION 7.8**

- The expressions for the input resistance \( R_i \), the output resistance \( R_o \), and the no-load voltage gain \( A_{vo} \) are summarized in Table 7.3.
- MOSFETs are commonly used in IC technology, operated with a MOS current source, a PMOS active load, or an NMOS active load.
Here $K = K_n R_{SR}$. For example, if $K_n = 3.25 \text{ mA/V}^2$, $R_{SR} = 23.1 \text{ k\Omega}$, and $V_t = 1 \text{ V}$, we get $v_O(2) = 0.695 \text{ V}$ and $v_O(3) = 1.546 \text{ V}$. This gives the voltage gain as

$$A_{VO} = \frac{\Delta v_O}{\Delta v_{GS}} = \left[ \frac{v_O(3) - v_O(2)}{v_{GS}(3) - v_{GS}(2)} \right] = \frac{(1.546 - 0.695)}{1} = 0.851 \text{ V/V}$$

The value of $R_{SR}$ should be large because the voltage gain becomes closer to unity as $R_{SR}$ increases to very large, tending to infinity.

### 7.9.1 Active-Biased Source Follower

The source resistance in Fig. 7.24(c) can be replaced by a sinking current source as shown in Fig. 7.29(b). The simplified circuit is shown in Fig. 7.39(a). Replacing the transistor by its small-signal model, Fig. 7.39(b) shows the small-signal equivalent.

#### Input Resistance $R_i$

Since the gate current of a MOSFET is almost zero, $R_i = v_g/i_g = \infty$.

#### Voltage Gain $A_{Vo}$

Since the drain current $i_d = g_m v_{gs1}$ flows through the parallel combination of $r_{o1}$ and $r_{o2}$, the small-signal output voltage $v_o$ is given by

$$v_o = g_m v_{gs} (r_{o1} \parallel r_{o2}) \quad (7.75)$$

Substituting $v_{gs} = v_g - v_o$ in Eq. (7.75), we get

$$v_o = g_m (r_{o1} \parallel r_{o2})(v_g - v_o) \quad (7.76)$$

This, after simplification, gives the small-signal voltage gain $A_{vo}$ as

$$A_{vo} = \frac{v_o}{v_g} = \frac{g_m (r_{o1} \parallel r_{o2})}{1 + g_m (r_{o1} \parallel r_{o2})} \quad (7.77)$$

For $(r_{o1} \parallel r_{o2}) >> 1$, $A_{vo} \approx 1$.

#### Output Resistance $R_o$

We can obtain the output resistance $R_o$ and after $0 v_{gs1} = 0$ by applying a test voltage $v_x$ and finding the current $i_x$ as shown in Fig. 7.39(c). By inspection, we can write $R_o$ as

$$R_o = \frac{v_x}{i_x} = \frac{1}{g_m \parallel r_{o1} \parallel r_{o2}} \quad (7.78)$$

which can be approximated to $R_o \approx 1/g_m$ for $r_{o1} \parallel r_{o2} >> 1/g_m$.
EXAMPLE 7.8

**Design of a CD amplifier with a MOS current source**

(a) Use the sinking current in Fig. 7.29(b) to bias the source follower in Fig. 7.39(a) at a drain current of $I_D = 3.25$ mA. The DC supply voltage is $V_{DD} = 15$ V, and $R_{ref} = 4$ kΩ. The MOS parameters are $V_t = 1$ V, $K_n = 3.25$ mA/V$^2$, $K_p = 6.5$ mA/V$^2$, and $W = L$, and $|V_M| = 1/\lambda = 100$ V.

(b) Find the small-signal voltage $A_{vo}$ and the output resistance $R_o$.

(c) Use SPICE to plot the small-signal output voltage $v_o$ for a sinusoidal input signal $v_s$ of 1 mV at 1 kHz.

**SOLUTION**

(a) From Example 7.6, we get $V_{GS} = 2$ V, $R_{ref} = 4$ kΩ, $r_o = r_{o1} = r_{o2} = 30.77$ kΩ, $g_m1 = 6.5$ mA/V, $R_2 = 100$ kΩ, and $R_1 = 650$ kΩ.

(b) From Eq. (7.77),

$$A_{vo} = \frac{g_{m1}(r_{o1} \parallel r_{o2})}{1 + g_{m1}(r_{o1} \parallel r_{o2})} = \frac{6.5 \times (30.77 \parallel 30.77)}{1 + 6.5 \times (30.77 \parallel 30.77)} = 0.99$$

From Eq. (7.78),

$$R_o = \frac{1}{g_{m1} r_{o1} \parallel r_{o2}} = \frac{1}{6.5 \times (30.77 \parallel 30.77)} = 152.3 \Omega$$

(c) The PSpice schematic is shown in Fig. 7.40. The plot of the output voltage is shown in Fig. 7.41, which gives a voltage gain 0.988 V/V, which is close to the calculated value of 0.99 V/V. Note there is no phase shift.

**FIGURE 7.40** PSpice schematic for a source follower with an NMOS biasing for Example 7.8

**FIGURE 7.41** PSpice plot of small-signal output voltage for Example 7.8
7.9.2 Resistive-Biased Source Follower

A source follower with resistive biasing is shown in Fig. 7.42(a). Let us assume that $C_1$ and $C_2$ are very large, tending to infinity. That is, $\frac{C_1}{H_{1105}} \approx \frac{C_2}{H_{11015}}$. The small-signal AC equivalent circuit of the amplifier is shown in Fig. 7.42(b).

**Input Resistance $R_i (= v_g/i_g)$**

The input resistance $R_i$ is given by

$$R_i = \frac{v_g}{i_g} = \frac{R_1 \parallel R_2}{R_G}$$

We can apply Eqs. (7.75) and (7.76) to obtain the output voltage, which, after simplification, gives the open-circuit voltage gain $A_{vo}$ as

$$A_{vo} = \frac{v_o}{v_g} = \frac{g_m(r_{o1} \parallel R_{SR})}{1 + g_m(r_{o1} \parallel R_{SR})}$$  \hspace{1cm} (7.79)
Output Resistance $R_o$

We can obtain the output resistance $R_o$ by setting $v_s$ equal to zero and then applying a test voltage $v_x$ at the output side.

The output resistance $R_o$ is given by

$$R_o = \frac{v_x}{i_x} = \frac{1}{g_m} \parallel r_{o1} \parallel R_{SR}$$ (7.80)

**NOTE** The no-load voltage gain $A_v$ of a common-drain amplifier approaches unity. The input resistance $R_i$ is very high. The output resistance $R_o$ is low.

**EXAMPLE 7.9**

**Designing a depletion MOSFET source follower** Design a source follower as shown in Fig. 7.42(a) to yield $R_i \approx 500 \, \text{k}\Omega$ and $i_D = 10 \, \text{mA}$. The MOS parameters are $V_p = -4 \, \text{V}$, $I_{DSS} = 20 \, \text{mA}$, and $V_M = -200 \, \text{V}$. Assume $V_{DD} = 20 \, \text{V}$.

**SOLUTION**

The design of a common-drain (CD) amplifier is very simple; it requires determining the values of $R_{SR}$. We know that

$$K_n = \frac{I_{DSS}}{V_p^2} = \frac{20 \, \text{mA}}{(-4)^2} = 1.25 \, \text{mA/V}^2$$

**Step 1.** For the depletion MOSFET, we can use the self-biasing circuit arrangement as shown in Fig. 7.24(b) where $R_i = \infty$. Calculate the gate resistance $R_2$:

$$R_2 = R_i = 500 \, \text{k}\Omega$$

**Step 2.** For known values of $i_D$, $I_{DSS}$, and $V_p$, calculate $v_{GS}$ from Eq. (7.21), $i_D = K_n(v_{GS} - V_p)^2$:

$$10 \, \text{mA} = 1.25 \, \text{mA/V}^2 \times (v_{GS} + 4)^2$$

which gives $v_{GS} = -1.172 \, \text{V}$ or $-6.828 \, \text{V}$. The acceptable value is $v_{GS} = -1.172 \, \text{V}$.

**Step 3.** For the known value of $v_{GS}$, calculate $R_{SR}$:

Using KVL through the loop formed by the gate, $R_G$ and $R_{SR}$ in Fig. 7.24(b), we get

$$0 = v_{GS} + R_{SR}i_D$$

Which gives

$$R_{SR} = -\frac{v_{GS}}{i_D} = -\left(\frac{-1.172 \, \text{V}}{10 \, \text{mA}}\right) = 117.2 \, \Omega$$

**Step 4.** Find the small-signal parameters of the transistor. From Eq. (7.30),

$$g_m = 2K_n(v_{GS} - V_p) = 2 \times 1.25 \, \text{m} \times (-1.172 + 4) = 7.07 \, \text{mA/V}$$
7.10 Common-Gate Amplifiers

A common-gate (CG) amplifier is shown in Fig. 7.43(a). The circuit can be redrawn as shown in Fig. 7.43(b). The biasing of this circuit is identical to that of the common-source amplifier, and the DC bias circuit can be designed using the same technique. Let us assume that the values of \( C_1 \) and \( C_2 \) are very large, tending to infinity. That is, 
\[
\frac{C_1}{R_{D}} = \frac{C_2}{R_{L}}.
\]
The small-signal AC equivalent circuit of the amplifier is shown in Fig. 7.44(a), which can be simplified to Fig. 7.44(b).

Input Resistance \( R_i \) \((- v_{gs}/i_d)\)

The input resistance \( R_i \) depends on \( R_D \), which becomes parallel to the load resistance \( R_L \). Thus, \( R_L \) must be included with \( R_D \) in the determination of \( R_i \) when the amplifier is operated with a load resistance \( R_L \). Using KVL around the source–gate–drain loop of Fig. 7.44(b) gives an expression for the gate-to-source voltage:
\[
-v_{gs} = \mu_g v_{gs} \left( r_{o1} + R_D \| R_L \right) i_d
\]
which yields
\[
i_d = \frac{(1 + \mu_g) v_{gs}}{r_{o1} + R_D \| R_L}
\]
Using KCL at source node S in Fig. 7.44(b) yields an expression for the input current \( i_s \):

\[
i_s = -\frac{v_{gs}}{R_{SR}} - i_d = -\frac{v_{gs}}{R_{SR}} - \frac{(1 + \mu_g)v_{gs}}{r_{o1} + R_D || R_L}
\]

which gives the input resistance \( R_i \) of the amplifier as

\[
R_i = \frac{-v_{gs}}{i_s} = R_{SR}\left[\frac{r_{o1} + R_D \parallel R_L}{1 + \mu_g}\right]
\]

(7.81)

Since \( \mu_g > 1 \), the input resistance \( R_i \) becomes low. This is a limitation of the common-gate configuration, unless a low \( R_i \) (or \( Z_i \)) is desirable for impedance matching.

\[\text{FIGURE 7.44} \quad \text{Small-signal AC equivalent circuits for a CG amplifier}\]
No-Load Voltage Gain $A_{vo}$ ($= v_o/v_{gs}$)

Using KVL around loop I in Fig. 7.44(b) yields an expression for the gate-to-source voltage $v_{gs}$:

$$-v_{gs} = \mu_g v_{gs} - r_{o1} i_d - i_d R_D$$

which gives

$$i_d = \frac{(1 + \mu_g) v_{gs}}{r_{o1} + R_D}$$

The no-load output voltage $v_o$ is

$$v_o = -R_D i_d = -\frac{R_D (1 + \mu_g) v_s}{r_{o1} + R_D}$$

which gives the no-load voltage gain $A_{vo}$ as

$$A_{vo} = \frac{v_o}{-v_{gs}} = \frac{R_D (1 + \mu_g)}{r_{o1} + R_D}$$

(7.82)

Output Resistance $R_o$

Assuming that the output resistance of the transistor is very large, tending to infinity (i.e., $r_{o1} \approx \infty$), the output resistance $R_o$ can be found by inspection to be $R_o = R_C$.

**EXAMPLE 7.10**

Finding the parameters of a common-gate amplifier

The CG amplifier of Fig. 7.44(a) has $R_s = 500 \, \Omega$, $R_{SR} = 1 \, \Omega$, $R_D = 5 \, \Omega$, and $R_L = 10 \, \Omega$. The transistor parameters are $r_{o1} = 100 \, k\Omega$ and $\mu_g = 230$. Assume that $C_1$ and $C_2$ are very large, tending to infinity. That is, $C_1 = C_2 = \infty$. Calculate (a) the input resistance $R_{in} = v_s/i_s$, (b) the no-load voltage gain $A_{vo} = v_o/(-v_{gs})$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L/v_s$.

**SOLUTION**

$R_s = 500 \, \Omega$, $R_{SR} = 1 \, \Omega$, $R_D = 5 \, \Omega$, $R_L = 10 \, \Omega$, $r_{o1} = 100 \, k\Omega$, and $\mu_g = 230$.

(a) From Eq. (7.81),

$$R_i = R_{SR} \left\| \left( \frac{r_{o1} + R_D}{1 + \mu_g} \right) \right\| R_L = 1 \, k \mid \frac{100 \, k + 5 \, k}{1 + 230} = 309 \, \Omega$$

$$R_{in} = R_i + R_s = 309 + 500 = 809 \, \Omega$$

(b) From Eq. (7.82),

$$A_{vo} = \frac{v_o}{-v_{gs}} = \frac{R_D (1 + \mu_g)}{r_{o1} + R_D}$$

$$= 5 \, k \times \frac{1 + 230}{100 \, k + 5 \, k} = 11$$
7.11 Multistage Amplifiers

The design requirements of amplifiers normally specify an overall high voltage gain, a high input resistance, and a low output resistance. A single-transistor amplifier rarely satisfies the design requirements, and multistage are often used to satisfy the design specifications. To achieve the design specifications, multiple transistor stages are connected in such a way that the output of one stage is the input to the next stage and so on. The most common types of arrangements are (a) capacitor-coupled cascaded, (b) direct-coupled, and (c) cascoded.

7.11.1 Capacitor-Coupled Cascaded Amplifiers

In a capacitively coupled amplifier, the output of one stage is connected to the input of the next stage via a capacitor as shown in Fig. 7.45(a). The first stage is generally a common-source amplifier that is designed to offer the maximum voltage and a high input resistance $R_i$, which is inherent in MOS amplifiers. The source follower in the third stage satisfies the requirement of a low output resistance $R_o$. The second stage is a common-source amplifier which is needed to yield additional gain in meeting the overall voltage gain requirement $A_v$. If $A_{vo1}$ and $A_{vo3}$ ($\approx 1$) are the voltage gains of the first and third stages, respectively, then the required gain for the second stage is $A_{vo2} = A_{vo}/(A_{vo1} \times A_{vo3})$. The DC biasing point of each stage can be determined independently for each stage because the coupling capacitor provides DC isolation between the stages. The biasing drain current should be low to reduce the power drain from the DC voltage source.

Each stage can be represented by its parameters $R_i$, $R_o$, and $A_{vo}$ as shown in Fig. 7.45(b). The output resistance of a stage acts as the source resistance of the following stage, and the input resistance of a stage is the load resistance of the preceding stage. There will be a loading effect due to the interaction between stages, and the effective voltage gain will be reduced (see Sec. 2.4). While designing an amplifier, we should keep in mind the gain reduction due to the loading effect and should start the design with a voltage gain higher than $A_{vo1}$ to satisfy the overall design requirement.

7.11.2 Direct-Coupled Amplifiers

In direct-coupled amplifiers, the output of one stage is directly connected to the input of the next stage. We can make the amplifier in Fig. 7.45(a) a direct-coupled amplifier if we remove the coupling capacitors $C_2$.

---

(c) $R_o = R_D = 5 \, k\Omega$.

(d) For the overall voltage gain $A_v$,

$$A_v = \frac{v_L}{v_s} = \frac{A_{vo}R_iR_L}{(R_i + R_o)(R_L + R_o)} = \frac{11 \times 309 \times 10 \, k}{(309 + 500) \times (10 \, k + 5 \, k)} = 2.83$$

---

KEY POINT OF SECTION 7.10

- A common gate amplifier which has no Miller’s effect is used in high-frequency applications. Both the input resistance and the voltage are low. There is no phase of the output voltage and it can be used for impedance matching.
and C3 and connect the following stage directly to the preceding stage. In this case, we can also remove the biasing resistances.

7.11.3 Cascoded Amplifiers

We can increase the effective output resistance of a transistor by connecting two transistors in a configuration commonly referred to as a cascaded amplifier. The input signal is applied to one transistor M1 operating as a common-source amplifier, whose output is the input to the other transistor M2 operating as a common-gate amplifier. This is shown in Fig. 7.46(a). The input signal is applied to the common-source amplifier, and the output is obtained at the drain of the common-gate amplifier. Figure 7.46(a) is an example of a resistive biased cascoded amplifier. The cascading can be done with more than two transistors as shown in Fig. 7.46(e). The transistors M2, . . ., M4 are biased by level-shifted transistors M2B, . . ., M4B. This type of cascoding is commonly done in differential amplifiers (Chapter 9) to obtain large voltage gains.

DC Biasing

The DC equivalent circuit for determining the DC operating point of the transistors is shown in Fig. 7.46(b). We can simplify the analysis by assuming identical transistors \( V_{t1} = V_{t2} = V_t \). Since the same drain current will flow through all the transistors, \( V_{GS1} = V_{GS2} = V_{GS} \). Therefore, we can find the DC biasing gate voltages as given by

\[
V_{G1} = \frac{R_1}{R_1 + R_2 + R_3} V_{DD} \tag{7.83}
\]

\[
V_{G2} = \frac{R_1 + R_2}{R_1 + R_2 + R_3} V_{DD} \tag{7.84}
\]
FIGURE 7.46  Cascoded amplifier
Using KVL in the gate-to-source loop of M1, we can write

\[ V_{G1} = V_{GS1} + R_{SR} I_{D1} = V_{GS1} + R_{SR} K_n (V_{GS1} - V_t)^2 \]  \hspace{1cm} (7.85)

which can be solved for \( V_{GS1} = V_{GS2} = V_{GS} \) and then the drain currents:

\[ I_{D1} = I_{D2} = I_D = K_n (V_{GS1} - V_t)^2 \]  \hspace{1cm} (7.86)

**Small-Signal Voltage Gain**

Once we have found the DC biasing values, we can find the small-signal model parameters of \( g_{m1} = g_{m2} = \frac{1}{r_{o1}} \) for both transistors. Assuming that output resistances \( r_{o1} \) and \( r_{o2} \) of the transistors are very high, tending to infinity, the small-signal equivalent circuit is shown in Fig. 7.44(c).

We can find the output voltage as

\[ v_o = -i_{d1} R_D = -g_{m1} R_D v_g \]

which gives the voltage gain as

\[ A_{vol} = \frac{v_o}{v_g} = -g_{m1} R_D \]  \hspace{1cm} (7.87)

**Small-Signal Output Resistance**

\[ R_o = R_D. \]

---

**KEY POINTS OF SECTION 7.11**

- A single-transistor amplifier rarely satisfies the design requirements, and multistages are often used to satisfy the design specifications.
- The multiple transistor stages are connected in such a way that the output of one stage is the input to the next stage and so on.
- The most common types of arrangements are (a) capacitor-coupled cascaded, (b) direct-coupled, and (c) cascoded.

---

**7.12 DC Level Shifting and Amplifier**

In all the amplifiers discussed so far, we used coupling capacitors to superimpose the small AC signal on the DC biasing voltage at the gate terminal of the transistors. These capacitors provide DC isolation of each stage from the previous or subsequent stage. The amplified AC signal is superimposed on the DC biasing voltage at the output terminal of the transistors. The coupling capacitors cannot be used in the design of amplifiers that amplify only DC signals. In some cases, it may be necessary to shift the quiescent voltage of one stage before applying its output to the following stage. Level-shifting circuits can adjust the DC bias levels between amplification stages. Level shifting is also required in order for the output to be close to
zero in the quiescent state (at no input signal). The input resistance of the level-shifting stage should be high to prevent loading of the previous stage (usually the gain stage). Also, the output resistance should be low to effectively drive the subsequent stage.

### 7.12.1 Level-Shifting Methods

The source follower configuration is normally used to shift the level. The source follower has an inherent characteristic of level shifting by $v_{GS}$ such that the output voltage is $v_O = v_G - v_{GS}$. Thus, the main idea is to create a voltage in the source terminal, and it can be accomplished by (a) a potential divider network, (b) a current source, and (c) a zener diode.

#### Potential Divider Level Shifting

This arrangement is shown in Fig. 7.47(a). The voltage shift is

$$v_O - v_G = -v_{GS} - R_1i_D = -v_{GS} - R_1K_n(v_{GS} - V_i)^2$$

which gives the output voltage as

$$v_O = v_G - v_{GS} - R_1K_n(v_{GS} - V_i)^2$$

(7.88)

Since $V_{GS}$ is fixed for a specific drain current, a small change $\Delta v_G$ will cause the same change to the output voltage; that is, $\Delta v_O = \Delta v_G$.

#### Current Source Level Shifting

Resistance $R_2$ in Fig. 7.47(a) can be replaced with a current source at a constant current $I_O$ as shown in Fig. 7.47(b). The voltage shift is

$$v_O - v_G = -(v_{GS} + R_1I_O) = -v_{GS} - R_1K_n(v_{GS} - V_i)^2$$

![FIGURE 7.47 Level shifting](image-url)
which gives the output voltage as
\[ v_O = v_G - v_{GS} - R_1K_n(v_{GS} - V_i)^2 \]  
\[ (7.89) \]

Since the current through \( R_1 \) is fixed, the voltage drop across it is also fixed. It is important to note that \( v_O \) is independent of the negative DC supply voltage \(-V_{SS}\).

### Zener Level Shifting

Resistance \( R_1 \) in Fig. 7.47(a) can be replaced by a zener diode with zener voltage \( V_Z \); this arrangement is shown in Fig. 7.47(c). The voltage shift is
\[ v_O - v_S = -(v_{GS} + V_Z) \]

which gives the output voltage as
\[ v_O = v_S - v_{GS} - V_Z \]  
\[ (7.90) \]

Since the zener voltage \( V_Z \) is fixed, the voltage drop across it is also fixed.

### 7.12.2 Level-Shifted MOS Amplifier

A MOS amplifier using level shifting is shown in Fig. 7.48(a) with four stages. The first stage generates the reference current for the second stage, which also acts as the reference current for the third stage. The fourth stage is the source follower. We will assume that all transistors are matched devices and have equal parameters: the current gains \( K_{n1} = \cdots = K_{n8} = K_n \), \( V_{t1} = \cdots = V_{t8} = V_t \), the modulation voltages, and \( V_{M1} = \cdots = V_{M8} = V_M \).

### Current Mirror Source

Assuming that we want to set the reference drain current at \( I_D = I_{D1} = I_{D2} = I_{D3} = I_{D8} \), their gate-to-source voltages must be equal. That is,
\[ V_{GS1} = V_{GS2} = V_{GS3} = V_{GS8} = V_{GS} = \frac{I_D}{\sqrt{K_n}} + V_i \]  
\[ (7.91) \]

Using KVL in the gate-to-source loop of \( M_1 \) and \( M_2 \), we find the value of \( R \) as given by
\[ R = R_1 + R_2 = \frac{V_G + V_{SS} - V_{GS1} - V_{GS2}}{I_{D1}} \]  
\[ (7.92) \]

Using KVL in the gate-to-source loop of \( M_4 \) and \( M_5 \), and applying the relationship for the gate-to-source voltage of a MOS \( V_{GS} = (\sqrt{I_D/K_n} + V_i) \), we find the value of \( R_3 \) as given by
\[ R_3 = \frac{V_{DD} + V_{SS} - V_{GS4} - V_{GS5}}{I_{D4}} = \frac{V_{DD} + V_{SS} - (\sqrt{I_{D4}/K_n} + V_i) - (\sqrt{I_{D5}/K_n} + V_i)}{I_{D4}} \]  
\[ (7.93) \]
We can simplify this by assuming \( I_{D5} \approx I_{D3} = I_D \) and using the relation \( I_{D4} = I_{D3} + I_{D5} \), as given by

\[
R_3 = \frac{V_{DD} + V_{SS} - (\sqrt{2I_D/K_n} + V_t) - (\sqrt{I_D/K_n} + V_t)}{2I_D}
\]

\[
= \frac{V_{DD} + V_{SS} - 2.14 \sqrt{I_D/K_n} - 2V_t}{2I_D}
\]

(7.94)
Under these conditions, \( I_{D5} = I_{D6} \approx I_D \) and \( I_{D4} \approx 2I_D \). The voltage \( V_x \) at the gate of \( M_7 \) is given by
\[
V_x = V_{DD} - R_x I_{D6} = V_{DD} - R_x I_D
\] (7.95)

\( M_7 \) and \( M_8 \) act as a source follower. Therefore, the output voltage is given by
\[
V_o = V_x - V_{GS7} = V_{DD} - R_x I_D - V_{GS7}
\] (7.96)

Therefore, we can shift to \( V_O = 0 \) by making \( R_x \) as given by
\[
R_x = \frac{V_{DD} - V_{GS7}}{I_D} = \frac{V_{DD} - (\sqrt{I_D/K_n} + V_t)}{I_D}
\] (7.97)

A resistance \( R_y \) as shown in Fig. 7.48(a) is connected at the gate terminal of \( M_5 \) in order to divert the drain current \( I_{D5} \), which acts as the reference current for \( I_{D6} \). As a result, \( I_{D5} \) becomes closer to the value of \( I_{D3} = I_D \). In MOS IC design, there is no need for \( R_y \) because the widths of \( M_3 \), \( M_5 \), and \( M_6 \) can be scaled to carry equal drain currents \( I_{D3} = I_{D5} = I_{D6} \).

**Small-Signal Voltage Gain**

By replacing the transistor by its small-signal mode, the small-signal equivalent circuit is shown in Fig. 7.48(b). The capacitor \( C_A \) and the batteries are shorted to ground. Using KVL through the resistance \( R_2 \) loop, we can find the small-signal voltage \( v_g \) at the source terminal of \( M_1 \):
\[
v_g - v_{gs1} = (R_2 + 1/g_m)g_{m2}v_{gs1}
\]

This relates the small-signal gate-to-source voltage to the input signal \( v_g \) as
\[
v_{gs1} = \frac{v_g}{1 + (R_2 + 1/g_m)g_{m1}}
\] (7.98)

Therefore, we can get the small-signal reference current \( i_{d2} \):
\[
i_{d2} = i_{d3} = g_{m1}v_{gs1} = \frac{g_{m1}v_g}{1 + (R_2 + 1/g_m)g_{m1}}
\] (7.99)

Since \( i_{d3} \) is the input current to the drain terminals of \( M_3 \) and \( M_5 \), we can find the drain current \( i_{d5} \) by applying the current divider rule as given by
\[
i_{d5} = i_{d6} = \frac{R_3 + 1/g_m}{R_3 + 1/g_m + 1/g_m} i_{d2}
\] (7.100)

which, after we substitute \( i_{d2} \) from Eq. (7.99), becomes
\[
i_{d5} = i_{d6} = \frac{R_3 + 1/g_m}{R_3 + 1/g_m + 1/g_m} \times \frac{g_{m1}v_g}{1 + (R_2 + 1/g_m)g_{m1}}
\] (7.101)

Thus the small-signal voltage \( v_x \) becomes
\[
v_x = R_x i_{d6} = \frac{R_3 + 1/g_m}{R_3 + 1/g_m + 1/g_m} \times \frac{R_x g_{m1}v_g}{1 + (R_2 + 1/g_m)g_{m1}}
\] (7.102)
which gives the voltage gain between \( v_g \) and \( v_x \) as

\[
A_{vx} = \frac{v_x}{v_g} = \frac{(R_3 + 1/g_{m4}) R_x g_{m1}}{(R_3 + 1/g_{m4} + 1/g_{m5})(1 + (R_2 + 1/g_{m2})g_{m1})}
\]  
(7.103)

Due to the source follower, the voltage gain \( A_{vx} \) is attenuated, and the no-load voltage gain \( A_{vo} \) becomes

\[
A_{vo} = \frac{g_{m7}(r_{o7} || r_{o8})}{1 + g_{m7}(r_{o7} || r_{o8})} \quad A_{vx} = \frac{(R_3 + 1/g_{m4}) R_x g_{m1}}{(R_3 + 1/g_{m4} + 1/g_{m5})(1 + (R_2 + 1/g_{m2})g_{m1})}
\times \frac{g_{m7}(r_{o7} || r_{o8})}{1 + g_{m7}(r_{o7} || r_{o8})}
\]  
(7.104)

For \( R_2 \gg 1/g_{m2}, R_3 \gg (1/g_{m4} + 1/g_{m5}), \) and \( (r_{o7} || r_{o8}) \gg 1, \) Eq. (7.104) can be approximated to

\[
A_{vo} \approx \frac{R_x}{R_2}
\]  
(7.105)

It is important to note that any increase in \( v_g \) causes the current \( i_{d1} \) to increase, which is mirrored to \( i_{d3} \); this in turn decreases \( i_{d5} \) by the same amount. \( i_{d6} \), which is a mirror of \( i_{d5} \), causes the voltage \( v_x \) to increase by \( R_x i_{d6} \). The transistors, which act as current mirrors and shift the voltage levels, do not produce any voltage amplification. The voltage gain described by Eq. (7.104) is accomplished by shunting \( R_1 \) by the capacitor \( C \) for AC signals. The maximum voltage gain can be obtained by shunting both \( R_1 \) and \( R_2 \) by capacitor \( C_A \); that is, for \( R_2 = 0, \) Eq. (7.104) gives the maximum voltage gain as

\[
A_{vo(max)} = \frac{(R_3 + 1/g_{m4}) R_x g_{m1}}{(R_3 + 1/g_{m4} + 1/g_{m5})(1 + g_{m7}/g_{m2})} \times \frac{g_{m7}(r_{o7} || r_{o8})}{1 + g_{m7}(r_{o7} || r_{o8})}
\]  
(7.106)

The design of this amplifier is very simple. It requires only finding the values of \( R_1, R_2, R_y, \) and \( R_x = (R_1 + R_2) \) to give a specific voltage gain \( A_{vo} \).

**EXAMPLE 7.11**

**Finding the small-signal voltage gain of a level-shifted amplifier**

The parameters of the amplifier in Fig. 7.48(a) are \( V_{DD} = 15 \) V, \( V_{SS} = -15 \) V, \( R_1 = 9 \) kΩ, \( R_2 = 1 \) kΩ, \( R_3 = 4 \) kΩ, \( R_4 = 10 \) kΩ, and \( R_y = 6.5 \) kΩ. The circuit is biased at a DC gate-source voltage of \( V_G = 1 \) V. The MOS parameters are \( V_t = 1 \) V, \( K_n = 3.25 \) mA/V^2, \( K_p = 6.5 \) mA/V^2 for \( W = L, \) and \( |V_M| = 1/\lambda = 100 \) V.

(a) Find the small-signal voltage \( A_{vo} \) and the maximum possible gain.

(b) Use SPICE to plot the small-signal output voltage for a sinusoidal input signal of 1 mV at 1 kHz.

**SOLUTION**

From Eq. (8.92),

\[
V_G + V_{SS} - V_{GS1} - V_{GS1} - (R_1 + R_2)K_n(V_{GS1} - V_t)^2 = 0 \quad \text{or}
\]

\[
1 + 12 - V_{GS1} - V_{GS1} - (1 \times 9) \times 3.25 \times 10^{-3} \times (V_{GS1} - 1)^2 = 0
\]
which gives

\[ V_{GS} = 1.552 \text{ V} \]

\[ I_{D1} = K_n(V_{GS1} - V_t)^2 = 3.25 \text{ m} \times (1.552 - 1)^2 = 990 \text{ } \mu \text{A} \]

\[ V_{GS4} = \sqrt{2I_{D1}/K_n} = \sqrt{2 \times 990 \text{ } \mu \text{A}/3.25 \text{ m} + 1} = 1.784 \text{ V} \]

\[ g_{m2} = g_{m3} = g_{m5} = g_{m6} = g_{m7} = 2K_n(V_{GS1} - V_t) = 2 \times 3.25 \text{ m} \times (1.552 - 1) = 3.606 \text{ mA/V} \]

\[ g_{m4} = 2K_n(V_{GS4} - V_t) = 2 \times 3.25 \text{ m} \times (1.784 - 1) = 5.099 \text{ mA/V} \]

\[ r_{o7} = \frac{V_{M7}}{I_{D7}} = 100 \frac{100 \text{ } \mu \text{A}}{990 \text{ } \mu \text{A}} = 100 \text{ k} \Omega \]

Substituting the values in Eq. (7.103), we get

\[ A_{vx} = \frac{(R_3 + 1/g_{m4})R_x g_{m1}}{(R_3 + 1/g_{m4} + 1/g_{m5})[1 + (R_2 + 1/g_{m2})g_{m1}]} \]

\[ = \frac{(10 \text{ k} + 1/5.099 \text{ m}) \times 10 \text{ k} \times 3.606 \text{ m}}{(10 \text{ k} + 1/5.099 \text{ m} + 1/3.606 \text{ m})[1 + (1 \text{ k} + 1/3.606 \text{ m}) \times 3.606 \text{ m}]} = 6.262 \text{ V/V} \]

Substituting the values in Eq. (7.104), we get

\[ A_{vo} = \frac{g_{m7}(r_{o7})}{1 + g_{m7}(r_{o7})}A_{vx} = \frac{3.606 \text{ m} \times (100 \text{ k} \parallel 100 \text{ k})}{1 + 3.606 \text{ m} \times (100 \text{ k} \parallel 100 \text{ k})} \times 6.262 = 6.227 \text{ V/V} \]

Substituting the values for \( R_2 = 0 \) in Eq. (7.104) gives \( A_{vo(max)} = 17.454 \text{ V/V} \).

(b) The PSpice plot of the DC transfer function is shown in Fig. 7.49(a), which has an offset voltage of 36.93 mV at \( v_{G} = 0 \). The plot of the output voltage is shown in Fig. 7.49(b), which gives a voltage gain of 5.78, which is close to the calculated value of 6.227. Note that there is no phase shift and \( r_{ds} = 1/r_o \). The PSpice biasing drain currents and the small-signal parameters are listed here:

<table>
<thead>
<tr>
<th>Name</th>
<th>M_M1</th>
<th>M_M2</th>
<th>M_M3</th>
<th>M_M4</th>
<th>M_M5</th>
<th>M_M6</th>
<th>M_M7</th>
<th>M_M8</th>
</tr>
</thead>
<tbody>
<tr>
<td>VGS</td>
<td>1.49E+00</td>
<td>1.52E+00</td>
<td>1.52E+00</td>
<td>1.79E+00</td>
<td>1.53E+00</td>
<td>1.53E+00</td>
<td>1.52E+00</td>
<td>1.52E+00</td>
</tr>
<tr>
<td>GM</td>
<td>3.64E-03</td>
<td>3.44E-03</td>
<td>3.44E-03</td>
<td>5.23E-03</td>
<td>3.51E-03</td>
<td>3.93E-03</td>
<td>3.80E-03</td>
<td>3.80E-03</td>
</tr>
<tr>
<td>GDS</td>
<td>7.92E-06</td>
<td>8.85E-06</td>
<td>8.85E-06</td>
<td>2.03E-05</td>
<td>9.19E-06</td>
<td>9.19E-06</td>
<td>8.86E-06</td>
<td>8.85E-06</td>
</tr>
</tbody>
</table>

![FIGURE 7.49](image)
7.13 Frequency Response of MOSFET Amplifiers

The frequency response of MOS amplifiers will depend on the internal MOS junction capacitances and any external capacitances such as coupling and bypass capacitances. To determine the frequency characteristics, we need to add capacitances to the small-signal AC models of MOSFETs. In Secs. 2.7.4 and 2.7.5, we introduced short-circuit and zero-value methods. As examples, we will use these methods for determining the frequency response of MOSFETs, single-stage MOS amplifiers, and multistage amplifiers.

7.13.1 High-Frequency MOSFET Models

The small-signal high-frequency model of the MOSFETs of Fig. 7.50(a) in the saturation region is shown in Fig. 7.50(b). The gate-to-source capacitance $C_{gs}$ and the gate-to-drain capacitance $C_{gd}$ can be found approximately from [9, 10]

$$C_{gs} = \frac{C_{gs0}}{1 + |V_{GS}|/V_{bi}}^{1/3}$$

and

$$C_{gd} = \frac{C_{gd0}}{1 + |V_{GD}|/V_{bi}}^{1/3}$$

where $V_{bi}$ = built-in potential with a zero applied voltage

$C_{gs0}$ = value of $C_{gs}$ at $V_{GS} = 0$ and is typically in the range of 1 pF to 4 pF

$C_{gd0}$ = value of $C_{gd}$ at $V_{GD} = 0$ and is typically in the range of 0.3 pF to 1 pF

$C_{sb}$ and $C_{bd}$ = depletion-layer capacitances from the source to the substrate and from the substrate to the drain, respectively

(Note that in order to avoid confusion between substrate and source terminals of a MOSFET, substrate is being abbreviated with a subscript b.) These capacitances can be found approximately from

$$C_{sb} = \frac{C_{sb0}}{1 + |V_{SB}|/V_{bi}}^{1/2}$$

and

$$C_{bd} = \frac{C_{bd0}}{1 + |V_{DB}|/V_{bi}}^{1/2}$$

where $V_{bi}$ is the built-in (or barrier) potential and is typically 0.6 V and $C_{sb0}$ and $C_{bd0}$ are the zero-biased capacitances and are typically 0.1 pF. The values of $C_{sb}$ and $C_{bd}$ range from 0.01 pF to 0.05 pF. To reduce the values of $C_{sb}$ and $C_{bd}$, the substrate of a MOSFET is often connected to the negative DC supply voltage so that $|V_{SB}|$ and $|V_{DB}|$ have higher values.

$C_{gb}$ is the parasitic oxide capacitance between the gate contact material and the substrate, and its value depends on the oxide thickness. It ranges from 0.004 fF to 0.15 fF per square micron but is typically 0.1 pF.

$C_{gd}$ is the parasitic oxide capacitance between the gate and the drain. It is also called the overlap capacitance because the drain extends slightly under the gate electrode. Its typical value is in the range of 1 pF to 10 pF.

$C_{gs}$ consists of two capacitances: $C_{gs0}$ and $C_{gs0}$. $C_{gs0}$ is the constant parasitic capacitance due to the overlap of the source region because the source extends slightly under the gate electrode. Its typical
value is 10 fF. $C_{gsq}$ is the gate-to-channel capacitance. The channel has a tapered shape and is pinched off at the drain, so $C_{gsq}$ can be expressed as [1, 9]

$$C_{gsq} = \frac{2}{3} W L C_{ox}$$  \hspace{1cm} (7.111)

where

- $W$ = channel width
- $L$ = channel length
- $C_{ox}$ = capacitance per unit area, which is $3.54 \times 10^{-8}$ F/cm$^2$ for an oxide thickness of $t_{ox} = 0.1 \mu$m

For example, if $W = 30 \mu$m, $L = 10 \mu$m, and $t_{ox} = 0.1 \mu$m, we get $C_{gsq} = 0.07$ pF = 71 fF. Table 7.4 shows the capacitances and output resistances for MOSFETs.

In some applications, the substrate is connected to the source, and the frequency model is reduced to Fig. 7.50(c). Capacitance $C_{bd}$ can often be ignored, especially for hand calculations, and the model simplifies to Fig. 7.50(d). Let us apply a test current $i_g$ to the gate of a MOSFET and short-circuit the drain terminal for AC signals. The high-frequency AC equivalent circuit in the saturation region is shown in Fig. 7.50(e). The voltage at the gate terminal in Laplace’s domain of $s$ is given by

$$V_{gs}(s) = \frac{1}{(C_{gs} + C_{gd})s} I_g(s)$$  \hspace{1cm} (7.112)
Since \( r_o \) is very large and \( C_{gd} \) is very small, the currents through them will be very small. Thus,

\[
I_d(s) = [g_{m} - sC_{gd}]V_{gs}(s)
\]

Substituting \( V_{gs}(s) \) from Eq. (7.112), we get

\[
I_d(s) = \frac{g_{m} - sC_{gd}}{(C_{gs} + C_{gd})s}I_g(s)
\]  

(7.113)

For the frequencies at which the model in Fig. 7.50(e) is valid, \( g_m >> \omega C_{gd} \), and we get the current gain \( \beta_1(j\omega) \) in the frequency domain as

\[
\beta_1(j\omega) = \frac{I_d(j\omega)}{I_g(j\omega)} = \frac{g_m}{(C_{gs} + C_{gd})j\omega}
\]  

(7.114)

which indicates that the current gain will fall as the frequency increases, at a slope of \(-20 \text{ dB/decade}\). This relationship is shown in Fig. 7.50(f). The current gain will be unity, \(|\beta_1(j\omega)| = 1\), and the unity-gain bandwidth \( \omega_T \) is

\[
\omega = \omega_T = \frac{g_m}{C'_{gs} + C_{gd}} = \frac{g_m}{C_{gs} + C_{gd} + C_{gb}} \quad \text{in rad/s}
\]  

(7.115)

or

\[
f_T = \frac{g_m}{2\pi(C_{gs} + C_{gd} + C_{gb})} \quad \text{in Hz}
\]  

(7.116)

For MOSFETs, the value of frequency \( f_T \) ranges from 100 MHz to 2 GHz.

### 7.13.2 Small-Signal PSpice Model

The small-signal parameters of MOSFETs can be determined from the manufacturer’s data sheet or from practical measurements [6–8]. Alternatively, PSpice/SPICE can calculate the DC biasing point and then generate the small-signal parameters. The small-signal AC equivalent circuits generated by PSpice for MOSFETs are shown in Fig. 7.51, where \( r_d \) and \( r_s \) are the parasitic resistances of the drain and source terminals, respectively.

| TABLE 7.4 Parasitic capacitances and output resistances |
|----------------|---------------|
| MOSFETs        |               |
| \( C_{ds} \)   | 0.1–1 pF      |
| \( C_{gs} \)   | 1–10 pF       |
| \( C_{gd} \)   | 1–10 pF       |
| \( r_o \)      | 1–50 k\Omega   |
| \( g_m \)      | 0.1–20 mA/V    |
7.13.3 Common-Source Amplifiers

Once the DC biasing point of a MOSFET amplifier has been determined, the small-signal parameters can be determined, as discussed in Sec. 7.13.1. The high-frequency model of a MOSFET, shown in Fig. 7.52(a), can be simplified to Fig. 7.52(b) for low frequencies. We will apply the short-circuit and zero-value methods to determine the cutoff frequencies of common-source (CS) amplifiers, common-drain (CD) amplifiers, and common-gate (CG) amplifiers.

EXAMPLE 7.12

**Finding the high-frequency model parameters of a depletion MOSFET** The DC biasing values of the MOSFET are \( I_D = 6.3 \text{ mA}, \ V_{DS} = 5 \text{ V}, \) and \( V_{GS} = -1.03 \text{ V} \). The parameters of the MOSFET are \( C_{gs0} = 2.4 \text{ pF}, \ V_{bi} = 0.8 \text{ V for } C_{gs0}, \ C_{gb0} = 1 \text{ pF}, \ C_{gd0} = 1.6 \text{ pF}, \ V_{bi} = 0.8 \text{ V for } C_{gd0}, \ g_m = 4.98 \text{ mA/V}, \) and \( r_o = 26.77 \text{ k}\Omega \).

(a) Calculate the capacitances of the MOS model in Fig. 7.50(d).
(b) Find the unity-gain bandwidth \( f_T \).

**SOLUTION**

(a) From Eq. (7.107), \( C_{gs} = 2.4 \text{ pF}/\left[1 + 1.03/0.8\right]^{1/3} = 1.8 \text{ pF} \). The gate-drain voltage \( V_{GD} \) is

\[ V_{GD} = V_{GS} + V_{SD} = V_{GS} - V_{DS} = -1.03 - 5 = -6.03 \text{ V} \]

From Eq. (7.108), \( C_{gd} = 1.6 \text{ pF}/\left[1 + 6.03/0.8\right]^{1/3} = 0.78 \text{ pF} \).

(b) From Eq. (7.116), the unity-gain bandwidth \( f_T \) is

\[ f_T = \frac{g_m}{2\pi(C_{gs} + C_{gd})} = \frac{4.98 \text{ mA/V}}{2\pi \times (1.8 \text{ pF} + 0.78 \text{ pF})} = 307.2 \text{ MHz} \]

7.13.3 Common-Source Amplifiers

Once the DC biasing point of a MOSFET amplifier has been determined, the small-signal parameters can be determined, as discussed in Sec. 7.13.1. The high-frequency model of a MOSFET, shown in Fig. 7.52(a), can be simplified to Fig. 7.52(b) for low frequencies. We will apply the short-circuit and zero-value methods to determine the cutoff frequencies of common-source (CS) amplifiers, common-drain (CD) amplifiers, and common-gate (CG) amplifiers.
A common-source MOSFET amplifier is shown in Fig. 7.53(a).

Low Cutoff Frequencies

If the MOSFET in Fig. 7.53(a) is replaced by its small-signal model in Fig. 7.52(b), we get the low-frequency equivalent circuit shown in Fig. 7.53(b). There are three capacitors—two coupling capacitors, $C_1$ and $C_2$, and one bypass capacitor, $C_s$. If we assume $C_2$ and $C_s$ are short-circuited, as shown in Fig. 7.54(a), Thevenin’s equivalent resistance presented to $C_1$ is

$$R_{C1} = R_s + R_G$$

where $R_G = R_1 \parallel R_2$.

The equivalent circuit, with $C_1$ and $C_s$ short-circuited, is shown in Fig. 7.54(b). Thevenin’s equivalent resistance presented to $C_2$ is given by

Since $g_m v_{gs}$ behaves as open circuit at $v_{gs} = 0$.

$$R_{C2} = R_D + R_L$$

The equivalent circuit, with $C_1$ and $C_s$ short-circuited, is shown in Fig. 7.54(c). There is no voltage across $R_s$ or $R_G$, so $v_{s} = -v_{gs}$. Therefore, the resistance representing the current source is

$$R_t = \frac{v_{sr}}{g_m v_{gs}} = \frac{-v_{gs}}{-g_m v_{gs}} = \frac{1}{g_m}$$
The Thevenin’s equivalent $C_s$ at the source terminal is

$$R_{CS} = R_{SR} || R_l = R_{SR} || \frac{1}{g_m} \tag{7.119}$$

In general, $R_{CS} < R_{C2} < R_{C1}$, and $R_{CS}$ controls the low 3-dB frequency. Therefore, $f_L = f_{CS}$.

**High Cutoff Frequencies**

If the MOSFET of the CS amplifier in Fig. 7.53(a) is replaced by its high-frequency $\pi$ model in Fig. 7.52(a), we get the high-frequency equivalent circuit shown in Fig. 7.55(a). Since $C_{gd}$ is connected between the input and the output terminals and the output voltage is phase shifted, we can apply either the zero-value method or Miller’s method. We will apply the zero-value method. If we assume $C_{gd}$ is open-circuited and $v_s = 0$, the equivalent circuit is shown in Fig. 7.55(b). Thevenin’s equivalent resistance presented to $C_{gs}$ is

$$R_{Cgs} = R_s || R_G \tag{7.120}$$

The equivalent circuit, with $C_{gs}$ open-circuited, is shown in Fig. 7.55(c). To find the resistance faced by $C_{gd}$, we replace $C_{gd}$ by a test voltage $v_x$, as shown in Fig. 7.55(d). Using KVL around the loop formed by $R_s$ in parallel with $R_G$ and by $R_L$ in parallel with $R_D$, we get

$$v_x = i_x(R_s || R_G) + (g_m v_{gs} + i_x)(R_L || R_D)$$

$$= i_x(R_s || R_G) + [g_m i_x(R_s || R_G) + i_x](R_L || R_D)$$
which gives the resistance faced by $C_{gd}$ as

$$R_{Cgd} = \frac{V_x}{I_x} = R_s \parallel R_G + [1 + g_m(R_s \parallel R_G)](R_L \parallel R_D) \tag{7.121}$$

$$= (R_L \parallel R_D) + (R_s \parallel R_G)[1 + g_m(R_L \parallel R_D)]$$

Thus, the high 3-dB frequency is given by

$$f_H = \frac{1}{2\pi(R_{Cgs}C_{gs} + R_{Cgd}C_{gd})} \tag{7.122}$$

**EXAMPLE 7.13**

**Designing a common-source amplifier to give a specified frequency response**

(a) Design a common-source MOSFET amplifier as shown in Fig. 7.53(a) to give a low 3-dB frequency of $f_L = 150 \text{ Hz}$ and a high 3-dB frequency of $f_H = 2 \text{ MHz}$. The circuit parameters are $C_{gd} = 2 \text{ pF}$, $C_{gs} = 5 \text{ pF}$, $R_s = 200 \text{ }\Omega$, $g_m = 10 \times 10^{-3} \text{ A/V}$, $R_{SR} = 2 \text{ k}\Omega$, $R_D = R_L = 5 \text{ k}\Omega$, $R_1 = 200 \text{ k}\Omega$, and $R_2 = 200 \text{ k}\Omega$.

(b) Use Miller’s method to check the high-frequency design.

**SOLUTION**

$$R_G = R_1 \parallel R_2 = 200 \text{ k}\Omega \parallel 200 \text{ k}\Omega = 100 \text{ k}\Omega.$$

(a) The design will have two parts: one in which we set the low 3-dB frequency at $f_L = 150 \text{ Hz}$ and one in which we set the high 3-dB frequency at $f_H = 2 \text{ MHz}$.
The steps to set $f_L = 150$ Hz are as follows:

**Step 1.** Calculate the equivalent resistances $R_{C1}$, $R_{C2}$, and $R_{CS}$. From Eq. (7.117),

$$R_{C1} = R_s + R_G = 200 + 100 \text{k\Omega} = 300 \text{k\Omega}$$

From Eq. (7.118),

$$R_{C2} = R_D + R_L = 5 \text{k\Omega} + 5 \text{k\Omega} = 10 \text{k\Omega}$$

From Eq. (7.119),

$$R_{CS} = 2 \text{k\Omega} \left( \frac{1}{10 \times 10^{-3}} \right) = 95.2 \Omega$$

**Step 2.** Assume that $f_{CS}$ is the dominant cutoff frequency. Then $f_{CS} = f_L = 150$ Hz.

**Step 3.** Calculate the required value of $C_S$:

$$f_{CS} = \frac{1}{2\pi R_{CS} C_S} = \frac{1}{2\pi \times 95.2 \times C_S} = 150 \text{ Hz} \quad \text{or} \quad C_S = 11.1 \ \mu\text{F}$$

**Step 4.** Assume $f_{C2} = f_L / 10 = 150 / 10 = 15$ Hz.

**Step 5.** Calculate the required value of $C_2$:

$$f_{C2} = \frac{1}{2\pi R_{C2} C_2} = \frac{1}{2\pi \times 10 \text{k\Omega} \times C_2} = 15 \text{ Hz} \quad \text{or} \quad C_2 = 1.06 \ \mu\text{F}$$

**Step 6.** Assume $f_{C1} = f_L / 20 = 150 / 20 = 7.5$ Hz.

**Step 7.** Calculate the required value of $C_1$:

$$f_{C1} = \frac{1}{2\pi R_{C1} C_1} = \frac{1}{2\pi \times 100.2 \text{k\Omega} \times C_1} = 7.5 \text{ Hz} \quad \text{or} \quad C_1 = 0.21 \ \mu\text{F}$$

The steps to set $f_H = 500$ Hz are as follows:

**Step 1.** From Eq. (7.120),

$$R_{Cgs} = R_s \parallel R_G = 200 \ \Omega \parallel 100 \ \text{k\Omega} = 199.6 \ \text{k\Omega}$$

From Eq. (7.121),

$$R_{Cgd} = (5 \text{k\Omega} \parallel 5 \text{k\Omega}) + (200 \ \Omega \parallel 100 \ \text{k\Omega}) \times [1 + 10 \text{ mA/V} \times (5 \text{k\Omega} \parallel 5 \text{k\Omega})] = 7.69 \ \text{k\Omega}$$

**Step 2.** From Eq. (7.122),

$$f_H = \frac{1}{2\pi [199.6 \ \Omega \times 5 \text{ pF} + 7.69 \text{k\Omega} \times (C_{gd} + C_x)]} = 2 \text{ MHz} \quad \text{or} \quad C_{gd} + C_x = 10.2 \ \text{pF}$$

which gives $C_x = 10.2 - 2 = 8.2 \ \text{pF}$. This is the value of the additional capacitor $C_x$ that is to be connected between the gate and drain terminals.

(b) Applying Eq. (2.98), we have for the effective Miller’s capacitance between the gate and source terminals

$$C_{eq} = (C_{gd} + C_x)[1 + g_m(R_L \parallel R_D)] + C_{gs}$$

$$= 10.2 \ \text{pF} \times [1 + 10 \text{ mA/V} \times (5 \text{k\Omega} \parallel 5 \text{k\Omega})] + 5 \ \text{pF} = 270.2 \ \text{pF}$$

The equivalent resistance faced by $C_{eq}$ is $R_{eq} = R_{Cgs} = R_s \parallel R_G = 199.6 \ \Omega$. Thus the high 3-dB frequency is

$$f_H = \frac{1}{2\pi C_{eq} R_{eq}} = \frac{1}{2\pi \times 270.2 \ \text{pF} \times 199.6 \ \Omega} = 2.95 \text{ MHz}$$
7.13.4 Common-Drain Amplifiers

A common-drain MOSFET amplifier is shown in Fig. 7.56(a).

Low Cutoff Frequencies

Replacing the MOSFET in Fig. 7.56(a) by its small-signal model in Fig. 7.52(b) gives the low-frequency equivalent circuit shown in Fig. 7.56(b), which has two coupling capacitors \( C_1 \) and \( C_2 \). If we assume \( C_2 \) is short-circuited, as shown in Fig. 7.57(a), Thevenin’s equivalent resistance presented to \( C_1 \) is

\[
R_{C1} = R_s + R_G
\]

(7.124)

If \( C_1 \) is short-circuited, the equivalent circuit is shown in Fig. 7.57(b). From Eq. (7.119), the output resistance is given by

\[
R_o = R_{SR} \| \frac{1}{g_m}
\]

(7.125)

Thevenin’s equivalent resistance presented to \( C_2 \) is

\[
R_{C2} = R_L + R_o
\]

(7.126)

\( R_{C2} \), which is normally less than \( R_{C1} \), controls the low cutoff frequency.

---

**FIGURE 7.56** Common-drain MOSFET amplifier

**FIGURE 7.57** Equivalent circuits of a common-drain MOSFET amplifier for the short-circuit method
High Cutoff Frequencies

Replacing the MOSFET in Fig. 7.56(a) by its high-frequency model in Fig. 7.52(a) gives the high-frequency equivalent circuit shown in Fig. 7.58(a). If we assume $C_{gs}$ is open-circuited and $v_s = 0$, the equivalent circuit is shown in Fig. 7.58(b). Thevenin’s equivalent resistance presented to $C_{gd}$ is

$$R_{C_{gd}} = R_s \parallel R_G$$  \hspace{2cm} (7.127)

If we assume $C_{gd}$ is open-circuited, the equivalent circuit is shown in Fig. 7.58(c). To find $R_{C_{gs}}$, we remove $C_{gs}$ and apply a test voltage $v_x$, as shown in Fig. 7.58(d). Using KVL around the loop formed by $R_s$ in parallel with $R_G$ and by $R_L$ in parallel with $R_{SR}$, we get

$$v_x = (R_s \parallel R_G)i_x + (R_L \parallel R_{SR})(i_x - g_m v_x)$$

which can be simplified to

$$i_x(R_s \parallel R_G + R_L \parallel R_{SR}) = v_x[1 + g_m(R_L \parallel R_{SR})]$$

or

$$\frac{i_x}{v_x} = \frac{1 + g_m(R_L \parallel R_{SR})}{R_s \parallel R_G + R_L \parallel R_{SR}}$$

Thus, Thevenin’s equivalent resistance presented to $C_{gs}$ is

$$R_{C_{gs}} = \frac{v_x}{i_x} = \frac{R_s \parallel R_G + R_L \parallel R_{SR}}{1 + g_m(R_L \parallel R_{SR})}$$  \hspace{2cm} (7.128)

and the high 3-dB frequency is

$$f_H = \frac{1}{2\pi(R_{C_{gd}}C_{gd} + R_{C_{gs}}C_{gs})}$$  \hspace{2cm} (7.129)

**FIGURE 7.58**  High-frequency equivalent circuits of a common-drain MOSFET amplifier
EXAMPLE 7.14

Finding the high cutoff frequency of a common-drain MOSFET amplifier

The circuit parameters of the MOSFET amplifier in Fig. 7.56(a) are $C_{gd} = 2 \text{ pF}$, $C_{gs} = 5 \text{ pF}$, $R_s = 200 \text{ k}\Omega$, $g_m = 10 \times 10^{-3} \text{ A/V}$, $R_{SR} = 2 \text{ k}\Omega$, $R_L = 5 \text{ k}\Omega$, $R_1 = 200 \text{ k}\Omega$, and $R_2 = 200 \text{ k}\Omega$. Calculate the high 3-dB frequency $f_H$.

SOLUTION

$R_G = R_1 \parallel R_2 = 200 \text{ k}\Omega \parallel 200 \text{ k}\Omega = 100 \text{ k}\Omega$. From Eq. (7.127),

$$R_{C_{gd}} = 200 \text{ \Omega} \parallel 100 \text{ \Omega} = 199.6 \text{ \Omega}$$

From Eq. (7.128),

$$R_{C_{gs}} = \frac{200 \text{ \Omega} \parallel 100 \text{ \Omega} + 5 \text{ \Omega} \parallel 2 \text{ \Omega}}{1 + 10 \text{ \mu} \Omega \times (5 \text{ \Omega} \parallel 2 \text{ \Omega})} = 106.5 \text{ \Omega}$$

From Eq. (7.129), the high 3-dB frequency is

$$f_H = \frac{1}{2\pi (R_{C_{gd}}C_{gd} + R_{C_{gs}}C_{gs})} = \frac{1}{2\pi \times (199.6 \text{ \Omega} \times 2 \text{ pF} + 106.5 \text{ \Omega} \times 5 \text{ pF})} = 170.8 \text{ MHz}$$

7.13.5 Common-Gate Amplifiers

A common-gate MOSFET amplifier is shown in Fig. 7.59(a).

Low Cutoff Frequencies

Replacing the MOSFET in Fig. 7.59(a) by its low-frequency model gives the low-frequency equivalent circuit shown in Fig. 7.59(b), which contains two coupling capacitors $C_1$ and $C_2$. If we assume $C_2$ and $C_G$ are short-circuited, the equivalent circuit is shown in Fig. 7.60(a). The resistance representing the current source is $1/g_m$, and Thevenin’s equivalent resistance presented to $C_1$ is

$$R_{C_1} = R_s + \left( R_{SR} \parallel \frac{1}{g_m} \right)$$

(7.130)
If $C_1$ and $C_G$ are short-circuited, as shown in Fig. 7.60(b), Thevenin’s equivalent resistance presented to $C_2$ is

$$R_{C2} = R_D + R_L$$

(7.131)

If $C_1$ and $C_2$ are short-circuited, as shown in Fig. 7.60(c), Thevenin’s equivalent resistance becomes

$$R_{CS} = R_G = R_1 \parallel R_2$$

(7.132)

In general, $R_{CG} > R_{C2} > R_{C1}$, and $R_{C1}$ controls the low cutoff frequency.

**High Cutoff Frequencies**

The high-frequency equivalent circuit of the common-gate amplifier in Fig. 7.59(a) is shown in Fig. 7.61(a). The equivalent circuit, with $C_{gs}$ open-circuited, is shown in Fig. 7.61(b). $g_{mVgs}$ behaves as open circuit at $v_{gs} = 0$. Thevenin’s equivalent resistance presented to $C_{gd}$ is

$$R_{Cgd} = R_L \parallel R_D$$

(7.133)
If we assume $C_{gd}$ is open-circuited, the equivalent circuit is shown in Fig. 7.61(c). The resistance representing the current source is $1/gm$, which forms a parallel circuit with $R_s$ and $R_{SR}$. Thevenin’s equivalent resistance presented to $C_{gs}$ is given by

$$R_{Cgs} = R_s \parallel R_{SR} \parallel \frac{1}{g_m} \quad (7.134)$$

Thus, the high 3-dB frequency is given by

$$f_H = \frac{1}{2\pi(R_{Cgd}C_{gd} + R_{Cgs}C_{gs})} \quad (7.135)$$

**NOTE** $f_H$ is almost independent of the transistor gain $g_m$ since $R_{Cgs} << R_{Cgd}$ and there is no Miller’s capacitance multiplication effect. Common-gate amplifiers are used for high-frequency applications.

### EXAMPLE 7.15

**Finding the high cutoff frequency of a common-gate MOSFET amplifier** The circuit parameters of the MOSFET amplifier in Fig. 7.59(a) are $C_{gd} = 2$ pF, $C_{gs} = 5$ pF, $R_s = 200 \Omega$, $g_m = 10 \times 10^{-3}$ A/V, $R_{SR} = 2$ k$\Omega$, $R_D = R_L = 5$ k$\Omega$, $R_1 = 200$ k$\Omega$, and $R_2 = 200$ k$\Omega$. Calculate the high 3-dB frequency $f_H$.

**SOLUTION**

$$R_G = R_1 \parallel R_2 = 200 \Omega \parallel 200 \Omega = 100 \Omega.$$ From Eq. (7.133),

$$R_{Cgd} = 5 \text{ k}\Omega \parallel 5 \text{ k}\Omega = 2.5 \text{ k}\Omega$$

From Eq. (7.134),

$$R_{Cgs} = R_s \parallel R_{SR} \parallel \frac{1}{g_m} = 200 \Omega \parallel 2 \text{ k}\Omega \parallel \frac{1}{10 \text{ mA}/\text{V}} = 64.5 \Omega$$

From Eq. (7.135), the high 3-dB frequency is

$$f_H = \frac{1}{2\pi(R_{Cgd}C_{gd} + R_{Cgs}C_{gs})} = \frac{1}{2\pi(2.5 \text{ k}\Omega \times 2 \text{ pF} + 64.5 \Omega \times 5 \text{ pF})} = 29.9 \text{MHz}$$

### EXAMPLE 7.16

**Finding the frequency response of a two-stage CD–CS MOSFET amplifier** A two-stage MOSFET amplifier is shown in Fig. 7.62. The circuit parameters are $C_{gd1} = C_{gd2} = 2$ pF, $C_{gs1} = C_{gs2} = 5$ pF, $R_s = 200 \Omega$, $g_{m1} = g_{m2} = 10 \times 10^{-3}$ A/V, $R_s = 200 \Omega$, $R_{G1} = 50$ k$\Omega$, $R_{SR1} = 250$ k$\Omega$, $R_{D2} = 5$ k$\Omega$, $R_{SR2} = 150$ k$\Omega$, $R_L = 10$ k$\Omega$, $C_1 = 1 \mu\text{F}$, $C_2 = 10 \mu\text{F}$, and $C_{S2} = 5.3$ $\mu\text{F}$.

(a) Calculate the low 3-dB frequency $f_L$.

(b) Calculate the high 3-dB frequency $f_H$. 
(a) The low-frequency equivalent circuit is shown in Fig. 7.63(a). There are two coupling capacitors, $C_1$ and $C_2$, and one source bypass capacitor, $C_{S2}$.

The time constant $\tau_1$ due to $C_1$ is

$$\tau_1 = (R_s + R_{G1})C_1$$

$$= (200 \ \Omega + 50 \ \text{k}\Omega) \times 1 \ \mu\text{F} = 50.2 \ \text{ms}$$

The time constant $\tau_2$ due to $C_2$ is

$$\tau_2 = (R_{D2} + R_L)C_2$$

$$= (5 \ \text{k}\Omega + 10 \ \text{k}\Omega) \times 10 \ \mu\text{F} = 150 \ \text{ms}$$

FIGURE 7.62 Two-stage MOSFET amplifier

SOLUTION

(b) High-frequency equivalent circuit

FIGURE 7.63 Equivalent circuits for Fig. 7.62
The time constant \( \tau_3 \) due to \( C_{S2} \) is

\[
\tau_3 = (R_{SR2} \parallel \frac{1}{g_{m2}})C_{S2} \tag{7.138}
\]

\[
= [150 \, \Omega \parallel (1000/10 \, \text{A/V})] \times 5.3 \, \mu\text{F} = 0.32 \, \text{ms}
\]

From Eq. (2.106), the low 3-dB frequency is

\[
f_L = \frac{1}{2\pi} \left[ \frac{1}{50.2 \, \text{ms}} + \frac{1}{150 \, \text{ms}} + \frac{1}{0.32 \, \text{ms}} \right] = 502 \, \text{Hz}
\]

(b) The high-frequency equivalent circuit is shown in Fig. 7.63(b). Applying Eq. (7.128) gives Thevenin’s equivalent resistance presented to \( C_{gs1} \) as

\[
R_{gs1} = R_g \left[ \frac{R_{G1} + R_{SR1}}{1 + g_{m1}R_{SR1}} \right] \tag{7.139}
\]

\[
= \frac{200 \, \Omega \parallel 50 \, \text{k}\Omega + 250 \, \Omega}{1 + 10 \times 10^{-3} \, \text{A/V} \times 250 \, \Omega} = 128.3 \, \Omega
\]

and the time constant \( \tau_{gs1} \) is

\[
\tau_{gs1} = R_{gs1}C_{gs1} \tag{7.140}
\]

\[
= 128.3 \, \Omega \times 5 \, \text{pF} = 0.642 \, \text{ns}
\]

If \( R_{gs2} \) is Thevenin’s equivalent resistance faced by \( C_{gs2} \) with \( C_{gs1}, C_{gd1}, \) and \( C_{gd2} \) open-circuited, \( R_{gs2} \) will be the parallel combination of \( R_{SR1} \) and the output resistance of transistor M1. That is,

\[
R_{gs2} = R_{SR1} \parallel \frac{1}{g_{m1}} \tag{7.141}
\]

\[
= 250 \, \Omega \parallel \frac{1000}{10 \, \text{A/V}} = 71.4 \, \Omega
\]

and the time constant \( \tau_{gs2} \) is

\[
\tau_{gs2} = R_{gs2}C_{gs2} \tag{7.142}
\]

\[
= 71.4 \, \Omega \times 5 \, \text{pF} = 0.36 \, \text{ns}
\]

If \( R_{gd1} \) is Thevenin’s equivalent resistance faced by \( C_{gd1} \) with \( C_{gs1}, C_{gs2}, \) and \( C_{gd2} \) open-circuited, the time constant \( \tau_{gd1} \) is

\[
\tau_{gd1} = (R_g \parallel R_{G1})C_{gd1} \tag{7.143}
\]

\[
= (200 \, \Omega \parallel 50 \, \text{k}\Omega) \times 2 \, \text{pF} = 0.398 \, \text{ns}
\]

With \( C_{gs1}, C_{gs2}, \) and \( C_{gd1} \) open-circuited, Thevenin’s equivalent resistance faced by \( C_{gd2} \) can be found by applying Eq. (2.116):

\[
R_{gd2} = (R_{D2} \parallel R_L) + R_{gs2}[1 + g_{m2}(R_{D2} \parallel R_L)] \tag{7.144}
\]

\[
= (5 \, \text{k}\Omega \parallel 10 \, \text{k}\Omega) + 71.4 \, \Omega \times [1 + 10 \times 10^{-3} \, \text{A/V} \times (5 \, \text{k}\Omega \parallel 10 \, \text{k}\Omega)] = 5.78 \, \text{k}\Omega
7.14 Design of MOSFET Amplifiers

When an amplifier is being analyzed, the components are specified; however, when an amplifier is being designed, the designer must select the values of the circuit components. The design task can be simplified if a simple transistor model is used to find approximate values of the components. After the initial design stage, the next step is to analyze the amplifier with these approximate values and to compare the performance parameters with the desired values. Often the specifications are not met, and it is necessary to modify the component values. An amplifier is normally specified by the input resistance $R_i$, the output resistance $R_o$, and the voltage gain $A_{vo}$. These specifications are normally defined by the following values:

Source resistance $R_s$

DC supply voltages $V_{DD}$ and $V_{SS}$ for MOSFETs

Load resistance $R_L$

Overall voltage gain $A_v$ ($=v_L/v_s$) (at a specified $R_L$)

Output resistance $R_o$

Input resistance $R_i$

After the specifications of an amplifier have been established, we will develop the necessary design conditions and the steps in meeting design specifications.

We have noted that the technique of DC analysis differs from that of AC analysis. For DC analysis, the load line is set by the DC resistance $R_{dc}$. That is,

$$R_{dc} = \begin{cases} R_D + R_{SR} & \text{for the CS amplifier of Fig. 7.34(a)} \\ R_{SR} & \text{for the CD amplifier of Fig. 7.42(a)} \end{cases}$$

(7.146)
For AC analysis, the load line is set by the AC resistance. That is,

\[ R_{ac} = \begin{cases} \frac{R_D}{R_L} & \text{for the CS amplifier of Fig. 7.34(a)} \\ \frac{R_{SR}}{R_L} & \text{for the CD amplifier of Fig. 7.42(a)} \end{cases} \] (7.147)

Under the no-load condition, the load resistance \( R_L \) is disconnected; the AC resistance \( R_{ac} \) equals \( R_D \). Thus, there are two load lines that must be considered in designing an amplifier circuit. So far, we have considered the DC load line only while designing a biasing circuit. The AC and DC load lines for CS amplifiers are shown in Fig. 7.64. The \( Q \)-point, which is specified for a zero AC input signal, lies on both the AC and the DC load lines. The AC load line passes through the \( Q \)-point and has a slope of \(-1/R_{ac}\). The slope of the AC line is greater in magnitude than that of the DC line. The AC load line may be described by

\[ i_D = i_D = \frac{v_{DS} - V_{DS}}{R_{ac}} \]

which gives

\[ i_D = -\frac{v_{DS}}{R_{ac}} + \left( \frac{V_{DS}}{R_{ac}} + i_D \right) \] (7.148)

The maximum AC drain current \( i_{D(max)} \), which occurs at \( v_{DS} = 0 \), can be found from Eq. (7.148):

\[ i_{D(max)} = \frac{V_{DS}}{R_{ac}} + i_D \] (7.149)
From Eq. (7.149), the quiescent drain current $I_D$ can be related to the AC and DC load lines by

$$\frac{V_{DD}}{I_D} = R_{dc} + R_{ac} \tag{7.150}$$

The input resistance of MOSFETs is high and can be selected independently of the voltage gain. MOSFET amplifiers are normally designed to provide a specified voltage gain $A_V$. Three possible circuit configurations are shown in Fig. 7.65. $R_{SR} (=R_{SR1} + R_{SR2})$ provides the required biasing voltage, and $R_{SR1}$ gives the

---

**FIGURE 7.65** Circuit configurations for MOSFET amplifiers
necessary voltage gain $A_{vo}$. After establishing the specifications of the amplifier, choose a suitable MOSFET and note its particular pinch-down voltage $V_p$ (or threshold voltage $V_t$), drain current $I_{DSS}$ (for $v_{GS} = 0$) (or MOSFET constant $K_n$), and channel modulation voltage $V_M$ (or assume a typical value of 200 V). Then choose the drain current $I_D$ at the $Q$-point. When choosing $I_D$, find the maximum value of $I_D(\text{max})$ from the data sheet for the transistor you have in mind. Then choose $I_D \leq I_D(\text{max})/2$ and the circuit topology of Fig. 7.65[(a), (b), or (c)].

The design steps required to accomplish the specifications are as follows:

**Step 1.** Using either Eq. (7.8) or Eq. (7.21), find the gate-to-source voltage $V_{GS}$ for known values of $I_D$, $I_{DSS}$, $V_t$, and $V_p$.

$$I_D = \begin{cases} K_n(V_{GS} - V_t)^2 & \text{for enhancement MOSFETs} \\ K_n(V_{GS} - V_p)^2 & \text{for depletion MOSFETs} \end{cases}$$

**Step 2.** For the known value of $V_{GS}$, calculate $R_{SR}$. One method is to use

$$V_{GS} = R_{SR} |I_D| \quad \text{for depletion MOSFETs as in Fig. 7.65(a)}$$

For other configurations, use

$$V_{SR} = \frac{V_{DD}}{3} = R_{SR}I_D$$

which gives $R_{SR} = V_{DD}/(3I_D)$, and

$$\begin{cases} \frac{R_2 V_{DD}}{R_1 + R_2} - R_{SR}I_D & \text{for MOSFETs as in Fig. 7.65(b)} \\ V_{DD} - V_{SR} = V_{DD} - R_{SR}I_D & \text{for MOSFETs as in Fig. 7.65(c)} \end{cases}$$

**Step 3.** From Eq. (7.26), calculate the output resistance $r_o$ of the MOSFET:

$$r_o = \frac{|V_M|}{i_D}$$

**Step 4.** From either Eq. (7.28) or Eq. (7.31), calculate the transconductance $g_m$ of the MOSFET:

$$g_m = \begin{cases} g_{mo} \left(1 - \frac{V_{GS}}{|V_p|}\right) & \text{for depletion MOSFETs} \\ g_{mo} \left(1 - \frac{V_{GS}}{|V_t|}\right) & \text{for enhancement MOSFETs} \end{cases}$$

where

$$g_{mo} = \begin{cases} -2K_nV_p = \frac{2I_{DSS}}{|V_p|} & \text{for depletion MOSFETs} \\ -2K_n|V_t| & \text{for enhancement MOSFETs} \end{cases}$$
Step 5. Calculate the gate resistance $R_G$ or resistances $R_1$ and $R_2$. Calculate $R_G$ from

$$R_G = R_i \quad \text{for MOSFETs as in Fig. 7.65(a)}$$

We set

$$v_G = \frac{V_{DD}R_2}{R_1 + R_2} = \frac{V_{DD}}{1 + R_1/R_2}$$

$$R_G = \frac{R_1R_2}{R_1 + R_2}$$

which can be solved to calculate $R_1$ and $R_2$

$$R_1 = \frac{R_iV_{DD}}{V_G} \quad \text{for MOSFETs as in Fig. 7.65(b)}$$

$$R_2 = \frac{R_iV_{DD}}{V_{DD} - V_G} \quad \text{for MOSFETs as in Fig. 7.65(b)}$$

where $V_G = V_{SR} + V_{GS}$ and $V_{SR}$ is the DC voltage at the source terminal.

Step 6. For known values of $R_L$, $I_D$, $V_{DD}$, and $R_{SR}$, calculate the drain resistance $R_D$. With $R_{dc} = R_D + R_{SR}$ and $R_{ac} = R_D \parallel R_L$, Eq. (7.150) gives

$$\frac{V_{DD}}{I_D} = R_{dc} + R_{ac} = R_D + R_{SR} + \frac{R_D R_L}{R_D + R_L}$$

Step 7. Assuming a voltage gain $A_{vo}$, let the no-load voltage gain $A_{vo}$ be equal to $A_v$. That is, let $A_{vo} = A_v$ as the first approximation. From Eq. (7.70), the no-load voltage gain $A_{vo}$ is given by

$$|A_{vo}| = \frac{v_o}{v_g} = \frac{\mu g R_D}{R_D + r_o + (1 + \mu g) R_{SR1}}$$

from which the source resistance $R_{SR1}$ can be found:

$$R_{SR1} = \frac{\mu g R_D - |A_{vo}| (R_D + r_o)}{|A_{vo}| (1 + \mu g)}$$

where $\mu g = g_m r_o$.

Step 8. Calculate the value of bypassed source resistance $R_{SR2}$:

$$R_{SR2} = R_{SR} - R_{SR1}$$

If $R_{SR2} < 0$, $A_{vo}$ is too high; choose a transistor with a higher value of $g_m$.

Step 9. Using Eq. (7.65), calculate the output resistance $R_o$:

$$R_o = [r_o + (1 + \mu g) R_{SR1}] \parallel R_D$$
Step 10. Calculate the voltage gain $A_v$:

$$A_v = \frac{v_L}{v_s} = \frac{A_{vo}R_iR_L}{(R_i + R_s)(R_L + R_o)}$$

Step 11. If the value of $A_v$ in step 10 is not greater than or equal to the desired value of $A_{vo}$, repeat steps 7 through 10 with progressively higher values of $A_{vo}$ until you obtain the desired value for $A_v$ in step 10. If the gain requirement cannot be obtained, choose a transistor with a higher value of $g_m$.

**KEY POINTS OF SECTION 7.14**

- In general, designing involves decision making and an iterative process. The design steps developed in this section will be helpful in finding component values to satisfy specifications.
- Designing an amplifier requires prior knowledge of desired specifications, choice of a MOSFET, and choice of a $Q$-point.
- Once the type of transistor and the $Q$-point have been chosen, the next step is to choose the biasing circuit and find its component values.
- The small-signal parameters, which are calculated from the values of the $Q$-point, are then used to find the emitter (or source) resistance needed to obtain the desired voltage gain or input resistance.

**Summary**

MOSFETs, which are voltage-dependent devices, are of two types: junction MOSFETs and MOSFETs. MOSFETs are of two types: enhancement and depletion. Each type can be either $p$-channel or $n$-channel. Depending on the value of the drain-to-source voltage, a MOSFET can operate in one of three regions: ohmic, saturation, or cutoff. In the ohmic region, a MOSFET is operated as a voltage-controlled device. In the saturation region, a MOSFET is operated as an amplifier. An enhancement MOSFET conducts only when the gate-to-source voltage exceeds the threshold voltage. The gate current of a MOSFET is very small (on the order of nA). A MOSFET can be modeled by a voltage-controlled current source. MOSFETs should be biased properly to set the gate-to-source voltage in appropriate polarity and magnitude. The $Q$-point should be stable, and a biasing circuit should be designed to minimize the effect of parameter variations. MOSFETs are widely used in very-large-scale integrated (VLSI) circuits.

**References**


**Review Questions**

1. What is a DC load line?
2. What is an AC load line?
3. What are the advantages of MOSFETs?
4. What are the types of MOSFETs?
5. What is an NMOS?
6. What is a PMOS?
7. What is the ohmic region of a MOSFET?
8. What are the effects of MOSFET characteristics on the biasing point?
9. What is the transconductance gain $g_m$ of a MOSFET?
10. What is the small-signal output resistance $r_o$ of a MOSFET?
11. What is the channel modulation voltage of a MOSFET?
12. What is the purpose of a source-bypassed capacitor?
13. What are the performance parameters of an amplifier?
14. What are the characteristics of CS-configuration amplifiers?
15. What are the characteristics of source followers?

**Problems**

The symbol $\textbf{P}$ indicates that a problem is a design problem. The symbol $\textbf{D}$ indicates that you can check the solution to a problem using PSpice/SPICE or Electronics Workbench.

7.2 and 7.4 *Enhancement and Depletion MOSFETs*

7.1 An NMOS has a channel width of $W = 40 \ \mu m$ and a channel length of $L = 2 \ \mu m$; the thickness of the silicon dioxide is $t_{ox} = 10\ nm$, the dielectric constant of the silicon dioxide layer is $\epsilon_{ox} = 4$, and the mobility of the electrons in the inversion layer is $\mu_n = 500\ cm^2/(volt-sec)$. Determine the MOS constants $K_n$ and $K_p$.

7.2 An NMOS has a substrate impurity doping concentration of $N_a = 2 \times 10^{16}\ cm^{-3}$, a threshold voltage of $V_{IN} = 1\ V$, and a channel length of $L = 10\ \mu m$; $V_{GS} = 2.5\ V$ and $V_{DS} = 5\ V$. Determine the channel modulation voltage $V_M$.

7.3 An NMOS has a substrate impurity doping concentration of $N_a = 2 \times 10^{16}\ cm^{-3}$, a threshold voltage of $V_{IN} = 1\ V$, and a channel length of $L = 10\ \mu m$; $V_{GS} = 2.5\ V$. Plot the channel modulation voltage $V_M$ for $V_{DS} = 5\ V$ to $20\ V$.

7.4 An NMOS has a substrate impurity doping concentration of $N_a = 2 \times 10^{16}\ cm^{-3}$ and an intrinsic concentration of $n_i = 1.5 \times 10^{10}\ cm^{-3}$; $T = 25^\circ C$. (a) Determine the depletion width $x_p$ extending to the $p$-region substrate for $V_{DS} = 10\ V$, and (b) plot $x_p$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
7.5 An NMOS has a drain current of $I_{D1} = 1$ mA at $V_{GS1} = 1.5$ V and $I_{D2} = 2.5$ mA at $V_{GS2} = 2.5$ V. If the NMOS operates in the saturation region, determine (a) its threshold voltage $V_t$, (b) the MOS constant $K_n$, (c) the drain current $I_D$ at $V_{GS} = 2$ V, and (d) the minimum drain-to-source voltage $V_{DS(sat)}$ at $V_{GS} = 2$ V to operate in the saturation region.

7.6 A PMOS has a drain current of $I_{D1} = 1$ mA at $V_{GS1} = -1.5$ V; $I_{D2} = 2.5$ mA at $V_{GS2} = -2.5$ V. If the PMOS operates in the saturation region, determine (a) its threshold voltage $V_t$, (b) the MOS constant $K_n$, (c) the drain current $I_D$ at $V_{GS} = -2$ V, and (d) the minimum drain-to-source voltage $V_{DS(sat)}$ at $V_{GS} = -2$ V to operate in the saturation region.

7.7 A depletion NMOS has a drain current of $I_{D1} = 1$ mA at $V_{GS1} = -2.5$ V, and $I_{D2} = 2.5$ mA at $V_{GS2} = -1$ V. If the NMOS operates in the saturation region, determine (a) its pinch-off voltage $V_{p}$, (b) the MOS constant $K_n$, (c) the drain current $I_D$ at $V_{GS} = -1.5$ V, and (d) the minimum drain-to-source voltage $V_{DS(sat)}$ at $V_{GS} = -1.5$ V to operate in the saturation region.

7.8 A depletion PMOS has a drain current of $I_{D1} = 1$ mA at $V_{GS1} = 2.5$ V, and $I_{D2} = 2.5$ mA at $V_{GS2} = 1.5$ V. If the PMOS operates in the saturation region, determine (a) its pinch-off voltage $V_{p}$, (b) the MOS constant $K_n$, (c) the drain current $I_D$ at $V_{GS} = 2$ V, and (d) the minimum drain-to-source voltage $V_{DS(sat)}$ at $V_{GS} = 2$ V to operate in the saturation region.

7.9 An NMOS has a substrate impurity doping concentration of $N_a = 2 \times 10^{16}$ cm$^{-3}$ and an intrinsic concentration of $N_i = 1.5 \times 10^{10}$ cm$^{-3}$; $T = 25$°C, and the oxide thickness $t_{ox} = 0.10$ μm. The threshold voltage is $V_o = 1$ V at $V_{SB} = 0$ V. Plot $V_t$ for $V_{SB} = 5$ V to 30 V.

7.10 An NMOS has a threshold voltage $V_t = 1$ V and a MOS constant $K_n = 0.5$ mA/V$^2$. It operates in the ohmic region and offers a drain-to-source resistance $R_{ds} = 50$ Ω at $V_{DS} = 2$ V. Determine the gate-to-source voltage $V_{GS}$.

7.11 An NMOS has a threshold voltage $V_t = 1$ V and MOS constant $K_n = 0.5$ mA/V$^2$. It operates in the ohmic region and offers a drain-to-source resistance $R_{ds} = 100$ Ω at $V_{GS} = 1.5$ V. Determine the drain-to-source voltage $V_{DS}$.

7.7 DC Biasing of MOSFETs

7.12 The NMOS biasing circuit in Fig. P7.12 has $R_D = 1.5$ kΩ, $R_G = 500$ kΩ, and $V_{DD} = 12$ V. The MOS parameters are $K_n = 0.5$ mA/V$^2$, $V_t = 1$ V, and $\lambda = 0.01$. Determine (a) the drain current $I_D$, (b) the gate-to-source voltage $V_{GS}$, (c) the drain-to-source voltage $V_{DS}$, (d) the small-signal transconductance $g_m$, and (e) the output resistance $r_o$.

**FIGURE P7.12**

7.13 The NMOS biasing circuit in Fig. P7.13 has $R_D = 1.5$ kΩ, $R_{SR} = 500$ Ω, $R_G = 500$ kΩ, and $V_{DD} = 12$ V. The MOS parameters are $K_n = 0.5$ mA/V$^2$, $V_t = 1$ V, and $\lambda = 0.01$. Determine (a) the drain current $I_D$, (b) the gate-to-source voltage $V_{GS}$, (c) the drain-to-source voltage $V_{DS}$, (d) the small-signal transconductance $g_m$, and (e) the output resistance $r_o$.
7.14 The MOS biasing circuit in Fig. P7.14 has \( R_D = 1.5 \text{k}\Omega, R_{SR} = 500 \Omega, R_1 = 400 \text{k}\Omega, R_2 = 600 \text{k}\Omega, \) and \( V_{DD} = 12 \text{V}. \) The MOS parameters are \( K_n = 0.5 \text{mA/V}^2, V_t = -1.5 \text{V}, \) and \( \lambda = 0.01. \) Determine (a) the drain current \( I_D, \) (b) the gate-to-source voltage \( V_{GS}, \) (c) the drain-to-source voltage \( V_{DS}, \) (d) the small-signal transconductance \( g_m, \) and (e) the output resistance \( r_o. \)

![FIGURE P7.13](image)

7.15 The pinch-down voltage of an \( n \)-channel depletion NMOS is \( V_p = -5 \text{V}, \) and the saturation current is \( I_{DSS} = 40 \text{mA}. \) The value of \( V_{DS} \) is such that the transistor is operating in the saturation region. The drain current is \( i_D = 15 \text{mA}. \) Calculate the gate-to-source voltage \( V_{GS}. \)

7.16 The pinch-down voltage of a \( p \)-channel depletion NMOS is \( V_p = 5 \text{V}, \) and the saturation current is \( I_{DSS} = -40 \text{mA}. \) The value of \( V_{DS} \) is such that the transistor is operating in the saturation region. The drain current is \( i_D = -15 \text{mA}. \) Calculate the gate-to-source voltage \( V_{GS}. \)

7.17 An \( n \)-channel enhancement MOSFET has \( V_t = 3.5 \text{V} \) and \( i_D = 8 \text{mA} \) (at \( V_{GS} = 5.8 \text{V} \)). Find (a) \( i_D \) when \( V_{GS} = 5 \text{V}, \) (b) \( V_{GS} \) when \( i_D = 6 \text{mA}, \) (c) the value of \( V_{DS} \) at the boundary between the ohmic and saturation regions if \( i_D = 6 \text{mA}, \) and (d) the ratio \( W/L \) if \( \mu_n = 600 \text{cm}^2/\text{volt-sec}, \) \( t_{ox} = 0.1 \mu\text{m}, \) and \( C_{ox} = 3.5 \times 10^{-11} \text{F/cm}^2. \) Assume operation in the saturation region.

7.18 A \( p \)-channel enhancement MOSFET has \( V_t = -3.5 \text{V} \) and \( i_D = -8 \text{mA} \) (at \( V_{GS} = -5.8 \text{V} \)). Find (a) \( i_D \) when \( V_{GS} = -5 \text{V}, \) (b) \( V_{GS} \) when \( i_D = -6 \text{mA}, \) (c) the value of \( V_{DS} \) at the boundary between the ohmic and saturation regions when \( i_D = -6 \text{mA}, \) and (d) the ratio \( W/L \) if \( \mu_n = 600 \text{cm}^2/\text{volt-sec}, \) \( t_{ox} = 0.1 \mu\text{m}, \) and \( C_{ox} = 3.5 \times 10^{-11} \text{F/cm}^2. \)

7.19 An \( n \)-channel depletion MOSFET has \( V_p = -5 \text{V} \) and \( i_D = 0.5 \text{mA} \) (at \( V_{GS} = -4 \text{V} \)). Find (a) \( i_D \) when \( V_{GS} = -2 \text{V}, \) (b) \( V_{GS} \) when \( i_D = 6 \text{mA}, \) (c) the value of \( V_{DS} \) at the boundary between the ohmic and saturation regions when \( i_D = 6 \text{mA}, \) and (d) the ratio \( W/L \) if \( \mu_n = 600 \text{cm}^2/\text{volt-sec} \) and \( C_{ox} = 3.5 \times 10^{-11} \text{F/cm}^2. \) Assume operation in the saturation region.

7.20 The \( n \)-channel depletion NMOS circuit of Fig. 7.25(a) has \( R_D = 1.5 \text{k}\Omega, R_{SR} = 1 \text{k}\Omega, R_1 = \infty, R_2 = 500 \text{k}\Omega, \) and \( V_{DD} = 15 \text{V}. \) Calculate \( I_D, V_{GS}, \) and \( V_{DS} \) if (a) \( I_{DSS} = 25 \text{mA} \) and \( i_D = 0.5 \text{mA} \) (at \( V_{GS} = -6.5 \text{V} \)) and (b) \( I_{DSS} = 5 \text{mA} \) and \( i_D = 0.5 \text{mA} \) (at \( V_{GS} = -1.5 \text{V} \)).
7.21 The biasing circuit for the $n$-channel depletion NMOS of Fig. 7.25(a) has $R_1 = 350 \, \text{k}\Omega$, $R_2 = 100 \, \text{k}\Omega$, $V_{DD} = 15 \, \text{V}$, $R_D = 1.5 \, \text{k}\Omega$, and $R_{SR} = 2.3 \, \text{k}\Omega$. The transistor parameters are $I_{DSS} = 15 \, \text{mA}$ and $V_p = -4.5 \, \text{V}$. Assume operation in the saturation region.

a. Calculate the values of $I_D$, $V_{DS}$, and $V_{GS}$ at the $Q$-point.

b. Calculate the minimum value of $R_{SR}$ so that $V_{GS} \leq 0$.

c. Use PSpice/SPICE to verify your design in part (a).

7.22 An $n$-channel depletion NMOS amplifier is shown in Fig. P7.22(a). The drain characteristic is shown in Fig. P7.22(b). The quiescent values are $I_D = 5 \, \text{mA}$, $V_{DS} = 10 \, \text{V}$, and $V_{GS} = -2 \, \text{V}$. Calculate the values of $R_D$ and $R_{SR}$.

7.23 For the $n$-channel depletion NMOS circuit shown in Fig. P7.23, $R_D = 2.5 \, \text{k}\Omega$ and $V_{DD} = 18 \, \text{V}$. The parameters of the depletion NMOS are $V_p = -1.5 \, \text{V}$ and $I_{DSS} = 5 \, \text{mA}$. Calculate the quiescent values of $I_D$, $V_{DS}$, and $V_{GS}$.

7.24 For the $n$-channel depletion NMOS circuit shown in Fig. P7.24, the quiescent values are $I_D = 7.5 \, \text{mA}$ and $V_{DS} = 10 \, \text{V}$. The parameters of the MOSFET are $I_{DSS} = 10 \, \text{mA}$ and $V_p = -5 \, \text{V}$. If the drain characteristic is described by

$$i_D = I_{DSS} \left[1 - \frac{V_{GS}}{V_p}\right]^2$$

calculate (a) the quiescent value of $V_{GS}$ and (b) the values of $R_{SR}$ and $R_D$. Assume $V_{DD} = 20 \, \text{V}$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
7.25 The NMOS biasing circuit shown in Fig. 7.25(a) has $V_{DD} = 15 \text{ V}$, $R_1 = 400 \text{ k}\Omega$, $R_2 = 150 \text{ k}\Omega$, $R_D = 2.5 \text{ k}\Omega$, and $R_{SR} = 4 \text{ k}\Omega$. The parameters of the NMOS are $V_t = 2.5 \text{ V}$ and $K_p = 1 \text{ mA/V}^2$. Calculate $V_{DS}$ and $V_{GS}$.

7.26 Design a biasing circuit as shown in Fig. 7.25(a) for an $n$-channel depletion NMOS. The operating point must be maintained at $I_D = 8 \text{ mA}$ and $V_{DS} = 7.5 \text{ V}$. The DC supply voltage is $15 \text{ V}$. The MOSFET parameters are $I_{DSS} = 15 \text{ mA}$ and $V_p = -5 \text{ V}$. Assume operation in the saturation region.

7.27 For the biasing circuit for an NMOS shown in Fig. 7.25(a), $V_t$ varies from 1 V to 2.5 V and $K_p$ varies from 200 $\mu$A/V$^2$ to 150 $\mu$A/V$^2$. If the variation of the drain current must be limited to 350 $\mu$A $\pm$ 20%, calculate the values of $R_{SR}$, $R_1$, $R_2$, and $R_D$.

7.28 A circuit for an $n$-channel depletion MOSFET is shown in Fig. P7.28. The transistor parameters are $V_p = -5 \text{ V}$ and $I_{DSS} = 10 \text{ mA}$. Calculate the quiescent values of $I_D$, $V_{DS}$, and $V_{GS}$. Assume $R_1 = 1 \text{ M}\Omega$, $R_2 = 60 \text{ k}\Omega$, and $R_D = 1 \text{ k}\Omega$.

7.29 A circuit for an $n$-channel enhancement-type MOSFET is shown in Fig. P7.29. The parameters of the NMOS are $V_t = 4 \text{ V}$ and $K_n = 1.2 \text{ mA/V}^2$. If the quiescent values are to be set at $I_D = 10 \text{ mA}$ and $V_{DS} = 8 \text{ V}$, calculate the values of $R_1$, $R_2$, and $R_D$. Assume $V_{DD} = 20 \text{ V}$.
7.30 Plot the approximate transfer characteristic of the NMOS circuit of Fig. P7.30 for $V_i = 0$ to 5 V. The circuit parameters are $R_D = 25 \, \text{k}\Omega$, $K_n = 20 \, \mu\text{A}/\text{V}^2$ ($v_O \text{ versus } v_I$), and $V_t = 2$ V.

**FIGURE P7.30**

```
+V_{dd} = 5 V

\[ R_D \]

\[ \rightarrow \]

\[ + \]

\[ M_1 \]

\[ \rightarrow \]

\[ \downarrow \]

\[ \downarrow \]

\[ v_I \]

\[ \leftarrow \]

\[ v_O \]

\[ \rightarrow \]

\[ - \]
```

7.31 The parameters of the NMOS circuit shown in Fig. P7.31 are $K_n = 1 \, \text{mA}/\text{V}^2$, $V_t = 2$ V, and $V_{DD} = 12$ V. Determine the values of $V_O$, $I_D$, and $V_{DS}$.

**FIGURE P7.31**

```
+V_{dd} = 12 V

\[ R_{SR} \]

\[ 10 \, \text{k}\Omega \]

\[ \rightarrow \]

\[ \downarrow \]

\[ \downarrow \]

\[ M_1 \]

\[ \rightarrow \]

\[ \downarrow \]

\[ \downarrow \]

\[ I_D \]

\[ \leftarrow \]

\[ \downarrow \]

\[ \downarrow \]
```

7.32 The parameters of the NMOS circuit in Fig. P7.31 are $K_n = 1 \, \text{mA}/\text{V}^2$, $V_t = 2$ V, and $V_{DD} = 12$ V. Determine the value of $R_{SR}$ so that $V_O = 5$ V.

7.33 The parameters of the MOSFET circuit shown in Fig. P7.33 are $K_n = 1.5 \, \text{mA}/\text{V}^2$, $V_t = -2$ V, $R_{SR} = 1.5 \, \text{k}\Omega$, and $V_{DD} = 12$ V. Determine the values of $V_O$, $I_D$, and $V_{DS}$.

**FIGURE P7.33**

```
+V_{dd} = 12 V

\[ R_{SR} \]

\[ \rightarrow \]

\[ \downarrow \]

\[ \downarrow \]

\[ M_1 \]

\[ \rightarrow \]

\[ \downarrow \]

\[ \downarrow \]

\[ V_O \]

\[ \leftarrow \]

\[ \downarrow \]

\[ \downarrow \]
```

7.8–7.10 **MOSFET Amplifiers**

7.34 The depletion NMOS amplifier of Fig. P7.34 has $R_s = 500 \, \Omega$, $R_L = 10 \, \text{k}\Omega$, $R_{SR} = R_D = 5 \, \text{k}\Omega$, $R_G = 100 \, \text{k}\Omega$, $I_{DSS} = 10 \, \text{mA}$, $V_p = -4$ V, $|V_M| = 200$ V, and $V_{DD} = 12$ V. Calculate (a) the input resistance $R_{in} = v_s/\dot{i}_n$, (b) the no-load voltage gain $A_v = v_o/\dot{i}_g$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L/v_s$. 

---

**Note:** The diagrams and specific calculations are not rendered in this text representation. The text format is designed to represent the natural English text as accurately as possible without visual aids. For full understanding, visual presentation is recommended.
7.35 The MOSFET amplifier of Fig. P7.35 has $R_s = 500 \, \Omega$, $R_D = R_L = 5 \, k\Omega$, $R_{G1} = 7 \, M\Omega$, $R_{G2} = 5 \, M\Omega$, $K_p = 20 \, mA/V^2$, $V_i = 3.5 \, V$, $|V_M| = 200 \, V$, and $V_{DD} = 12 \, V$. Calculate (a) the input resistance $R_{in} = v_i / i_s$, (b) the no-load voltage gain $A_{vo} = v_o / v_g$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L / v_s$.

7.36 The NMOS amplifier of Fig. P7.36 has $V_{DD} = 15 \, V$, $R_s = 500 \, \Omega$, $R_L = 10 \, k\Omega$, $R_{SR} = 3 \, k\Omega$, $R_D = 5 \, k\Omega$, $R_{G1} = 700 \, k\Omega$, $R_{G2} = 300 \, k\Omega$, $V_M = -150 \, V$, $V_i = 2.4 \, V$, and $K_n = 2.042 \, mA/V^2$. Calculate (a) the input resistance $R_{in} = v_i / i_s$, (b) the no-load voltage gain $A_{vo} = v_o / v_g$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L / v_s$. 
7.37 The NMOS amplifier of Fig. P7.36 has $V_{DD} = 15$ V, $R_s = 1$ kΩ, $R_L = 5$ kΩ, $R_{SR} = 1$ kΩ, $R_D = 5$ kΩ, $R_{G1} = 400$ MΩ, $R_{G2} = 600$ MΩ, $V_M = -100$ V, $V_i = 2$ V, and $K_p = 10$ mA/V$^2$. Calculate (a) the input resistance $R_{in} = v_s/i_s$, (b) the no-load voltage gain $A_{vo} = v_o/v_g$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L/v_s$.

7.38 The MOSFET amplifier of Fig. P7.38 has $R_s = 500$ Ω, $R_1 = 30$ kΩ, $R_2 = 50$ kΩ, $R_D = 10$ kΩ, and $R_L = 15$ kΩ. Assume $V_M = -200$ V, $V_i = 2$ V, and $K_n = 30$ mA/V$^2$. Calculate (a) the input resistance $R_{in} = v_s/i_s$, (b) the no-load voltage gain $A_{vo} = v_o/v_g$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L/v_s$.

**FIGURE P7.38**

7.39 The parameters of the NMOS amplifier in Fig. P7.39 are $V_{DD} = 15$ V, $R_1 = 600$ kΩ, $R_2 = 400$ kΩ, $R_L = 20$ kΩ, $R_{SR1} = 100$ Ω, $R_{SR2} = 900$ Ω, $R_D = 2.5$ kΩ, $C_1 = C_2 = C_s = \infty$, $|V_M| = \infty$, $K_n = 1$ mA/V$^2$, and $V_i = 1.5$ V. Calculate (a) the input resistance $R_1 = v_s/i_s$, (b) the no-load voltage gain $A_o = v_o/v_g$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L/v_s$.

**FIGURE P7.39**

7.40 The parameters of the PMOS amplifier in Fig. P7.40 are $V_{DD} = 15$ V, $R_L = 50$ MΩ, $R_G = 500$ kΩ, $R_{SR1} = 500$ Ω, $R_D = 2.5$ kΩ, $C_1 = C_2 = C_s = \infty$, $|V_M| = \infty$, $K_n = 1$ mA/V$^2$, and $V_i = -1.5$ V. Calculate (a) the input resistance $R_1 = v_s/i_s$, (b) the no-load voltage gain $A_o = v_o/v_g$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L/v_s$. 
7.41 The parameters of the PMOS amplifier in Fig. P7.41 are $V_{DD} = 15$ V, $R_1 = 500$ kΩ, $R_2 = 800$ kΩ, $R_L = 20$ kΩ, $R_{SR1} = 100$ Ω, $R_{SR2} = 900$ Ω, $R_D = 2.5$ kΩ, $C_1 = C_2 = C_s \approx \infty$, $|V_M| \approx \infty$, $K_n = 1$ mA/V², and $V_i = -2$ V. Calculate (a) the input resistance $R_i = v_i/i_s$, (b) the no-load voltage gain $A_o = v_o/v_g$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L/v_g$.

7.42 The parameters of the PMOS amplifier in Fig. P7.42 are $V_{DD} = 15$ V, $R_1 = 300$ kΩ, $R_2 = 700$ kΩ, $R_L = 20$ kΩ, $R_S = 1$ kΩ, $R_D = 2.5$ kΩ, $C_1 = C_2 = C_G \approx \infty$, $|V_M| \approx \infty$, $K_n = 1$ mA/V², and $V_i = -1.5$ V. Calculate (a) the input resistance $R_i = v_i/i_s$, (b) the no-load voltage gain $A_o = v_o/v_g$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L/v_g$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
7.43 The parameters of the NMOS amplifier in Fig. P7.43 are \( V_{DD} = 15 \text{ V}, \ R_1 = 700 \text{ k}\Omega, \ R_2 = 300 \text{ k}\Omega, \ R_L = 20 \text{ k}\Omega, \ R_{SR} = 1 \text{ k}\Omega, \ R_D = 2.5 \text{ k}\Omega, \ C_1 = C_2 \approx \infty, \ \left| V_M \right| \approx \infty, \ K_n = 1 \text{ mA/V}^2, \) and \( V_t = 1.7 \text{ V}. \) Calculate (a) the input resistance \( R_i = v_s/i_s, \) (b) the no-load voltage gain \( A_v = v_o/v_g, \) (c) the output resistance \( R_o, \) and (d) the overall voltage gain \( A_v = v_L/v_s. \)

![Figure P7.43](image)

7.44 The source follower of Fig. 7.42(a) has \( R_s = 1 \text{ k}\Omega, \ R_L = 1 \text{ k}\Omega, \ R_{SR} = 1 \text{ k}\Omega, \ R_1 = 700 \text{ k}\Omega, \ R_2 = 300 \text{ k}\Omega, \ I_{DSS} = 20 \text{ mA}, \ V_p = -4 \text{ V}, \ \left| V_M \right| = 200 \text{ V}, \) and \( V_{DD} = 12 \text{ V}. \) Calculate (a) the input resistance \( R_{in} = v_s/i_s, \) (b) the no-load voltage gain \( A_{vo} = v_o/v_g, \) (c) the output resistance \( R_o, \) and (d) the overall voltage gain \( A_v = v_L/v_s. \)

7.45 The depletion MOS source follower of Fig. P7.45 has \( R_s = 500 \text{ \Omega}, \ R_L = 10 \text{ k}\Omega, \ R_{SR} = 5 \text{ k}\Omega, \) and \( R_G = 10 \text{ M}\Omega. \) Assume \( V_p = -4 \text{ V}, \ V_M = -100 \text{ V}, \) and \( g_{mo} = 20 \text{ mA/V}. \) Calculate (a) the input resistance \( R_{in} = v_s/i_s, \) (b) the no-load voltage gain \( A_{vo} = v_o/v_g, \) (c) the output resistance \( R_o, \) and (d) the overall voltage gain \( A_v = v_L/v_s. \)

![Figure P7.45](image)

7.46 A depletion NMOS CS amplifier is shown in Fig. P7.46. The transistor parameters are \( V_p = -5 \text{ V}, \ I_{DSS} = 50 \text{ mA}, \) and \( V_M = -150 \text{ V}. \)

a. Calculate the small-signal parameters of the MOSFET.

b. Calculate the input resistance \( R_{in} = v_s/i_s, \) the output resistance \( R_o, \) the no-load voltage gain \( A_{vo} = v_o/v_g, \) and the overall voltage gain \( A_v = v_L/v_s. \)
A depletion NMOS source follower is shown in Fig. P7.47. The transistor parameters are $V_p = -5 \text{ V}$, $I_{DSS} = 50 \text{ mA}$, and $V_M = -150 \text{ V}$.

(a) Calculate the small-signal parameters of the MOSFET.

(b) Calculate the input resistance $R_{in} = v_s/i_s$, the output resistance $R_o$, the no-load voltage gain $A_{vo} = v_o/v_g$, and the overall voltage gain $A_v = v_L/v_s$.

An NMOS amplifier is shown in Fig. P7.48. The transistor parameters are $V_t = 4 \text{ V}$, $K_n = 50 \text{ mA}/V_2$, and $V_M = -150 \text{ V}$.

(a) Calculate the small-signal parameters of the MOSFET.

(b) Calculate the input resistance $R_{in} = v_s/i_s$, the output resistance $R_o$, the no-load voltage gain $A_{vo} = v_o/v_g$, and the overall voltage gain $A_v = v_L/v_s$. 
7.49 The parameters of the NMOS amplifier in Fig. P7.49 are \( V_{DD} = 15 \text{ V} \), \( R_1 = 700 \text{ k}\Omega \), \( R_2 = 300 \text{ k}\Omega \), \( R_L = 20 \text{ k}\Omega \), \( R_{SR} = 10 \text{ k}\Omega \), \( C_1 = C_2 \approx \infty \), \( |V_M| \approx \infty \), \( K_n = 1 \text{ mA/V}^2 \), and \( V_i = 1.7 \text{ V} \). Calculate (a) the input resistance \( R_i = v_s/i_s \), (b) the no-load voltage gain \( A_v = v_o/v_s \), (c) the output resistance \( R_o \), and (d) the overall voltage gain \( A_v = V_L/v_s \).

**FIGURE P7.49**

7.50 The parameters of the PMOS amplifier in Fig. P7.50 are \( V_{DD} = 15 \text{ V} \), \( R_1 = 400 \text{ k}\Omega \), \( R_2 = 600 \text{ k}\Omega \), \( R_L = 20 \text{ k}\Omega \), \( R_{SR} = 10 \text{ k}\Omega \), \( C_1 = C_2 \approx \infty \), \( |V_M| \approx \infty \), \( K_n = 1 \text{ mA/V}^2 \), and \( V_i = -2 \text{ V} \). Calculate (a) the input resistance \( R_i = v_s/i_s \), (b) the no-load voltage gain \( A_v = v_o/v_s \), (c) the output resistance \( R_o \), and (d) the overall voltage gain \( A_v = V_L/v_s \).

**FIGURE P7.50**

7.11 Multistage Amplifiers

7.51 The parameters of the MOS amplifier in Fig. P7.51 are \( V_{DD} = 15 \text{ V} \), \( R_1 = 700 \text{ k}\Omega \), \( R_2 = 300 \text{ k}\Omega \), \( R_L = 20 \text{ k}\Omega \), \( R_{SR1} = R_{SR2} = 1 \text{ k}\Omega \), \( R_{DI} = R_{D2} = 2.5 \text{ k}\Omega \), \( C_1 = C_2 \approx \infty \), \( |V_M| \approx \infty \), \( K_n = 1 \text{ mA/V}^2 \), \( V_{IN} = 1.7 \text{ V} \), and \( V_{IP} = -2 \text{ V} \). Calculate (a) the input resistance \( R_i = v_s/i_s \), (b) the no-load voltage gain \( A_v = v_o/v_s \), (c) the output resistance \( R_o \), and (d) the overall voltage gain \( A_v = V_L/v_s \).

**FIGURE P7.51**
7.52 The parameters of the MOS amplifier in Fig. P7.52 are $V_{DD} = 15$ V, $R_1 = R_2 = R_3 = 500$ kΩ, $R_L = 20$ kΩ, $R_{SR} = 500$ Ω, $R_D = 2.5$ kΩ, $C_1 = C_2 = C_3 = C_4 \approx \infty$, $|V_M| \approx \infty$, $K_n = 1$ mA/V², and $V_t = 1$ V. Calculate (a) the input resistance $R_i = v_s/i_s$, (b) the no-load voltage gain $A_o = v_o/v_g$, (c) the output resistance $R_o$, and (d) the overall voltage gain $A_v = v_L/v_s$.

FIGURE P7.52

7.53 A cascoded depletion MOS amplifier is shown in Fig. P7.53. The circuit parameters are $v_s = 2$ mV, $V_{DD} = 10$ V, $R_G = 20$ MΩ, $R_S = 500$ Ω, $R_{SR} = 500$ Ω, $R_D = 1$ kΩ, and $R_L = 10$ kΩ. The transistor parameters are $V_p = -4$ V, $I_{DSS} = 20$ mA, and $V_M = -150$ V. Calculate (a) the input resistance $R_{in} = v_s/i_s$, (b) the output resistance $R_o$, (c) the no-load voltage gain $A_{vo} = v_o/v_g$, and (d) the overall voltage gain $A_v = v_L/v_s$.

FIGURE P7.53

7.54 The parameters of the CMOS amplifier in Fig. P7.54 are $V_{DD} = 5$ V, $V_G = 2$ V, $I_{ref} = 0.5$ mA, $R_G = 500$ kΩ, $C_L \approx \infty$, $V_{MN} = -100$ V, $V_{MP} = 200$ V, $K_n = K_p = 0.5$ mA/V², $V_{ip} = -1.5$ V, and $V_{in} = 1$ V. Calculate the small-signal no-load voltage gain $A_{vo} = v_o/v_g$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
7.55 The parameters of the CMOS amplifier in Fig. P7.55 are $V_{DD} = 5$ V, $V_G = 2$ V, $I_{ref} = 0.5$ mA, $R_G = 500$ kΩ, $C_L \approx \infty$, $V_{MN} = -100$ V, $V_{MP} = 200$ V, $K_n = K_p = 0.5$ mA/V², $V_{TP} = -1.5$ V, and $V_{IN} = 1$ V. Calculate the small-signal no-load voltage gain $A_{v0} = v_o/v_s$.

7.56 The parameters of the CMOS amplifier in Fig. P7.56 are $V_{DD} = 5$ V, $V_G = 2$ V, $I_{ref} = 0.5$ mA, $R_G = 500$ kΩ, $C_L \approx \infty$, $V_{MN} = -100$ V, $V_{MP} = 200$ V, $K_n = K_p = 0.5$ mA/V², $V_{TP} = -1.5$ V, and $V_{IN} = 1$ V. Calculate the small-signal no-load voltage gain $A_{v0} = v_o/v_s$. 

FIGURE P7.54

FIGURE P7.55
7.12 DC Level Shifting and MOS Amplifier

7.57 The potential level-shifting circuit shown in Fig. 7.48(a) has \( V_{DD} = -V_{SS} = 15 \) V, \( R_1 = 2 \) k\( \Omega \), and \( R_2 = 4 \) k\( \Omega \). Determine the voltage shift \( V_{sh} \) and the output voltage \( V_o \) at \( v_G = 0 \).

7.58 The potential level-shifting circuit shown in Fig. 7.48(a) operates at a DC source current \( I_o = 1 \) mA, and the DC voltages are \( V_{DD} = -V_{SS} = 15 \) V. Determine the values of \( R_1 \) and \( R_2 \) to produce a voltage shift of 3 V at an output voltage \( v_o = -7 \) V.

7.59 Determine the current source \( I_o \) needed as shown in Fig. 7.47(b) and \( R_1 \) to produce a voltage shift of \( V_{sh} = 4 \) V at an output voltage of \( v_O = -8 \) V. Assume \( v_G = 0 \).

7.60 The parameters of the MOS level-shifted amplifier in Fig. 7.48(a) are \( V_{DD} = 15 \) V, \( -V_{SS} = -15 \) V, \( R_1 = 18 \) k\( \Omega \), \( R_2 = 2 \) k\( \Omega \), \( R_3 = 5 \) k\( \Omega \), \( R_4 = 20 \) k\( \Omega \), and \( R_5 = 6.5 \) k\( \Omega \). The circuit is biased at a DC voltage of \( V_G = 1 \) V. The MOS parameters are \( V_t = 1.5 \) V, \( K_n = 1.25 \text{ mA/V}^2 \), \( K_p = 2.5 \text{ mA/V}^2 \), and \( |V_M| = 1/\lambda = 200 \) V. Assume the bypass capacitance \( C \) is large, tending to infinity. (a) Find the small-signal voltage \( A_{vo} \) and the maximum possible gain. (b) Use SPICE to plot the small-signal output voltage for a sinusoidal input signal of 1 mV at 1 kHz.

7.61 Design an NMOS level-shifting amplifier as shown in Fig. 7.48(a) to produce a voltage gain of \( A_{vo} = 50 \) V/V at a DC input signal of \( v_s = 1 \) mV. Use identical NMOS \( V_M = -200 \) V, \( K_n = 1 \text{ mA/V}^2 \), and \( V_{IN} = 1 \) V. Assume \( V_{DD} = 15 \) V.

7.62 Design a PMOS level-shifting amplifier as shown in Fig. 7.48(a) to produce a voltage gain of \( A_{vo} = 50 \) V/V at a DC input signal of \( v_s = 1 \) mV. Use identical PMOS \( V_{MP} = 200 \) V, \( K_n = 1 \text{ mA/V}^2 \), and \( V_{IN} = -1.5 \) V. Assume \( V_{DD} = 15 \) V.
7.13 Frequency Response of MOSFET Amplifiers

7.63 A depletion NMOS is biased at \( I_D = 4 \, mA \), \( V_{DS} = 4 \, V \), and \( V_{GS} = -2 \, V \). The parameters of the JFET are \( C_{gs0} = 3.49 \, pF \), \( C_{gd0} = 5.85 \, pF \), \( g_m = 4.98 \, mA/V \), \( r_o = 47 \, k\Omega \), and \( V_{bi} = 0.8 \, V \).

a. Calculate the capacitances of the MOSFET model in Fig. 8.48(d).
b. Find the unity-gain bandwidth \( \beta_T \).
c. Use PSpice/SPICE to generate the model parameters and plot the frequency characteristic (\( \beta_T \) versus frequency).

7.64 Repeat Prob. 7.63 for \( I_D = 2 \, mA \), \( V_{DS} = 4 \, V \), and \( V_{GS} = -2.5 \, V \).

7.65 An NMOS transistor of type 2N4351 is biased at \( I_D = 6 \, mA \), \( V_{DS} = 5 \, V \), \( V_{GS} = 8.6 \, V \), \( V_{SB} = 1 \, V \), and \( V_{DB} = 4 \, V \). The NMOS parameters are \( K_p = 125 \, \mu A/V^2 \), \( g_m = 4.98 \, mA/V \), \( C_{gd} = 1.5 \, pF \), \( C_{sb0} = 0.5 \, pF \), \( C_{gs0} = 3.7 \, pF \) at \( V_{DB} = 10 \, V \), and \( V_{bi} = 0.6 \, V \).

a. Calculate the capacitances of the MOSFET model in Fig. 8.48(d).
b. Find the unity-gain bandwidth \( \beta_T \).

7.66 Design a common-source depletion MOSFET amplifier as shown in Fig. P7.66 to give a midband gain of \( 20 \leq |A_{mid}| \leq 25 \), \( Z_{in(mid)} \geq 50 \, k\Omega \), a low 3-dB frequency of \( f_L \leq 10 \, kHz \), and a high 3-dB frequency of \( f_H = 100 \, kHz \).

**FIGURE P7.66**

7.67 Design a common-source NMOS amplifier as shown in Fig. P7.67 to give a passband gain of \( 20 \leq |A_{PB}| \leq 30 \), \( Z_{in(mid)} \geq 100 \, k\Omega \), a low 3-dB frequency of \( f_L \leq 10 \, kHz \), and a high 3-dB frequency of \( f_H = 200 \, kHz \).

**FIGURE P7.67**
7.68 Design a common-source NMOS amplifier as shown in Fig. P7.68 to give a midband gain of $30 \leq |A_{PB}| \leq 35$, $Z_{\text{in(mid)}} \approx 100$ kΩ, a low 3-dB frequency of $f_L \approx 20$ kHz, and a high 3-dB frequency of $f_H = 100$ kHz.

**FIGURE P7.68**

7.69 Design a common-drain depletion NMOS amplifier as shown in Fig. P7.69 to give $Z_{\text{in(mid)}} = 1$ MΩ, a low 3-dB frequency of $f_L \leq 1$ kHz, and a high 3-dB frequency of $f_H = 50$ kHz.

**FIGURE P7.69**

7.70 Design a common-drain depletion NMOS amplifier as shown in Fig. P7.70 to give $Z_{\text{in(mid)}} \geq 100$ MΩ, a low 3-dB frequency of $f_L \leq 1$ kHz, and a high 3-dB frequency of $f_H = 50$ kHz.

**FIGURE P7.70**

7.71 A two-stage amplifier is shown in Fig. P7.71. The parameters are $R_s = 1$ kΩ, $R_{11} = 500$ kΩ, $R_{21} = 500$ kΩ, $R_{D1} = 10$ kΩ, $R_{12} = 500$ kΩ, $R_{22} = 500$ kΩ, $R_D = 15$ kΩ, $R_L = 10$ kΩ, $g_{m1} = 20$ mA/V, $g_{m2} = 50$ mA/V.
7.14 Design of MOSFET Amplifiers

**7.72** Design a common-source depletion NMOS amplifier as shown in Fig. 7.65(a). The requirements are \( I_D = 10 \, mA, \, A_V = -5, \) and \( R_I = 50 \, k\Omega \). The FET parameters are \( V_p = -4 \, V, \, I_{DSS} = 20 \, mA, \) and \( V_M = -200 \, V \). Assume \( R_s = 500 \, \Omega, \, V_{DD} = 20 \, V, \) and \( R_L = 50 \, k\Omega \).

**7.73** Design a common-source NMOS amplifier as shown in Fig. 7.65(b). The requirements are \( A_V = -5, \) \( R_I = 50 \, k\Omega, \) and \( I_D = 10 \, mA \). The MOSFET parameters are \( V_p = 2 \, V, \, K_n = 40 \, mA/V^2, \) and \( V_M = -200 \, V \). Assume \( R_s = 0, \, V_{DD} = 20 \, V, \) and \( R_L = 50 \, k\Omega \).

**7.74** Design a common-source depletion NMOS amplifier as shown in Fig. 7.65(a). The requirements are \( I_D = 20 \, mA, \) \( A_V = -4, \) and \( R_I = 50 \, k\Omega \). The MOSFET parameters are \( V_p = -5 \, V, \, I_{DSS} = 40 \, mA, \) and \( V_M = -100 \, V \). Assume \( R_s = 500 \, \Omega, \, V_{DD} = 20 \, V, \) and \( R_L = 5 \, k\Omega \).

**7.75** Design a common-source NMOS amplifier as shown in Fig. 7.65(b). The requirements are \( A_V = -15, \) \( R_I = 10 \, M\Omega, \) and \( I_D = 10 \, mA \). The MOSFET parameters are \( V_p = -4 \, V, \, K_n = 50 \, mA/V^2, \) and \( V_M = -100 \, V \). Assume \( R_s = 1 \, k\Omega, \, V_{DD} = 20 \, V, \) and \( R_L = 5 \, k\Omega \).

**7.76** Repeat Prob. 7.75 for the configuration shown in Fig. 7.65(c).

**7.77** Design a source follower as shown in Fig. 7.42(a). The requirements are \( R_I = 50 \, k\Omega \) and \( I_D = 10 \, mA \). The MOSFET parameters are \( V_p = -3 \, V, \, I_{DSS} = 40 \, mA, \) and \( V_M = -200 \, V. \) Assume \( R_s = 500 \, \Omega, \, V_{DD} = 20 \, V, \) and \( R_L = 10 \, k\Omega \).

**7.78** Design a source follower as shown in Fig. 7.42(a) to yield \( R_I = 50 \, k\Omega \) and \( I_D = 10 \, mA \). The MOSFET parameters are \( V_p = -4 \, V, \, I_{DSS} = 20 \, mA, \) and \( V_M = -200 \, V. \) Assume \( R_s = 0, \, V_{DD} = 20 \, V, \) and \( R_L = 10 \, k\Omega \).

**7.79** Design a cascoded amplifier as shown in Fig. P7.79 to give a voltage gain of \( A_V = v_L/v_s = -5. \) The MOSFET parameters are \( V_p = -4 \, V, \, I_{DSS} = 10 \, mA, \) and \( V_M = -200 \, V. \) Assume \( V_{DD} = 15 \, V \) and \( R_s = 250 \, \Omega \).

---

\[ C_1 = 1 \, \mu F, \, C_2 = 1 \, \mu F, \, C_3 = 10 \, \mu F, \, C_{gs1} = C_{gs2} = 2 \, pF, \] and \( C_{gs1} = C_{gs2} = 5 \, pF. \) Calculate the low 3-dB frequency \( f_L \) and the high cutoff frequency \( f_H. \)
7.80 Design a CS amplifier with a MOS current source by determining the value of $R_{ref}$ to obtain a biasing current of $I_D = 0.5$ mA and the small-signal voltage gain of the CS amplifier in Fig. 7.25(c). The DC supply voltage is $V_{DD} = 15$ V. The MOS parameters are $V_{IN} = -V_{IP} = 1$ V, $K_n = 1.25$ mA/V$^2$, $K_p = 2.5$ mA/V$^2$, and $|V_M| = 1/\lambda = 200$ V. Use SPICE to plot the small-signal output voltage for a sinusoidal input signal of 1 mV at 1 kHz.

7.81 Design a common-drain amplifier with a MOS current source as shown in Fig. 7.29(b) to bias the source follower in Fig. 7.39(a) at a drain current of $I_D = 3.25$ mA. The DC supply voltage is $V_{DD} = 15$ V, and $R_I = 10$ k$\Omega$. The MOS parameters are $V_I = 1.5$ V, $K_n = 1.25$ mA/V$^2$, $K_p = 2.5$ mA/V$^2$, and $|V_M| = 1/\lambda = 200$ V. Find the small-signal voltage $A_{vo}$ and the output resistance $R_o$. Use SPICE to plot the small-signal output voltage for a sinusoidal input signal of 1 mV at 1 kHz.

7.82 Design a multistage NMOS amplifier to meet the following specifications: voltage gain $|A| = v_L/v_s = 600 \pm 5\%$ (with load), input resistance $R_i = v_i/i_s \geq 25$ k$\Omega$, output resistance $R_o \leq 300$ $\Omega$, load resistance $R_L = 25$ k$\Omega$, source resistance $R_s = 1$ k$\Omega$, DC supply $V_{DD} = 15$ V, input signal $v_s = 1$ mV to 5 mV (peak sinusoidal), 1 kHz. Use identical NMOS $V_M = -200$ V, $K_n = 1$ mA/V$^2$, and $V_{IN} = 1$ V. Assume $V_{DD} = 15$ V. (Hints: The first CS stage should meet the input resistance requirement; the third CD stage should meet the output resistance requirement; and the middle CS stage should attain the remaining gain requirement. Set the biasing drain current at $I_D = I_{D(max)}/3$ of the NMOS.)

7.83 Design a multistage PMOS amplifier to meet the following specifications: voltage gain $|A| = v_L/v_s = 600 \pm 5\%$ (with load), input resistance $R_i = v_i/i_s \geq 25$ k$\Omega$, output resistance $R_o \leq 300$ $\Omega$, load resistance $R_L = 25$ k$\Omega$, source resistance $R_s = 1$ k$\Omega$, DC supply $V_{DD} = 15$ V, input signal $v_s = 1$ mV to 5 mV (peak sinusoidal), 1 kHz, type 2N2222. (Hints: The first CS stage should meet the input resistance requirement; the third CD stage should meet the output resistance requirement; and the middle CS stage should attain the remaining gain requirement. Set the drain biasing current at $I_D = I_{D(max)}/3$ of the PMOS.)

7.84 Design a multistage depletion NMOS amplifier to meet the following specifications: voltage gain $|A| = v_L/v_s = 600 \pm 5\%$ (with load), input resistance, $R_i = v_i/i_s \geq 25$ k$\Omega$, output resistance $R_o \leq 300$ $\Omega$, load resistance $R_L = 25$ k$\Omega$, source resistance $R_s = 1$ k$\Omega$, DC supply $V_{DD} = 15$ V, input signal $v_s = 1$ mV to 5 mV (peak sinusoidal), 1 kHz. Use identical depletion NMOS $V_M = -200$ V, $K_n = 1.5$ mA/V$^2$, and $V_P = -3.5$ V. Assume $V_{DD} = 15$ V. (Hints: The first CS stage should meet the input resistance requirement; the third CD stage should meet the output resistance requirement; and the middle CS stage should attain the remaining gain requirement. Set the drain biasing current at $I_D = I_{D(max)}/3$ of the NMOS.)
Learning Outcomes

After completing this chapter, students should be able to do the following:

- Describe the operation of bipolar junction transistors (BJTs).
- List the types of bipolar transistors and their characteristics.
- List the circuit configurations of transistor amplifiers and their relative advantages and disadvantages.
- Analyze and design bipolar transistor biasing circuits.
- Determine the small-signal model parameters of bipolar transistors.
- Analyze and design bipolar transistor amplifiers.
- Design a BJT amplifier to meet certain specifications.
- Determine the low and high cutoff frequencies of bipolar transistor amplifiers.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$v_o(t)$, $v_O(t)$</td>
<td>Small-signal AC and instantaneous DC output voltages</td>
</tr>
<tr>
<td>$v_b$, $v_B$, $V_B$</td>
<td>Small-signal, instantaneous DC, and quiescent DC base voltages</td>
</tr>
</tbody>
</table>
8.1 Introduction

In Chapter 2, we looked at an amplifier’s characteristics from an input-output perspective and found the specifications of amplifiers that satisfied certain input and output requirements. Internally, amplifiers use one or more bipolar transistors as amplifying devices, and these transistors are biased from a single DC supply to operate properly at a desired $Q$-point. Using bipolar transistors, we can build amplifiers that give a voltage (or current) gain, a high input impedance, or a high (or low) output impedance. The terminal behavior of an amplifier depends on the types of devices used within the amplifier.

Bipolar transistors are active devices with highly nonlinear characteristics. Thus, to analyze and design a bipolar transistor circuit, we need models of transistors. Creating accurate models requires detailed knowledge of the physical operation of transistors and their parameters as well as a powerful analytical technique. A circuit can be analyzed easily using simple models, but there is generally a trade-off between accuracy and complexity. A simple model, however, is always useful to obtain the approximate values of circuit elements for use in a design exercise and the approximate performance of the elements for circuit evaluation. The details of bipolar transistor operation, characteristics, biasing, and modeling are outside the scope of this text [1–3]. In this chapter, we consider the operation and external characteristics of bipolar junction transistors using simple linear models.

8.2 Bipolar Junction Transistors

The bipolar junction transistor (BJT), developed in the 1960s, was the first device for amplification of signals. BJTs continue to play a key role in microelectronics, especially in analog electronics. Integrated circuit–fabrication techniques have led to small, high-speed devices. A BJT consists of a silicon (or germanium) crystal to which impurities have been added such that a layer of $p$-type (or $n$-type) silicon is sandwiched between two layers of $n$-type (or $p$-type) silicon. Therefore, there are two types of transistors: $npn$ and $pnp$.
The basic structures of npn and pnp transistors are shown in Fig. 8.1[(a) and (b)]. A BJT may be viewed as two pn junctions connected back to back. It is called bipolar because two polarity carriers (holes and electrons) carry charge in the device. A BJT is often referred to simply as a transistor. It has three terminals, known as the emitter (E), the base (B), and the collector (C). The symbols are shown in Fig. 8.1[(c) and (d)]. The direction of the arrowhead by the emitter determines whether the transistor is an npn or a pnp transistor, as illustrated in Fig. 8.1[(c) and (d)].

The block diagrams of Fig. 8.1 are highly simplified but useful to understand the concepts of basic transistor theory. The internal structure of actual bipolar transistors is more complex due to the fact that terminal connections are made at the surface, heavily doped n+ buried layers must be included to minimize semiconductor resistances, and collector terminals of individual transistors must be isolated from each other to fabricate more than one bipolar transistor on a single piece of semiconductor material. Figure 8.2 shows a cross section of a conventional npn bipolar transistor fabricated in an integrated circuit configuration. In the epitaxial growth, a thin, single-crystal layer of material is grown on the surface of a single-crystal substrate, which acts as the seed, and the process takes place far below the melting temperature. The emitter and the collector regions are not symmetrical. The impurity-doping concentrations in the emitter and collector are different, and the geometry of these regions can also differ significantly.

FIGURE 8.1 Basic structures and symbols of BJTs

FIGURE 8.2 Cross section of a conventional integrated circuit npn bipolar transistor
The voltages between two terminals and the actual direction of the current-flow of transistor currents are shown in Fig. 8.1[(c) and (d)]. The emitter current \( I_E \) is the sum of the base current \( I_B \) and the collector current \( I_C \) such that \( I_E = I_B + I_C \). However, according to the Institute of Electrical and Electronic Engineers (IEEE) standard, the sum of the currents must be zero; that is, \( I_E + I_B + I_C = 0 \) or \( -I_E = I_B + I_C \). We will use the notation of actual current direction rather than the IEEE notation so that all currents have positive values. \( I_C, I_B, \) and \( I_E \) are positive for npn-type transistors, and they are negative for pnp-type transistors.

**KEY POINTS OF SECTION 8.2**

- The emitter and the collector regions are not symmetrical because the impurity-doping concentrations in the emitter and collector are different and the geometry of these regions can also differ significantly.
- We use the notation of actual current direction rather than the IEEE notation so that all currents have positive values. That is, \( I_C, I_B, \) and \( I_E \) are positive for npn-type transistors, and they are negative for pnp-type transistors.

**8.3 Principles of BJT Operation**

There are two \( pn \) junctions, which must be biased with external voltages to cause any current flow through any of the junctions, as discussed in Secs. 4.4 and 4.5. Recall from our discussion on semiconductor diodes that the current flows through a forward-biased \( pn \) junction due to the majority carriers and the current flows through a reverse-biased \( pn \) junction due to the minority carriers. The npn and pnp transistors are complementary devices. The principles of operation using the npn transistor are explained next, but the same basic principles and equations also apply to the pnp device. An npn transistor as shown in Fig. 8.3(a) is connected to two DC-voltage supplies \( v_{BE} \) and \( v_{CB} \) in order to cause a current flow. These are known as the biasing voltages. The transistor can operate in any of the four modes as shown in Fig. 8.3(b), depending on the biasing conditions: saturation, normal active, cutoff, and inverted. The potential distribution of the base–emitter (B-E) and the collector–base (C-B) junctions with zero-biasing conditions of \( v_{BE} = 0 \) and \( v_{CB} = 0 \) is shown in Fig. 8.3(c) where \( V_{b1} = V_{bi(BE)} \) and \( V_{b2} = V_{bi(CB)} \) are the built-in potentials of the B-E and C-B junctions, respectively. With zero-biasing conditions \( v_{BE} = 0 \) and \( v_{CB} = 0 \), there will be no potentials to overcome the potential barriers, and there will thus be no current flow through the transistors.

**8.3.1 Forward Mode of Operation**

The B-E \( pn \) junction is forward biased, and the base–collector (B-C) \( pn \) junction is reverse biased in the normal, active bias configuration as shown in Fig. 8.4(a). This configuration is called the forward-active operating mode. Using the \( pn \) junction theory developed in Sec. 6.5, the description of the device operation is as follows:
The B-E junction is forward biased so electrons from the emitter will diffuse into the base region as shown in Fig. 8.4(a). The flow of electrons in the emitter is one major component of the emitter current.

Since the number of injected electrons involved is very much higher, an excess of electrons will be in the base region. The concentration of these minority carrier electrons is a function of the B-E voltage.

Since by design, the impurity concentration in the base is very low, the number of holes in the base is very much smaller than that of electrons in the emitter, and the width of the base region is also made very small.

(continued)
**Cause**

The B-C junction is reverse biased, which causes a large gradient in the electron concentration in the base, so the minority carrier electrons diffuse across the base region. This is shown in Fig. 8.4(b).

There are some recombinations of minority carrier electrons with majority carrier holes in the neutral base region.

The reverse-biased B-C junction current also exists.

**Effects**

Therefore, most of the electrons injected into the base region are swept across the very thin base region by the large positive C-B potential $v_{CB}$ and are collected by the collector. The number of electrons in the collector is a function of the number of electrons injected into the base.

The lost majority carrier holes in the base must be replaced. This requires a second component of the base current as shown in Fig. 8.4(a).

It causes a small reverse-biased current from the base to the collector $I_{CBO}$ due to the minority carrier electrons in the base and holes in the collector.

---

**FIGURE 8.4** Biasing and current flow for forward-mode conditions

---

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
**Collector Current**

The directions for the different components of the electron and hole currents are shown in Fig. 8.5. Due to the transistor action, the current at the collector terminal $i_C$ is a function of the voltages $v_{BE}$ and $v_{CB}$ across the other two terminals. The total collector current, which is controlled by the B-E voltage, is the electron diffusion current $I_{E,n}$ minus the base electron recombination current $I_{B,n}$ and can be described by an exponential function from Eq. (6.69) as given by

$$i_C = I_{C,n} = I_{E,n} - I_{B,n} = I_S e^{v_{BE}/V_T} \tag{8.1}$$

where $I_S$ is the saturation current, whose value ranges from $10^{-12}$ A to $10^{-16}$ A, depending on the collector saturation current density and the doping profiles and levels. $V_T$ is the thermal voltage and equals $kT/q$, which is 25.8 mV at room temperature.

**Emitter Current**

The emitter current, as shown in Fig. 8.5, is due to the flow of electrons injected from the emitter into the base. This current, then, is ideally equal to the collector current given by Eq. (8.1). Since the B-E junction is forward biased, majority carrier holes in the base are injected across the B-E junction into the emitter. These injected holes produce a $pn$ junction current $I_{E,p}$, also as indicated in Fig. 8.5. This current is only a B-E junction current, so this component of emitter current is not part of the collector current. The total emitter current is the sum of the electron diffusion current $I_{E,n}$ and the hole diffusion current. Note that there will be a B-E depletion layer recombination current $I_{B,n}$ which is negligible. The total emitter current, which is also controlled by the base–emitter voltage, can be described by

$$i_E = I_{E,n} + I_{E,p} + I_{B,n} = I_{E,n} + I_{E,p} = I_{SE} e^{v_{BE}/V_T} \tag{8.2}$$

where $I_{SE}$ is the saturation current that depends on the emitter saturation current density and is related to the doping profiles and levels.

**FIGURE 8.5** Directions of electron and hole currents
Base Current

The base current is the sum of the hole diffusion current $I_{E,p}$ and the base recombination current $I_{B,n}$:

$$i_B = i_{B1} + i_{B2} = I_{E,p} + I_{B,n} = I_{SB} \ e^{v_{BE}/V_T} \tag{8.3}$$

where $I_{SB}$ is the base saturation current.

Forward-Current Ratio

The forward-current ratio (or the transport factor), $\alpha_F$, is defined as the ratio of the collector to the emitter current. Since all current components are functions of $\exp(v_{BE}/V_T)$, the ratio of collector current to emitter current is a constant. We can write

$$\frac{i_C}{i_E} = \alpha_F \tag{8.4}$$

where $\alpha_F$ is the common-base forward-current ratio, $\alpha_F < 1$, but it should be as close to unity as possible. The collector current consists of two terms: (1) the dominant term being a fraction of the emitter current $i_E$, which is written as $\alpha_F i_E$, and (2) the second term being the reverse-biased saturation current $I_{CBO}$ of the C-B junction diode. That is,

$$i_C = \alpha_F i_E + I_{CBO} \tag{8.5}$$

where $I_{CBO}$ is the reverse saturation current from the collector to the base.

Forward-Current Gain

The forward-current gain $\beta_F$ is defined as the ratio of the collector to the base current. Since all current components are functions of $\exp(v_{BE}/V_T)$, the ratio of collector current to the base current is also a constant. Since the base current equals the difference between the emitter and collector current, we can write the current gain $\beta_F$ terms of the current ratio $\alpha_F$ as given by

$$\beta_F = \frac{i_C}{i_B} = \frac{\alpha_F}{1 - \alpha_F} \tag{8.6}$$

8.3.2 Cutoff, Saturation, and Inverse-Active Modes of Operation

In the cutoff mode, the B-E junction is either reverse biased, or zero biased, and the B-C junction is also reverse biased. That is, $V_{BE}$ has negative voltage or zero, and $V_{CB}$ has a positive voltage. For reverse-biased junctions, the minority carrier concentrations are ideally zero at each depletion edge. The potential barrier heights of both the B-E and B-C junctions are increased, so there is essentially no charge flow.

In the saturation mode, both junctions are forward biased. The B-E potential barrier is smaller than the potential barrier of the B-C junction. There is a gradient in the minority carrier concentration in the base to induce the collector current. Since both junctions are forward biased, the minority carrier concentrations are greater than the thermal equilibrium values at the depletion region edges. There will be a net flow of electrons from the emitter to the collector.
In the inverse-active mode, the B-E junction is reverse biased, and the B-C junction is forward biased. It is a mirror image of the forward-active mode. The potential barrier height of the B-E junction will increase while the potential barrier height of the B-C junction will decrease. Electrons from the collector will diffuse across the B-C junction into the base and then diffuse into the emitter. The bipolar transistor is not a symmetrical device and the characteristics will therefore be different from those of the active-mode operation. The B-C area is normally much larger than the B-E area, and as a result, not all of the injected electrons will be collected by the emitter. The relative doping concentrations in the base and collector are also different compared with those of the base and emitter. Therefore, we expect a significantly different characteristic between the forward-active and inverse-active modes of operation. The transistor is not normally operated in this mode.

### 8.3.3 Base Narrowing

We have assumed so far that the effective base width is essentially independent of the biasing voltages $V_{BE}$ and $V_{CB}$ of the emitter and collector junctions. The collector voltage affects the width of the space charge or depletion regions as shown in Fig. 8.6(a) for an npn transistor. Since the base region is usually lightly doped, the depletion region at the reverse-biased collector junction can extend significantly into the base region. As the collector voltage is increased, the space charge layer can take up more of the metallurgical base width $w_B$, and the effective base width $w'_B$ is decreased. This effect is called *base narrowing*, or *base-width modulation*, and is known as the *Early effect* after J. M. Early, who first interpreted it.

![FIGURE 8.6 Effects of base junction narrowing](image)
The decrease in the effective base width causes the collector current \( i_C \) to increase as well as the current gain \( \beta_F \). As a result, the collector current \( i_C \) increases with the collector bias voltage \( V_{CE} \). Figure 8.6(b) shows the variations of the collector current \( i_C \) against the collector–emitter voltage \( V_{CE} \) for various values of the B–E voltage \( V_{BE} \). The collector current increases with an increased collector–emitter (C–E) voltage and an increased B–E voltage (or increased base current). The Early effect produces a nonzero slope of the \( i_C \) versus \( V_{CE} \) characteristics and gives a finite output conductance. For an ideal characteristic with collector current independent of the collector voltage \( V_{CE} \), the slope of the line will be zero; thus, the output conductance will be zero. The slope introduced by the Early effect is almost linear with characteristics. If the collector current characteristics are extrapolated to zero collector current, the curves intercept the voltage axis at a point known as the Early voltage. The Early voltage is positive for an npn transistor and negative for a pnp transistor. The typical values of Early voltage are in the range of 100 V to 300 V. From Fig 8.6(b), we can write the output conductance as

\[
 g_o = \frac{1}{r_o} = \frac{\frac{dI_C}{dV_{CE}}}{\frac{I_C}{V_{CE} + V_A}} \quad \text{(8.7)}
\]

where \( r_o \) is the output resistance of the transistor. If we include the finite slope of the \( i_C \) versus \( V_{CE} \) characteristics due to the Early effect, the collector current in Eq. (8.1) can be modified to

\[
 i_C = I_{SC} e^{V_{BE}/V_T} \left( 1 + \frac{V_{CE}}{V_A} \right) \quad \text{(8.8)}
\]

If \( w_E \), \( w_B \), and \( w_C \) are the metallurgical widths of the emitter, base, and collector regions, respectively, we can calculate their corresponding effective widths \( w_E', w_B', \) and \( w_C' \) as follows:

\[
 w_E' = w_E - x_{n(BE)} \quad \text{(8.9)}
\]

\[
 w_B' = w_B - x_{p(BE)} - x_{p(BC)} \quad \text{(8.10)}
\]

\[
 w_C' = w_C - x_{n(BC)} \quad \text{(8.11)}
\]

Applying Eq. (6.54), the space charge width extending to the base region due to \( V_{BE} \) is given by

\[
 x_{n(BE)} = \sqrt{\frac{2\varepsilon_s(V_{bi(BE)} - V_{BE})}{q} \frac{N_E}{N_B} \left( \frac{1}{N_B + N_E} \right)} \quad \text{(8.12)}
\]

Applying Eq. (6.55), the space charge width extending to the emitter region due to \( V_{BE} \) is given by

\[
 x_{p(EB)} = \sqrt{\frac{2\varepsilon_s(V_{bi(BE)} - V_{BE})}{q} \frac{N_E}{N_B} \left( \frac{1}{N_B + N_E} \right)} \quad \text{(8.13)}
\]

Applying Eq. (6.45), the space charge width extending to the collector region due to \( V_{CB} \) is given by

\[
 x_{n(BC)} = \sqrt{\frac{2\varepsilon_s(V_{bi(BC)} + V_{CB})}{q} \frac{N_C}{N_B} \left( \frac{1}{N_B + N_C} \right)} \quad \text{(8.14)}
\]

Applying Eq. (6.46), the space charge width extending to the base region due to \( V_{CB} \) is given by

\[
 x_{p(CB)} = \sqrt{\frac{2\varepsilon_s(V_{bi(BC)} + V_{CB})}{q} \frac{N_B}{N_C} \left( \frac{1}{N_B + N_C} \right)} \quad \text{(8.15)}
\]

Applying Eq. (6.23), the built-in potential of the B–E junction is given by

\[
 V_{bi(BE)} = V_T \ln \left( \frac{N_B N_E}{n_i^2} \right) \quad \text{(8.16)}
\]
Applying Eq. (6.23), the built-in potential of the B-C junction is given by

$$V_{bi(BC)} = V_T \ln \left( \frac{N_B N_C}{n_i^2} \right) \quad (8.17)$$

$N_E$, $N_B$, and $N_C$ are the emitter, base, and collector doping, respectively, in negative cubic centimeters.

The total depletion, or space charge width, $w_{d(BC)}$ of the B-C junction is the sum of two depletion components given by

$$w_{d(BC)} = x_{p(CB)} + x_{n(BC)} \quad (8.18)$$

Using Eqs. (8.14) and (8.15), the width of the collector junction depletion region can be found from

$$w_{d(BC)} = \sqrt{\frac{2e_s (V_{bi(BC)} + V_{CB}) N_B}{q N_C} \left( \frac{1}{N_B + N_C} \right)} + \sqrt{\frac{2e_s (V_{bi(BC)} + V_{CB}) N_C}{q N_B} \left( \frac{1}{N_B + N_C} \right)} \quad (8.19)$$

Since $V_{CB} \gg V_{bi(BC)}$, Eq. (8.19) can be simplified to

$$w_{d(BC)} = \sqrt{\frac{2e_s}{q} \left( \frac{N_B + N_C}{N_B N_C} \right)} \sqrt{V_{CB}} \quad (8.20)$$

which shows that the depletion width is proportional to $\sqrt{V_{CB}}$. If the reverse-bias voltage on the collector junction is increased far enough, it is possible to decrease the base width $w_B$ to the extent such that the effective base width $w_B^*$ becomes almost nonexistent. This is known as the punch-through condition in which the holes are swept directly from the emitter region to the collector and transistor action is lost. Punch-through is a breakdown effect that is generally avoided in circuit design.

**EXAMPLE 8.1**

**Finding the depletion region width**  Calculate the width of the B-C depletion region if the C-B voltages are $V_{CB} = 2 \text{ V}, 6 \text{ V}, 12 \text{ V}$. The physical parameters are $N_C = 2 \times 10^{16} \text{ cm}^{-3}, N_B = 5 \times 10^{15} \text{ cm}^{-3}, V_T = 25.8 \text{ mV}, T = 25\degree \text{C}, w_B = 0.7 \text{ m}, \xi_s = 11.7 \times 8.85 \times 10^{-14}, q = 1.6 \times 10^{-19},$ and $n_i = 1.5 \times 10^{10} \text{ cm}^{-3}$.

**SOLUTION**

Substituting the values in Eq. (8.17), we get the built-in potential as

$$V_{bi(BC)} = V_T \ln \left( \frac{N_B N_C}{n_i^2} \right) = 25.8 \times 10^{-3} \text{ V} \ln \left[ \frac{2 \times 10^{16} \text{ cm}^{-3} \times 5 \times 10^{15} \text{ cm}^{-3}}{(1.5 \times 10^{10} \text{ cm}^{-3})^2} \right] = 0.695 \text{ V}$$

Substituting the values in Eq. (8.19), we get the width of the depletion region with $V_{CB} = 2 \text{ V}$ as

$$w_{d(BC)} = \sqrt{\frac{2 \times 11.7 \times 8.85 \times 10^{-14}(0.695 + 2) \text{ V}}{1.6 \times 10^{-19}} \left( \frac{2 \times 10^{16} \text{ cm}^{-3} + 5 \times 10^{15} \text{ cm}^{-3}}{2 \times 10^{16} \times 5 \times 10^{15} \text{ cm}^{-3}} \right)}$$

$$= 9.338 \times 10^{-5} \text{ cm} = 0.9338 \mu\text{m}$$

For $V_{CB} = 6 \text{ V}$, we get $w_{d(BC)} = 1.472 \times 10^{-4} \text{ cm} = 1.472 \mu\text{m}$. For $V_{CB} = 12 \text{ V}$, we get $w_{d(BC)} = 2.027 \times 10^{-4} \text{ cm} = 2.027 \mu\text{m}$. 

8.3.4 Physical Parameters of Saturation Current $I_S$ and Current Gain $\beta_F$

Figure 8.7(a) shows the minority carrier concentrations through the $nnp$ transistor. The potential barrier between the emitter and the base is reduced due to the forward-biased condition discussed in Sec. 6.5, so electrons from the emitter diffuse across the B-E space charge region. The electrons diffuse across the base and are swept into the collector by the electric field in the B-C space charge region. The majority of these electrons reach the collector and create the major component of the collector current. Figure 8.7(b) shows

![Diagram of minority carrier distribution and current flows](image-url)
the flow of the B-E currents. From Eq. (6.62), the concentration of the minority carrier electrons at the edge of the space charge in the $p$-region is given by

$$n_p(x = 0) = n_{po}e^{v_{BE}/V_T}$$  \hspace{1cm} (8.21)

From Eq. (6.63), the concentration of the minority carrier holes at the edge of the space charge in the $n$-region is given by

$$p_n(x = 0) = p_{no}e^{v_{BE}/V_T}$$  \hspace{1cm} (8.22)

where $n_{po} = n_t^2/N_B$ is the thermal-equilibrium concentration of the minority carrier electrons in the $p$-region base and $p_{no} = n_t^2/N_E$ is the thermal-equilibrium concentration of the minority carrier holes in the $n$-region emitter.

**Collector Saturation Current $I_{SC}$**

If we assume linear electron distribution of the electrons in the base, we find the electron diffusion current $I_{E,n}$ (in the direction of $x$), which is the major component of the emitter current, as

$$I_{E,n} = qD_BA_{BE} \frac{dn_{B}(x)}{dx} = qD_BA_{BE} \frac{n_B(0) - 0}{0 - x_B} = -\frac{qD_BA_{BE}}{x_B}n_{BO}e^{v_{BE}/V_T}$$  \hspace{1cm} (8.23)

where $A_{BE}$ is the cross-sectional area of the B-E junction, $D_B$ is the electron diffusivity in the base, $n_{BO}$ is the thermal-equilibrium electron concentration in the base, $V_T$ is the thermal voltage, and $q$ is the magnitude of the electron charge. The negative slope of the minority carrier concentration causes a negative current; that is, the actual current flows from right to left in the negative direction of $x$. By neglecting the base-electron recombination current $I_{B,n}$ in Eq. (8.1) and equating Eq. (8.23) with Eq. (8.1), we can write the collector saturation current, known simply as the saturation current, as

$$I_S = I_{SC} = \frac{qD_BA_{BE}}{x_B}n_{po} = \frac{qD_BA_{BE}}{x_B}\left(\frac{n_t^2}{N_B}\right)$$  \hspace{1cm} (8.24)

where $n_t$ is the intrinsic carrier density and $N_B$ is the doping concentration in the base. It is important to note that the saturation current $I_S$ is inversely proportional to the base width $x_B$ and directly proportional to the area $A_{BE}$ of the emitter–base (E-B) junction. Because $I_S$ is proportional to $n_t^2$, it approximately doubles for every 5°C rise in temperature. Since $I_S$ is a direct function of the emitter area, transistors having different emitter areas will carry different emitter currents in relation to the emitter sizes for the same amount of applied $v_{BE}$. For example, let us consider two transistors that are identical but one of them having the E-B junction area, say, twice that of the other. The transistor with the larger junction area will have the saturation current twice that of the smaller one. Therefore, for the same value of $v_{BE}$, the larger device will have a collector current twice that of the smaller device. This concept is known as *emitter scaling*, which is frequently employed in integrated circuit design.

**Base Saturation Current $I_{SB}$**

The base current will have two components: (1) the hole diffusion current $I_{E,p} = i_B1$ from the base to the emitter and (2) the base recombination current $I_{B,n} = i_B2$ in order to replace the holes lost from the base
through the recombination. If we neglect the base recombination current, the base recombination current $I_{B,n}$ in Eq. (8.3), the base current approximates to the hole diffusion current $I_{E,p}$ as given by

$$i_{B1} = I_{E,p} = \frac{qD_EA_{BE}}{L_E} \left( \frac{n_t^2}{N_E} \right) e^{\nu_{BE}/V_T} \tag{8.25}$$

where $D_E$ is the hole diffusivity constant in the emitter, $L_E$ is the hole diffusion length in the emitter, and $N_E$ is the doping concentration of the emitter.

The emitter to the base recombination current $I_{B,n}$ is due to holes that have to be supplied by the external base circuit, to replace the holes lost from the base through the recombination. If we define $\tau_b$ as the average time for a minority electron in the base to recombine with a majority hole in the base and $Q_n$ is the minority carrier charge stored in the base that recombines with holes, we find the base current $I_{B,n}$ to replenish the holes from the external circuit as given by

$$i_{B2} = I_{B,n} = \frac{Q_n}{\tau_b} \tag{8.26}$$

where $\tau_b$ is also known as the minority carrier lifetime. From Fig. 8.7(b), we can find $Q_n$, which is approximately the area of the triangle under the straight-line charge distribution, as given by

$$Q_n = qA_{BE} \times \frac{1}{2} n_B(0) \times x_B \tag{8.27}$$

Substituting for $n_B(0)$ from Eq. (8.21) and $n_{po} = n_t^2/N_B$ into Eq. (8.27), we get

$$Q_n = qA_{BE}x_B \times \frac{1}{2} \left( \frac{n_t^2}{N_B} \right) e^{\nu_{BE}/V_T} \tag{8.28}$$

which, after substituting in Eq. (8.26), gives

$$i_{B2} = \frac{1}{2} \times \frac{qA_{BE}x_B}{\tau_b} \times \left( \frac{n_t^2}{N_B} \right) e^{\nu_{BE}/V_T} \tag{8.29}$$

From Eqs. (8.25) and (8.29), we can find the total base current as given by

$$i_B = i_{B1} + i_{B2} = \left[ \frac{qD_EA_{BE}}{L_E} \left( \frac{n_t^2}{N_E} \right) + \frac{1}{2} \frac{qA_{BE}x_B}{\tau_b} \times \left( \frac{n_t^2}{N_B} \right) \right] e^{\nu_{BE}/V_T} \tag{8.30}$$

which gives the base saturation current $I_{SB}$ as given by

$$I_{SB} = \frac{qD_EA_{BE}}{L_E} \left( \frac{n_t^2}{N_E} \right) + \frac{1}{2} \frac{qA_{BE}x_B}{\tau_b} \times \left( \frac{n_t^2}{N_B} \right)$$

Using $I_S$ from Eq. (8.24), we can find the relation between $I_{SB}$ and $I_S (=I_{SC})$ as

$$I_{SB} = I_S \left( \frac{D_E N_B x_B}{D_B N_E L_E} + \frac{1}{2} \frac{x_B^2}{D_B \tau_b} \right) \tag{8.31}$$

**Current Gain $\beta_F$**

Since $i_C = \beta_F i_B$ and $I_S = \beta_F I_{SB}$, Eq. (8.31) gives the current gain $\beta_F$

$$\beta_F = \left( \frac{D_E N_B x_B}{D_B N_E L_E} + \frac{1}{2} \frac{x_B^2}{D_B \tau_b} \right)^{-1} \tag{8.32}$$
Therefore, the value of $\beta_F$ should be a constant for a particular transistor and depends on the physical parameters of a particular transistor. Its value is highly influenced by two factors: (1) the width of the base region, $x_B$, and (2) the relative dopings of the base region and the emitter region, $N_B/N_E$. To obtain a high value of $\beta_F$, the transistors should be designed by making the base thin ($x_B$ small) and lightly doped and by making the emitter heavily doped ($N_B/N_E$ small).

**KEY POINTS OF SECTION 8.3**

- A BJT can operate in any of the four operating modes depending on the biasing conditions: saturation, normal active, cutoff, and inverted. For an amplification, the B-E junction is forward biased and the C-B junction is reverse biased, while for operation in the saturation region, both B-E and C-B junctions are forward biased.
- The major physical parameters of a BJT are the forward current gain, the forward current ratio, the saturation current, and the Early voltage.
- The collector voltage affects the width of the space charge or depletion regions and the width of the depletion region depends on the C-B voltage.

### 8.4 Input and Output Characteristics

To properly initiate current flow, a transistor must be biased. Figure 8.8 illustrates an example of biasing using two DC supplies, $V_{CC}$ and $V_{BB}$. This arrangement is not used in practice; it is shown only to illustrate the transistor characteristics. A practical biasing circuit uses only one DC supply for transistor biasing; this arrangement is discussed later in this section. $R_C$ serves as a load resistance.

However, the arrangement shown in Fig. 8.8(a) or (b) is useful in the development of the concept of transistor models and signal amplification.

Each of the three terminals of a transistor may be classified as an input terminal, an output terminal, or a common terminal. There are three possible configurations: (1) common emitter (CE), in which the emitter is the common terminal; (2) common collector (CC) or emitter follower, in which the collector is the common terminal; and (3) common base (CB), in which the base is the common terminal.
The CB configuration is not as commonly used as the other two. A transistor can be described by two characteristics: an input characteristic and an output characteristic. The input characteristic is similar to that of a forward-biased diode if the emitter is the common terminal; the input characteristic for \( npn \) and \( pnp \) transistors is shown in Fig. 8.9(a), which can be described mathematically by Eq. (8.33) as follows:

\[
i_B = \left( \frac{I_S}{\beta_F} \right) e^{v_{BE}/V_T}
\]

Applying Kirchhoff’s voltage law (KVL) as the base loop, we write

\[
V_{BB} = R_Bi_B + v_{BE}
\]

which can be solved for the base current \( i_B \) as given by

\[
i_B = \frac{V_{BB} - v_{BE}}{R_B}
\]

Equation (8.35), which describes the base load line for the input characteristic as shown in Fig. 8.9(a), gives \( v_{BE} = 0 \) at \( i_B = V_{BB}/R_B \) and \( v_{BE} = V_{BB} \) at \( i_B = 0 \). The intersection of the base load line with the input characteristic gives the base operating point defined by \( I_B \) and \( V_{BE} \). Equations (8.33) and (8.35) can be solved to find the DC biasing B-E voltage \( V_{BE} \) and also the DC base current \( I_B \) for known values of \( V_{BB} \) and \( R_B \).

A typical output characteristic for a BJT is shown in Fig. 8.9(b). \( v_{CE} \) and \( i_C \) are positive for \( npn \) transistors and negative for \( pnp \) transistors. If the base current \( i_B \) is kept constant, then the collector current \( i_C \) will increase with the C-E voltage \( v_{CE} \) until the collector current saturates—that is, reaches a level at which any increase in \( v_{CE} \) causes no significant change in the collector current. The output characteristic may be divided into three regions: an active region, a saturation region, and a cutoff region. The transistor can be used as a switch in the saturation region because \( v_{CE} \) is low, typically 0.3 V. In both the active and the saturation region, the B-E junction is forward biased and \( v_{BE} \approx 0.7 \). In the active region, \( 0 < v_{BE} < v_{CE} \) and \( v_{CB} = v_{CE} - v_{BE} > 0 \); that is, the B-E junction is forward biased, and the C-B junction is reverse biased. All transistors exhibit a high output impedance (or resistance), described by Eq. (8.7). Operation in the active region can give an amplification of signals with a minimum amount of distortion, because the output characteristic is approximately linear.
A transistor is a current-controlled device. The collector current \( i_C \) is related to the base current \( i_B \) by a forward-current amplification factor \( \beta_F \), which is defined as

\[
\beta_F = \left. \frac{i_C}{i_B} \right|_{v_{CE} = \text{constant}} \tag{8.36}
\]

Once base current \( i_B \) is determined, the collector current \( i_C \) and the emitter current \( i_E \) can be found as follows:

\[
i_C = \beta_F i_B \tag{8.37}
\]

\[
i_E = i_B + i_C = i_B + \beta_F i_B = (1 + \beta_F) i_B \tag{8.38}
\]

Using KVL around the loop formed by \( V_{CC}, R_C \), and the collector-emitter, we can relate the collector current \( i_C \) to \( v_{CE} \) by

\[
V_{CC} = v_{CE} + i_C R_C
\]

which gives the dependence of the collector current on the load resistance \( R_C \) and which can be rearranged to yield the following relation, known as the load-line equation:

\[
i_C = \frac{V_{CC}}{R_C} - \frac{v_{CE}}{R_C} \tag{8.39}
\]

Equation (8.39) gives \( v_{CE} = 0 \) at \( i_C = V_{CC} / R_C \) and \( v_{CE} = V_{CC} \) at \( i_C = 0 \). The intersection of the load line with the output characteristic gives the operating point (or \( Q \)-point), which is defined by three parameters: \( I_B, I_C \), and \( V_{CE} \). Thus, for a given value of \( i_B \), the value of \( i_C \) can be found, and then the load line gives the value of \( v_{CE} \), as shown in Fig. 8.9(b).

### KEY POINTS OF SECTION 8.4

- Each of the three terminals of a transistor may be classified as an input terminal, an output terminal, or a common terminal. There are three possible configurations: (1) common emitter (CE), in which the emitter is the common terminal; (2) common collector (CC) or emitter follower, in which the collector is the common terminal; and (3) common base (CB), in which the base is the common terminal.
- The output characteristic of a BJT can be divided into three regions: (1) a cutoff region in which the transistor is off, (2) an active region in which the transistor exhibits a high output resistance and has a current amplification, and (3) a saturation region in which the transistor offers a low resistance.

### 8.5 BJT Circuit Models

The purpose of an amplifier is to convert an input signal of small amplitude into an output signal of different amplitude while minimizing any distortion introduced by the amplifier. If the input is a sine wave, the output should also be a sine wave. If an AC small-signal \( v_{be} \) is superimposed on the DC biasing voltage \( V_{BE} \) at the base of the transistor, the base current \( I_B \) will change by a small amount \( i_b \), thereby causing...
an amplified change $i_c (\approx i_b \times \text{current gain})$ in the collector current $I_C$. This change will cause the operating point to move up and down along the load line around the $Q$-point. Too large an AC signal, however, will drive the transistor both into the saturation region (to the left of the $v_{CE}$-axis) and into the cutoff region (to the right of the $v_{CE}$-axis). Therefore, the design and analysis of an amplifier involves two signals: a DC signal and an AC signal. The DC analysis finds the $Q$-point defined by $I_C$, $I_B$, and $V_{CE}$. For an AC analysis, a small-signal AC model of a BJT around the $Q$-point is required.

### 8.5.1 Linear DC Model

Linear DC models are used for determining the operating point (or $Q$-point) of a BJT. The B-E junction, which is forward biased in the active region, can be represented by a forward-biased diode, as shown in Fig. 8.10(a). The C-B junction, which is reverse biased, can be represented by an open circuit. The base current varies with the base-to-emitter voltage, as shown in the input characteristic in Fig. 8.9(a). The input characteristic is replaced by a piecewise linear model with resistance $R_{BE}$ in series with a voltage source $V_{BE}$ whose value ranges from 0.5 V to 0.8 V, as shown in Fig. 8.10(b). The finite slope of the output characteristic can be represented by adding an output resistor $r_o$ between the collector and emitter terminals. For most applications, this model can be approximated by Fig. 8.10(c) by assuming $R_{BE} = 0$ and $r_o = \infty$. It is commonly used for obtaining quick results.

### 8.5.2 Small-Signal AC Model

Linear DC models are used for determining the $Q$-point; however, an AC model is used for determining the voltage or power gain when the transistor is operated as an amplifier in the active region. If we apply a small sinusoidal input voltage $v_{be} = V_m \sin \omega t$ while operating in the active region, the base potential will be $v_{BE} = V_{BE} + v_{be}$, and the corresponding base current will be $i_B = I_B + i_b$. The corresponding collector current will be $i_C = I_C + i_c$, as shown in Fig. 8.11(a). The small-signal AC resistance $r_\pi$ seen by $v_{be}$ will be the inverse slope of the $i_B - v_{BE}$ characteristic at the $Q$-point ($I_B$, $V_{BE}$), as shown in Fig. 8.11(b). That is, we can obtain $r_\pi$ by differentiating $i_B$:

$$\frac{1}{r_\pi} = \frac{i_B}{v_{be}} = \left. \frac{di_B}{dv_{BE}} \right|_{\text{at } Q\text{-point}} = \frac{I_B}{V_T} = \frac{I_B}{25.8 \text{ mV}}$$

(8.40)
If the base current $i_B$ swings between $I_B + i_b(\text{peak})$ and $I_B - i_b(\text{peak})$, the collector current $i_C$ will swing between $I_C + i_c(\text{peak})$ and $I_C - i_c(\text{peak})$. The C-E voltage $V_{CE}$ will vary accordingly from $V_{CE} - v_{ce(\text{peak})}$ to $V_{CE} + v_{ce(\text{peak})}$, as illustrated also in Fig. 8.11(b). The small-signal collector current $i_c$ will depend on the small-signal AC current gain $\beta_f$, defined by

$$\beta_f = \frac{i_c}{i_b} = \left. \frac{\Delta i_C}{\Delta i_B} \right|_{Q\text{-point}} \quad (8.41)$$

which may be considered approximately equal to the DC current gain $\beta_F$ for most applications. That is, $\beta_F = \beta_f$. We will make this assumption throughout.

The collector current can be related to the B-E voltage by transconductance $g_m$, defined by

$$g_m = \frac{i_c}{v_{be}} = \left. \frac{\beta_f dI_B}{dv_{BE}} \right|_{Q\text{-point}} = \frac{\beta_f I_B}{V_T} = \frac{I_C}{V_T} = \frac{r_T}{r_{\pi}} \quad (8.42)$$

where the derivative is evaluated at the $Q$-point. The output characteristic in the active region exhibits a finite slope representing an output resistance defined by Eq. (8.7)

$$\frac{1}{r_o} = \left. \frac{i_c}{v_{ce}} \right|_{Q\text{-point}} = \frac{dI_C}{dv_{CE}} = \frac{I_C}{V_A + V_{CE}} = \left. \frac{I_C}{V_A} \right|_{Q\text{-point}} \quad \text{for } V_A >> V_{CE} \quad (8.43)$$

where $V_A$ is a constant called the Early voltage whose value ranges from 100 V to 200 V, depending on the transistor [4]. The value of $r_o$ is large (on the order of 50 kΩ) and can be neglected for most analyses.

Any increase in $V_{CE}$ will increase the width of the collector depletion layer; consequently, the effective base width will be reduced, causing a reduction in $I_B$. The decrease in $I_B$ due to an increase in $V_{CE}$ can be modeled by a C-B resistance $r_\mu$. The value of $r_\mu$ can be approximated by $r_\mu = 10r_o\beta_f$, which is very large compared to $r_{\pi}$ and $r_o$ and is not normally included in the transistor model, especially for hand calculations.

Thus, the small-signal behavior of a transistor can be modeled by an input resistance $r_{\pi}$, a base current–dependent collector current $i_c = \beta f i_b$ along with an output resistance $r_o$, and a C-B resistance $r_\mu$. Since the C-B junction is reverse biased, $r_\mu$ can be neglected by assuming $r_\mu = \infty$. This model, shown in Fig. 8.12(a), can be approximated by Fig. 8.12(b). The transconductance representations are
shown in Fig. 8.12[(c) and (d)]. If Norton’s current source is converted to Thevenin’s voltage source, Fig. 8.12(c) can be represented by Fig. 8.12(e), where
\[ g_{m} r_{o} \]

Note that the units of the model parameters in Fig. 8.12(a) are different.

**NOTE** \( r_{\pi} \) is the small-signal base–emitter resistance \( r_{be} \). It uses the subscript \( \pi \) because it is the input resistance of the model, which looks like the symbol \( \pi \) and is also known as the \( \pi \) model.

### 8.5.3 Small-Signal Hybrid Model

The manufacturers of BJTs usually specify the common-emitter hybrid parameters corresponding to the hybrid model shown in Fig. 8.12(f). The parameters are as follows: (See also Appendix C.)

- \( h_{ie} (= r_{\pi}) \) is the short-circuit input resistance (or simply the input resistance).
- \( h_{fe} (= \beta_{f}) \) is the short-circuit forward-transfer current ratio (or small-signal current gain).
- \( h_{re} \) is the open-circuit reverse-voltage ratio (or voltage-feedback ratio), which takes into account the effect of \( v_{CE} \) on \( i_{B} \). This ratio is very small; its value is typically \( 0.5 \times 10^{-4} \). \( r_{\mu} \) represents the effect of \( h_{re} \).
- \( h_{oe} (= 1/r_{o}) \) is the open-circuit output admittance (or simply the output admittance) of the C-E junction. It is also very small; its value is typically \( 10^{-6} \) \( \Omega \).

Often \( h_{re} \) and \( h_{oe} \) can be omitted from a circuit model without significant loss of accuracy, especially in hand calculations. The subscript \( e \) on the \( h \) parameters indicates that these hybrid parameters are derived for a common-emitter configuration.

### 8.5.4 PSpice/SPICE Model

PSpice/SPICE generates a complex BJT model, provided a number of physical parameters are given. The symbol for a BJT is \( Q \), and it is described by the statement [5]

\[ \text{Q<name> NC NB NE QMOD} \]
where NC, NB, and NE are the collector, base, and emitter nodes, respectively. QMOD is the model name, which can be up to eight characters long. The model statement for an npn transistor has the general form

```
.MODEL QMOD NPN (P1=A1 P2=A2 P3=A3 .......PN=AN)
```

The model statement for a pnp transistor has the general form

```
.MODEL QMOD PNP (P1=A1 P2=A2 P3=A3 .......PN=AN)
```

In these model statements, NPN and PNP are the type symbols for npn and pnp transistors, respectively. P1, P2, . . . , PN and A1, A2, . . . , AN are the parameters and their values, respectively.

As an example, let us derive two parameters, $I_S$ and $\beta_F$, for transistor Q2N2222. Reading from the plot of $v_{BE}$ versus $i_C$ on the data sheet for Q2N2222, we get $v_{BE} = 0.7$ V at $i_C = 20$ mA. Inserting these values into Eq. (8.1) yields

$$20 \text{ mA} = I_S \exp \left( \frac{0.7 \text{ V}}{25.8 \text{ mV}} \right)$$

which gives $I_S = 3.295 \times 10^{-14}$ A. The DC gain $\beta_F$ for $i_C = 150$ mA can vary between 100 and 300. This variation is not defined, however, and can change randomly from one transistor to another of the same type. As a working approximation, the geometric mean value is usually used; that is, $\beta_F = \sqrt{100 \times 300} = 173$. Since the value for Early voltage is not given, let us assume that $V_A = 200$ V. With these values of $I_S$, $\beta_F$, and $V_A$, the transistor Q2N2222 can be specified in PSpice/SPICE by the following statements:

```
Q1 NC NB NE QMOD
.MODEL Q2N2222 NPN (IS=3.295E-14 BF=173 VA=200)
```

**NOTE** The full data sheets for BJTs (e.g., npn-type 2N2222 and pnp-type 2N2907A) can be found at http://www.alldatasheet.com/ or by searching BJT datasheet at http://www.google.com.

### 8.5.5 Small-Signal Analysis

Once the $Q$-point is established and the small-signal parameters are determined, we can find the small-signal parameters of the amplifier in Fig. 8.11(a) in response to a small-signal voltage $v_{be}$. For a small AC signal, the DC supply offers zero impedance; $V_{CC}$ and $V_{BB}$ can be short-circuited; that is, one side of $R_C$ is connected to the ground. The small-signal AC equivalent circuit of the amplifier is shown in Fig. 8.13(a). Replacing the transistor $Q_1$ by its transconductance model of Fig. 8.12(c), the small-signal AC equivalent circuit is shown in Fig. 8.13(b). The following steps are involved in analyzing an amplifier circuit:

1. Analyzing the DC biasing of the transistor circuit
2. Determining the small-signal parameters $g_m$, $r_\pi$, and $r_o$ of the transistor model in Fig. 8.12(c)
3. Determining the AC equivalent circuit of the amplifier
4. Performing the small-signal analysis for finding $R_i$, $A_v$, and $R_o$

From Fig. 8.13(b), the small-signal input resistance can be found from

$$R_i = \frac{v_{be}}{i_b} = r_\pi$$

Thevenin’s equivalent output resistance, looking from the output side for the condition $v_{be} = 0$, and $g_m v_{be}$ behaving as open circuited, can be found by inspection as

$$R_o = r_o \parallel R_C$$  \hspace{1cm} (8.44)
For the sake of simplicity, we often use the small-signal parameters $r_i$, $r_o$, and $g_m$ instead of $r_{i1}$, $r_{o1}$, and $g_{m1}$ for transistor Q1.

The transconductance of the amplifier, which is the same as the transconductance of the transistor, is

$$G_{mo} = \frac{i_o}{v_{be}} = g_m$$  \hspace{1cm} (8.45)

We can write the small-signal output voltage $v_o$ as

$$v_o = -i_o (r_o \parallel R_C) = -g_m(r_o \parallel R_C)v_{be}$$  \hspace{1cm} (8.46)

which gives the small-signal voltage gain as

$$A_{vo} = \frac{v_o}{v_{be}} = -g_m(r_o \parallel R_C) = -\left(\frac{I_C}{V_T}\right)\left(\frac{r_o R_C}{r_o + R_C}\right)$$  \hspace{1cm} (8.47)

Substituting $r_o = V_A/I_C$, Eq. (8.47) becomes

$$A_{vo} = -\left(\frac{I_C}{V_T}\right)\left(\frac{V_A R_C}{V_A + I_C R_C}\right) \approx \frac{I_C R_C}{V_T} \left(\frac{V_A}{V_A + I_C R_C}\right) \quad \text{for } V_A >> R_C I_C$$  \hspace{1cm} (8.48)

Therefore, for a large voltage gain, the $I_C R_C$ product must also be made large. This requires both a large DC supply voltage $V_{CC}$ and a large value of resistance $R_C$. 

**FIGURE 8.13** Small-signal AC equivalent circuits of the amplifier in Fig. 8.11(a)

**NOTE** For the sake of simplicity, we often use the small-signal parameters $r_i$, $r_o$, and $g_m$ instead of $r_{i1}$, $r_{o1}$, and $g_{m1}$ for transistor Q1.
EXAMPLE 8.2

Finding the small-signal parameters of an amplifier  The amplifiers in Fig. 8.11(a) are \( V_{BE} = 0.68 \) V, \( V_{CC} = 15 \) V, and \( R_C = 1 \) kΩ. The transistor parameters are \( \beta_F = 100 \), \( V_A = 200 \) V, \( V_T = 25.8 \) mV, and \( I_S = 3.3 \times 10^{-14} \) A.

(a) Find the DC-biasing point \( I_B, I_C \), and \( V_{CE} \).
(b) Find the small-signal transistor model parameters \( r_\pi, r_o \), and \( g_m \).
(c) Find the small-signal amplifier parameters \( R_i, R_o \), and \( A_{vo} \).

SOLUTION

(a) From Eq. (8.1), \( I_C = I_S \exp(v_{BE}/V_T) = 3.3 \times 10^{-14} \times \exp[0.682/(25.8 \times 10^{-3})] = 9.97 \) mA, \( I_B = I_C/\beta_F = 9.97 \) mA/100 = 99.7 \( \mu A \), and \( V_{CE} = V_{CC} - R_CI_C = 15 \) V - 9.97 mA \( \times 1 \) kΩ = 5.03 V.

(b) From Eq. (8.40), \( r_\pi = V_T/I_B = 25.8 \) mV/99.7 \( \mu A = 258.8 \) Ω.

From Eq. (8.42), \( g_m = I_C/V_T = 9.97 \) mA/25.8 mV = 386 mA/V.

From Eq. (8.43), \( r_o = V_A/I_C = 200 \) V/9.97 mA = 20.06 kΩ.

From Eq. (8.7), \( r_o = (V_{CE} + V_A)/I_C = (5.03 + 200) \) V/9.97 mA = 20.56 kΩ.

(c) \( R_1 = r_\pi = 258.8 \) Ω, \( R_o = r_o \parallel R_C = 952.5 \) Ω, \( G_{mo} = g_m = 386 \) mA/V, and \( A_{vo} = -g_m(r_o \parallel R_C) = -386 \) mA/V \( \times (20.56 \) kΩ \( \parallel 1 \) kΩ) = -368.5 V/V.

KEY POINTS OF SECTION 8.5

- For analysis of a BJT amplifier, the transistor must be represented by its DC and small-signal AC models. Therefore, two types of analysis are performed: AC analysis and DC analysis.
- Linear DC models are used for determining the \( Q \)-point; however, an AC model is used for determining the voltage or power gain when the transistor is operated as an amplifier in the active region. The parameters of the small-signal models depend on the DC-biasing point.
- The manufacturers of BJTs usually specify the common-emitter hybrid parameters corresponding to the hybrid model whose parameters can be determined from the other small-signal model parameters.

8.6 The BJT Switch

The BJT can be operated as a switch that will have the characteristic of a low on-state voltage at the maximum current so that the switch is subjected to the minimum power loss. This condition requires that the transistor is operated in the saturation region and the B-C junction is reverse biased such that \( V_{CE} \) is a low \( V_{CE} = V_{CB} + V_{BE} = -V_{BC} + V_{BE} = V_{CE(sat)} \). Figure 8.8[(a) or (b)] shows the circuit arrangement for
operating the BJT as a switch. Since \( v_{CE} = v_{CB} + v_{BE} \), and also using \( v_{CE} = V_{CC} - R_C i_C \) from the collector loop, we get

\[
v_{CB} = v_{CE} - v_{BE} = V_{CC} - R_C i_C - v_{BE}
\]

which means that \( v_{CE} \) must be less than \( v_{BE} \), \( v_{CE} < v_{BE} \), for operating the BJT as a switch. This condition can be satisfied by varying the product \( R_C i_C \). The value of \( v_{BE} \) that will make \( v_{CB} \approx 0 \) can be determined from the following condition:

\[
V_{CC} - v_{BE} = R_C i_C = R_C I_S e^{v_{BE}/V_T}
\]

The maximum value of the collector current \( I_{C_{(max)}} \) is specified by the manufacturer data sheet, which limits the minimum value of collector resistance \( R_C \). Assuming that \( V_{CE(sat)} \) is the C-E saturation voltage, we can find the corresponding collector saturation current as given by

\[
I_{C_{(sat)}} = \frac{V_{CC} - V_{CE(sat)}}{R_C}
\]

which sets the limits of the collect or current \( I_{C_{(sat)}} \leq i_C \leq I_{C_{(max)}} \) and the corresponding collector resistance \( R_{C_{(max)}} \leq R_C \leq R_{C_{(min)}} \).

Using KVL in the base loop, we get the base current as

\[
i_B = \frac{V_{BB} - v_{BE}}{R_B}
\]

which must be larger than the minimum base current \( I_{B_{(min)}} \) to drive the transistor into saturation; that is,

\[
I_{B_{(min)}} > I_B = \frac{V_{BB} - v_{BE}}{R_B}
\]

To operate the transistor in the saturation region, the base current must be sufficient enough to maintain the collector saturation current. That is, \( i_B \) must be greater than the value \( I_{C_{(sat)}}/\beta_F \) corresponding collector current \( I_{C_{(sat)}} \); that is,

\[
\frac{V_{BB} - v_{BE}}{R_B} \beta_F > \frac{V_{CC} - V_{CE(sat)}}{R_C}
\]

Too much base current will drive the transistor hard into saturation, giving a low value of \( V_{CE(sat)} \), but it will take a longer time to switch from the on-state to the off-state due to a larger amount of charge storage in the depletion regions. On the other hand, too little of base current may not be enough to keep the transistor into saturation to obtain a low C-E voltage; it needs to make a right judgment. It is recommended to use a 125% overdrive factor (ODF); that is,

\[
I_{B_{(max)}} = \frac{\text{ODF} \ I_{C_{(sat)}}}{\beta_F}
\]

or

\[
\frac{V_{BB} - v_{BE}}{R_B} \beta_F = \frac{V_{CC} - V_{CE(sat)}}{R_C} \times \text{ODF}
\]

Therefore, the condition for the maximum value of the base current in Eq. (8.53) is

\[
I_{B_{(min)}} = I_{B_{(sat)}} = \frac{V_{BB} - v_{BE}}{R_{B_{(max)}}}
\]

which gives limits to the base current \( I_{B_{(min)}} = I_{C_{(max)}}/\beta_F \leq i_B \leq I_{B_{(max)}} \) and the corresponding base resistance \( R_{B_{(max)}} \geq R_B \leq R_{B_{(min)}} \).
8.7 DC Biasing of Bipolar Junction Transistors

If a transistor is used for the amplification of voltage (or current), it is necessary to bias the device. The main reasons for biasing are to turn the device on and, in particular, to place the operating point in the region of its characteristic where the device operates most linearly so that any change in the input signal causes a proportional change in the output signal. In practice, a fixed DC supply is normally used, and the circuit elements are selected so as to bias the C-B and E-B junctions in appropriate magnitude and polarity.

The determination of the DC-biasing point described by \((I_B, I_C, V_{CE})\) is the first step in the analysis of the transistor circuit. Once the values of \(I_B\) and \(I_C\) are found, we can find \(g_m, r_{\pi},\) and \(r_o\); that is, 
\[
g_m = \frac{I_C}{V_T}, \quad r_{\pi} = 25.8 \text{ mV} / I_B, \quad r_o \approx \frac{V_A}{I_C}.
\]
Since the B-E junction behaves like a diode, the transistor needs a B-E voltage of \(V_{BE} \approx 0.7 \text{ V}\) to conduct. If we apply more than 0.7 V, the transistor will be damaged due to excessive current. Resistors are used to limit the transistor currents as shown in Fig. 8.14(a) with base resistor \(R_B\) and collector resistor \(R_C\), Fig. 8.14(b) with emitter resistor \(R_E\), and Fig. 8.14(c) with collector resistor \(R_C\) and emitter resistor \(R_B\). Although there are many types of biasing circuits, we will consider the following types, which are most commonly used:

- Active current–source biasing
- Single–base resistor biasing
- Emitter resistance–feedback biasing
- Emitter-follower biasing
- Two–base resistor biasing

![Resistors for limiting transistor currents](image)

**FIGURE 8.14** Resistors for limiting transistor currents
In the derivations of the currents for these biasing circuits, we will assume the following relations between the transistor currents:

\[ i_c = \beta_F i_B, \quad i_E = (1 + \beta_F) i_B, \quad \text{and} \quad i_C = \alpha_F i_E. \]

### 8.7.1 Active Current–Source Biasing

According to Eq. (8.48), the voltage gain can be increased by having a large value of the collector resistance \( R_C \) in the amplifier of Fig. 8.11(a). \( R_C \) can be replaced by a current source, which normally has a high output resistance, thereby producing a high voltage gain, as shown in Fig. 8.15(a). To allow for a wide output voltage swing, an amplifier is often connected to two DC supplies as shown in Fig. 5.15(b). The current \( I_C \) of the active load flows out of the current source circuit into \( Q_1 \), and this type of source is referred to as a **sourcing current source**. The source current in Fig. 8.15(b) flows from \( Q_1 \) **into** the current source, and this type of constant current source is often referred to as a **current sinking source**. An ideal current source should have a constant current and a very large output resistance \( r_o \), tending to infinity. Since a current source has a large output resistance \( r_o \), the voltage gain of the amplifier will also be large. Current sources are used for biasing transistors in integrated circuits. There are several types of BJT current sources, and these are covered in detail in Sec. 9.6.

The current source for the biasing transistors in Fig. 8.15(a) can be generated by two transistors and a resistor, as shown in Fig. 8.16(a). If a diode characteristic is required in integrated circuits, a transistor is generally operated as a diode so as to avoid another manufacturing process. The B-C junction of a transistor is shorted so that its B-E junction exhibits a diode characteristic. Then this transistor is said to be **diode connected**. Transistor \( Q_3 \) in Fig. 8.16(a) is diode connected, and its C-B voltage is forced to zero. \( Q_3 \) still operates internally as a transistor in the active region, but it exhibits the characteristic of a diode. Let us assume that \( Q_2 \) and \( Q_3 \) are two identical transistors, whose leakage currents are negligible and whose output resistances are large. Since the two transistors have the same B-E voltages \( V_{BE2} = V_{BE3} \), the collector and base currents will be equal. That is,

\[ I_{C2} = I_{C3} \quad \text{and} \quad I_{B2} = I_{B3} \]

Applying Kirchhoff’s current law (KCL) at the collector of \( Q_3 \), we get the reference current:

\[ I_{\text{ref}} = I_{C3} + I_{B2} + I_{B3} = I_{C3} + 2I_{B3} \]
Since \( I_{C3} = \beta_F I_{B3} \),

\[
I_{ref} = I_{C3} + 2I_{B3} = I_{C3} + \frac{2I_{C3}}{\beta_F}
\]

which gives the collector current \( I_{C3} \) as

\[
I_{C3} = I_{C2} \left( 1 + \frac{I_{ref}}{2/\beta_F} \right)
\]  

(8.58)

If \( \beta_F \gg 2 \), which is usually the case, Eq. (8.58) can be approximated by

\[
I_{C3} \approx I_{ref} = \frac{V_{CC} - V_{BE3}}{R} = I_{C2}
\]  

(8.59)

Thus, for two identical transistors, the reference and output currents are equal. In practice, however, the transistors may not be identical, and the two collector currents will have a constant ratio. The small-signal AC equivalent circuit is shown in Fig. 8.16(b). The equivalent circuit for finding \( r_o \) is shown in Fig. 8.16(c), where the output resistance \( r_o \) is the same as \( r_{o2} \):

\[
r_o = \frac{v_x}{i_x} = r_{o2} = \frac{V_A}{I_{C2}} \quad \text{(for } V_A = V_{A2} = V_{A3})
\]  

(8.60)

### 8.7.2 Single–Base Resistor Biasing

This type of biasing circuit, shown in Fig. 8.17, is the same as that in Fig. 8.14(a). Using KVL around the B-E loop, we can find the base current \( i_B \), the collector current \( i_C \), and C-E voltage \( v_{CE} \), for known value of \( V_{BE} \) that is typically 0.7 V, as given by

\[
i_B = \frac{V_{CC} - v_{BE}}{R_B}
\]

\[
i_C = \beta_F i_B = \frac{\beta_F(V_{CC} - v_{BE})}{R_B}
\]  

(8.61)

\[
v_{CE} = V_{CC} - R_C i_C = V_{CC} - \beta_F R_C i_B
\]  

(8.62)
This is a simple arrangement, but $i_C$ and $v_{CE}$ are dependent directly on $\beta_F$, which can vary over a wide range for transistors of the same type.

### 8.7.3 Emitter Resistance–Feedback Biasing

This type of biasing circuit is shown in Fig. 8.18. Using KVL around the B-E loop, we can write

$$V_{CC} = R_B i_B + v_{BE} + R_E i_E = R_B i_B + v_{BE} + R_E (1 + \beta_F) i_B \quad [\text{Use } i_E = (1 + \beta_F) i_B.]$$

which gives the base current, for known value of $V_{BE}$, as

$$i_B = \frac{V_{CC} - v_{BE}}{R_B + R_E (1 + \beta_F)} \quad (8.63)$$

$$i_C = \beta_F i_B = \frac{\beta_F (V_{CC} - v_{BE})}{R_B + R_E (1 + \beta_F)} \quad (8.64)$$

$$v_{CE} = V_{CC} - R_C i_C - R_E i_E = V_{CC} - R_C \beta_F i_B - (1 + \beta_F) R_E i_B \quad (8.65)$$

For $R_E (1 + \beta_F) >> R_B$ and $\beta_F >> 1$, Eq. (8.64) can be approximated to

$$i_C \approx \frac{V_{CC} - v_{BE}}{R_E}$$

FIGURE 8.18 Emitter resistance–feedback biasing
This is also a simple circuit. By proper biasing design, \( i_{C} \) and \( v_{CE} \) can be made almost independent of \( \beta_F \), which can vary over a wide range for transistors of the same type. It is important to note the emitter resistance \( R_E \) appears in series with \( R_B \) in the base circuit as with a multiplying factor of \( \frac{1}{R_B} \). Similarly, the base resistance \( R_B \) will appear in series with \( R_E \) in the emitter circuit as with a multiplying factor of \( \frac{1}{R_E} \).

### 8.7.4 Emitter-Follower Biasing

This type of biasing circuit is shown in Fig. 8.19. This circuit is known as the emitter follower because the output voltage at the emitter terminal is \( v_O = v_B - v_{BE}(\approx 0.7 \text{ V}) \). If the base voltage changes by \( \Delta v_B \), then the output voltage will also change by the same amount \( \Delta v_O = \Delta v_B \). Using KVL around the B-E loop, we can find the base current \( i_B \), the collector current \( i_C \), and the C-E voltage \( v_{CE} \) as given by

\[
i_B = \frac{V_{CC} - v_{BE}}{R_B + R_E(1 + \beta_F)} \tag{8.66}
\]

\[
i_C = \beta_F i_B = \frac{\beta_F(V_{CC} - v_{BE})}{R_B + R_E(1 + \beta_F)} \tag{8.67}
\]

\[
v_{CE} = V_{CC} - R_E i_C = V_{CC} - (1 + \beta_F)R_E i_B \tag{8.68}
\]

### 8.7.5 Two–Base Resistor Biasing

This type of biasing circuit is shown in Fig. 8.20(a). The analysis can be simplified by replacing \( R_1 \) and \( R_2 \) with Thevenin’s equivalent voltage \( V_{Th} \) and resistance \( R_{Th} \) as shown in Fig. 8.20(b). The replacement gives

\[
V_{Th} = \frac{R_2}{R_1 + R_2} V_{CC} \tag{8.69}
\]

\[
R_{Th} = \frac{R_1 R_2}{R_1 + R_2} \tag{8.70}
\]
Equations (8.69) and (8.70) can be solved for $R_1$ as

$$R_1 = \frac{R_{Th}V_{CC}}{V_{Th}} \quad (8.71)$$

Substituting $R_1$ from Eq. (8.71) into Eq. (8.70), we get

$$R_2 = \frac{R_{Th}V_{CC}}{V_{CC} - V_{Th}} \quad (8.72)$$

The biasing analysis is similar to Fig. 8.18 if we substitute $R_{Th}$ for $R_B$ and the base is connected to $V_{Th}$ instead of $V_{CC}$. Using KVL around the B-E loop, we can find the base current $i_B$, the collector current $i_C$, and the C-E voltage $v_{CE}$ as given by

$$i_B = \frac{V_{Th} - V_{BE}}{R_{Th} + R_E(1 + \beta_F)} \quad (8.73)$$

$$i_C = \beta_Fi_B = \frac{\beta_F(V_{Th} - V_{BE})}{R_{Th} + R_E(1 + \beta_F)} \quad (8.74)$$

$$v_{CE} = V_{CC} - R_CI_C - R_Ei_E = V_{CC} - R_CI_C - (1 + \beta_F)R_Fi_B \quad (8.75)$$

$R_E(1 + \beta_F) >> R_{Th}$ and $\beta_F >> 1$, Eq. (8.74) can be approximated to

$$i_C \approx \frac{V_{Th} - V_{BE}}{R_E} \quad (8.76)$$

The voltage at the base $v_B$ is fixed by potential divider consisting of $R_1$ and $R_2$. Since the B-E voltage is the difference between the base and emitter voltages, $v_{BE} = v_B - v_E$, any changes in the base voltage $v_B$ and the emitter voltage $v_E$ due to variations of transistor parameters cause minimum changes in $v_{BE} = v_B - v_E$ and thereby the transistor currents.
8.7.6 Biasing Circuit Design

Using the relationship $i_E = i_C/\alpha_F$, we can rewrite Eqs. (8.65) or (8.75) in terms of $i_C$ as given by

$$v_{CE} = V_{CC} - R_Ci_C - R_Ei_E = V_{CC} - \left( R_C + \frac{R_E}{\alpha_F} \right)i_C \tag{8.77}$$

The value of $\alpha_F$ ranges from 0.9 to 0.99 and is related to $\beta_F$ by

$$\alpha_F = \frac{i_E}{i_C} = \frac{\beta_F}{1 + \beta_F} \tag{8.78}$$

In practice, $\beta_F \gg 1$ and $\alpha_F \approx 1$. Thus, Eq. (8.77) can be approximated to

$$v_{CE} = V_{CC} - (R_C + R_E)i_C \tag{8.79}$$

which is the equation of a straight line and represents the load line, as shown in Fig. 8.21.

Since an AC voltage is normally superimposed on the operating base-to-emitter voltage $V_{BE}$ in order to operate the transistor as an amplifier, the $Q$-point is subjected to a swing in either direction. Therefore, the $Q$-point should be positioned so that it can provide enough range to accommodate the maximum voltage swing and it is least sensitive to variations in the DC gain $\beta_F$.

The given parameters are $\beta_F$ and $I_{C(max)}$ of a transistor, $V_{BE} = 0.7$ V (typically), and $V_{CC}$. Select suitable values of the DC biasing collector current $I_{CQ}$ and the C-E voltage $V_{CEQ}$ so that $i_C$ and $v_{CE}$ can have the maximum swings in both positive and negative directions, $i_C = I_{CQ} \pm i_C(\text{peak})$ and $v_{CE} = V_{CEQ} \pm v_{CE(\text{peak})}$. There, $I_{CQ}$ must be less than $I_{C(max)/2}$, and the DC supply $V_{CC}$ should equally be shared equally by all elements in the C-E loop. The guidelines for determining the biasing resistances of different circuits are summarized in Table 8.1.

![Figure 8.21 Load line and Q-point](image-url)
NOTE  Manufacturers usually specify three values for a parameter: minimum, nominal, and maximum. For example, the beta ($\beta_F$) of Q2N2222 has three values: minimum $\beta_F = 100$, nominal $\beta_F = 173$, and maximum $\beta_F = 300$. It is the designer’s task to choose the appropriate value of the transistor parameter(s) to find the component values. The minimum value of $\beta_F$ is normally used to yield the worst-case design of the biasing circuit—that is, to obtain the desired $Q$-point at the worst value of $\beta_F$. 

### TABLE 8.1  Summary of biasing design guidelines

<table>
<thead>
<tr>
<th>Biasing Parameters</th>
<th>Figure 8.17</th>
<th>Figure 8.18</th>
<th>Figure 8.19</th>
<th>Figure 8.20</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_{CEQ}$</td>
<td>$V_{CC}$/2</td>
<td>$V_{CC}$/3</td>
<td>$V_{CC}$/2</td>
<td>$V_{CC}$/3</td>
</tr>
<tr>
<td>$V_{EQ}$</td>
<td>$V_{CC}$/2</td>
<td>$V_{CC}$/3</td>
<td>$V_{CC}$/2</td>
<td>$V_{CC}$/3</td>
</tr>
<tr>
<td>$I_{CQ}$</td>
<td>$I_{C(max)}/3$</td>
<td>$I_{C(max)}/3$</td>
<td>$I_{C(max)}/3$</td>
<td>$I_{C(max)}/3$</td>
</tr>
<tr>
<td>$R_C$</td>
<td>$V_{CC}$/$2I_{CQ}$</td>
<td>$V_{CC}$/$3I_{CQ}$</td>
<td>0</td>
<td>$V_{CC}$/$3I_{CQ}$</td>
</tr>
<tr>
<td>$R_E$</td>
<td>0</td>
<td>$V_{CC}$/$3I_{CQ}\alpha_F$</td>
<td>$V_{CC}$/$2I_{CQ}\alpha_F$</td>
<td>$V_{CC}$/$3I_{CQ}\alpha_F$</td>
</tr>
<tr>
<td>$R_B$</td>
<td>$V_{CC} - V_{BE}$/$I_{BQ}$</td>
<td>$V_{CC} - V_{BE} - V_{EQ}$/$I_{BQ}$</td>
<td>$V_{CC} - V_{BE} - V_{EQ}$/$I_{BQ}$</td>
<td>$V_{CC} - V_{BE} - V_{EQ}$/$I_{BQ}$</td>
</tr>
<tr>
<td>$R_{Th}$</td>
<td>$0.1 \times (1 + \beta_F)R_E$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$R_1$</td>
<td>$R_{Th}V_{CC}$/$V_{Th}$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$R_2$</td>
<td>$R_{Th}V_{CC}$/$V_{CC} - V_{Th}$</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

NOTE  Manufacturers usually specify three values for a parameter: minimum, nominal, and maximum. For example, the beta ($\beta_F$) of Q2N2222 has three values: minimum $\beta_F = 100$, nominal $\beta_F = 173$, and maximum $\beta_F = 300$. It is the designer’s task to choose the appropriate value of the transistor parameter(s) to find the component values. The minimum value of $\beta_F$ is normally used to yield the worst-case design of the biasing circuit—that is, to obtain the desired $Q$-point at the worst value of $\beta_F$. 

### EXAMPLE 8.3

**Designing a BJT biasing circuit**

(a) Design a transistor biasing circuit as shown in Fig. 8.20(a). Use transistor Q2N2222, for which minimum $\beta_F = 100$, nominal $\beta_F = 173$, $I_S = 3.295 \times 10^{-14} \text{ A}$, and $V_A = 200 \text{ V}$. The operating collector current is to be set at $I_C = 10 \text{ mA}$. The DC power supply is $V_{CC} = 15 \text{ V}$. Assume $V_{BE} = 0.7 \text{ V}$.

(b) Calculate the small-signal parameters $r_\pi$, $g_m$, and $r_0$ of the transistor.

(c) Use PSpice/SPICE to verify your results in parts (a) and (b).

**SOLUTION**

We will design for the worst-case value of $\beta_F$ (i.e., minimum $\beta_F = 100$). $I_C = 10 \text{ mA}$, $I_B = 10 \text{ mA}/100 = 0.1 \text{ mA}$, and $V_{CC} = 15 \text{ V}$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
(a) **Step 1.** Calculate the values of $\alpha_F$ and $I_E$. From Eq. (8.78),

$$\alpha_F = \frac{\beta_F}{(1 + \beta_F)} = \frac{100}{(1 + 100)} = 0.99$$

From Eq. (8.4),

$$I_E = \frac{I_C}{\alpha_F} = \frac{10 \text{ mA}}{0.99} = 10.1 \text{ mA}$$

**Step 2.** Calculate the value of $V_E$. From Table 8.1 (Fig. 8.20),

$$V_E = \frac{V_{CC}}{3} = \frac{15}{3} = 5 \text{ V}$$

**Step 3.** Calculate the value of $R_E$ and its power rating:

$$R_E = \frac{V_E}{I_E} = \frac{5 \text{ V}}{10.1 \text{ mA}} = 495 \Omega$$

The power rating of $R_E$ is

$$P_{RE} = I_E^2 R_E = (10.1 \times 10^{-3})^2 \times 495 = 50.49 \text{ mW}$$

**Step 4.** Calculate the value of $V_{CE}$. From Table 8.1 (Fig. 8.20),

$$V_{CE} = \frac{V_{CC}}{3} = \frac{15}{3} = 5 \text{ V}$$

**Step 5.** Calculate the value of $R_C$ and its power rating.

$$I_C R_C = V_{CC} - V_E - V_{CE} = 15 - 5 - 5 = 5 \text{ V}$$

$$R_C = \frac{5}{I_C} = \frac{5}{10 \text{ mA}} = 500 \Omega$$

The power rating of $R_C$ is

$$P_{RC} = I_C^2 R_C = (10 \times 10^{-3})^2 \times 500 = 50 \text{ mW}$$

**Step 6.** Calculate the values of $R_{Th}$ and $V_{Th}$. From Table 8.1 (Fig. 8.20),

$$R_{Th} = \frac{(1 + \beta_F) R_E}{10} = \frac{(1 + 100) \times 495}{10} = 5 \text{ k}\Omega$$

From Fig. 8.20(b),

$$V_{Th} = V_E + V_{BE} + R_{Th} I_B = 5 \text{ V} + 0.7 \text{ V} + 5 \text{ k}\Omega \times 0.1 \text{ mA} = 5 + 0.7 + 0.5 = 6.2 \text{ V}$$

**Step 7.** Calculate the value of $R_1$ and its power rating. From Eq. (8.71),

$$R_1 = \frac{R_{Th} V_{CC}}{V_{Th}} = \frac{5 \times 15}{6.2} = 12.1 \text{ k}\Omega$$

The power rating of $R_1$ is

$$P_{R1} = \frac{(V_{CC} - V_{Th})^2}{R_1} = \frac{(15 - 6.2)^2}{12.1 \text{ k}\Omega} = 6.4 \text{ mW}$$

**Step 8.** Calculate the value of $R_2$ and its power rating. From Eq. (8.72),

$$R_2 = \frac{R_{Th} V_{CC}}{V_{CC} - V_{Th}} = \frac{5 \times 15}{15 - 6.2} = 8.52 \text{ k}\Omega$$
The power rating of $R_2$ is

$$P_{R2} = \frac{V_{Th}^2}{R_2} = \frac{6.2^2}{8.52 \text{ k}\Omega} = 4.51 \text{ mW}$$

(b) From Eq. (8.40),

$$r_m = \frac{25.8 \text{ mV}}{I_B} = 100 \times \frac{25.8 \text{ mV}}{10 \text{ mA}} = 258 \Omega$$

From Eq. (8.42),

$$g_m = \frac{I_C}{V_T} = \frac{10 \text{ mA}}{25.8 \text{ mV}} = 387.6 \text{ mA/V}$$

From Eq. (8.43),

$$r_o = \frac{V_{CE} + V_A}{I_C} = \frac{(5 + 200) \text{ V}}{10 \text{ mA}} = 20.5 \text{ k}\Omega$$

(e) The DC-biasing circuit for PSpice simulation is shown in Fig. 8.22.

The results of the .OP command (for EX8-3.SCH) are automatically printed on the output file. (The values obtained from hand calculations are shown in parentheses.)

** SMALL SIGNAL BIAS SOLUTION  TEMPERATURE = 27.000 DEG C

<table>
<thead>
<tr>
<th>NODE</th>
<th>VOLTAGE</th>
<th>NODE</th>
<th>VOLTAGE</th>
<th>NODE</th>
<th>VOLTAGE</th>
<th>NODE</th>
<th>VOLTAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1)</td>
<td>5.2519</td>
<td>(2)</td>
<td>15.0000</td>
<td>(3)</td>
<td>10.4270</td>
<td>(4)</td>
<td>4.5710</td>
</tr>
</tbody>
</table>

**NOTE:** All PSpice results given here are from running the simulation with the schematic (.SCH) files. If you run the simulation with the netlist circuit (.CIR) files, you may get different results, as the student’s version of PSpice has a limited number of active devices and models.

**FIGURE 8.22** DC-biasing circuit for PSpice simulation
8.8 Common-Emitter Amplifiers

Once the $Q$-point has been established by a biasing circuit, an input voltage can be applied through coupling capacitors, as shown in Fig. 8.23. $C_1$ and $C_2$ isolate the DC signals of the biasing circuit from the input signal $v_s$ and the load resistance $R_L$, respectively. If the input signal $v_s$ were connected directly to the base without $C_1$, the source resistance $R_s$ would form a parallel circuit with $R_2$, and the base potential $V_B$ would be disturbed. Similarly, the collector potential $V_C$ would depend on $R_L$ if $C_2$ were removed.

**FIGURE 8.23** Common-emitter amplifier circuit
Let us assume that the capacitors have large values so that they are virtually shorted at the frequency of the input signal $v_s$. With a sinusoidal input voltage $v_s = V_m \sin \omega t$, the base potential will be $v_B = V_B + v_b$. If the base current $i_B$ swings between $I_B + i_B(\text{peak})$ and $I_B - i_B(\text{peak})$, the collector current $i_C$ will swing between $I_C + i_C(\text{peak})$ and $I_C - i_C(\text{peak})$. The C-E voltage $v_{CE}$ will vary accordingly from $V_{CE} - v_{ce(\text{peak})}$ to $V_{CE} + v_{ce(\text{peak})}$, and the collector voltage $v_C$ will vary from $V_C - R_C(I_C + i_C(\text{peak}))$ to $V_C - R_C(I_C - i_C(\text{peak}))$. These waveforms are depicted in Fig. 8.23. Since $C_2$ will block any DC signal, the output voltage will vary from $-(RC || RL)i_{c(\text{max})}$ to $(RC || RL)i_{c(\text{min})}$.

### 8.8.1 Active-Biased Common-Emitter Amplifier

A CE amplifier with a current source is shown in Fig. 8.24(a). The current source consists of $pnp$ transistors to generate a sourcing current source, and its output resistance acts as the load of transistor $Q_1$. Since the collector load element is a $pnp$ transistor instead of a resistor, it is said to be active. Since a DC supply offers zero impedance to an AC signal, $V_{CC}$ behaves as short-circuited; that is, one side of both $Q_2$, $Q_3$, and $R_B$ is connected to the ground. Replacing the transistors by their small-signal models gives the AC equivalent circuit shown in Fig. 8.24(b). The small-signal analysis is similar to the basic amplifier circuit in Fig. 8.13(b) if we replace $R_C$ by $r_o2$ and by in parallel with $R_B$. For identical transistors of matched characteristics, we can assume $V_A1 = V_A2 = V_A$, $r_{o1} = r_{o2} = r_o$, and $g_{m1} = g_{m2} = g_m$. The small-signal input resistance can be found from

$$R_1 = \frac{V_b}{i_s} = r_{\pi1} \parallel R_B \tag{8.80}$$

The output resistance $R_o$ seen looking into the output is the parallel combination of the two transistor output resistances. That is,

$$R_o = r_{o2} \parallel r_{o1} \tag{8.81}$$

For $v_{eb2} = 0$, $g_m v_{eb2}$ is open circuited. The output voltage $v_o$ is

$$v_o = -g_{m1}(r_{o2} \parallel r_{o1})v_{be1}$$
which gives the no-load voltage gain \( A_{vo} \) as

\[
A_{vo} = \frac{V_o}{V_b} = -g_{m1}(r_{o2} || r_{o1}) \tag{8.82}
\]

which, after substituting \( r_{o1} = r_{o2} = V_A/I_C \) and \( g_{m1} = g_{m2} = I_C/V_T \), can be simplified to

\[
A_{vo} = \frac{I_C V_A}{V_T 2I_C} = \frac{V_A}{2V_T} \tag{8.83}
\]

It is interesting to note that the no-load voltage gain is independent of the DC biasing collector current, but it depends directly on the Early voltage and decreases inversely with the thermal voltage \( V_T \). For larger voltage gain, we should use transistors with a higher value of Early voltage \( V_A \).

**EXAMPLE 8.4**

**Designing a common-emitter amplifier with an active current source**

(a) Design a CE amplifier with an active current source, as shown in Fig. 8.24(a). Use transistors Q2N2222 and Q2N2907, for which nominal \( \beta_f = \beta_t = 173, I_S = 3.295 \times 10^{-14} \text{A}, \) and \( V_A = 100 \text{V}. \) The operating collector current is set at \( I_C = 10 \text{mA} \). The DC power supply is \( V_{CC} = 15 \text{V}. \) Assume \( V_{BE} = 0.7 \text{V}. \)

(b) Use PSpice/SPICE to verify your results in part (a).

**SOLUTION**

(a) **Step 1.** Design the biasing current source. \( I_C = 10 \text{mA} \), and \( V_{CC} = 15 \text{V}. \) From Eq. (8.59), we can find the value of \( R \) in order to set the biasing current to \( I_{ref} = 10 \text{mA} \).

\[
R = \frac{V_{CC} - V_{BE}}{I_{ref}} = \frac{15 - 0.7}{10 \text{mA}} = 1.43 \text{ k}\Omega
\]

**Step 2.** Find the small-signal parameters of the transistors. From Eq. (8.40),

\[
r_{\pi1} = r_{\pi2} = \frac{25.8 \text{ mV}}{I_B} = \frac{25.8 \text{ mV}}{10 \text{ mA}} = 446 \Omega
\]

From Eq. (8.42),

\[
g_{m1} = g_{m2} = \frac{I_C}{V_T} = \frac{10 \text{ mA}}{25.8 \text{ mV}} = 387.6 \text{ mA/V}
\]

From Eq. (8.43),

\[
r_{o1} = r_{o2} = \frac{V_A}{I_C} = \frac{100 \text{V}}{10 \text{mA}} = 10 \Omega
\]

**Step 3.** Evaluate the values of the input resistance \( R_i (=v_b/i_b) \), the open-circuit voltage gain \( A_{vo} \), and the output resistance \( R_o \). We know that

\[
R_i = r_{\pi1} = 446 \Omega
\]

From Eq. (8.81),

\[
R_o = r_{o2} \parallel r_{o1} = 10 \Omega \parallel 10 \Omega = 5 \text{k}\Omega
\]
From Eq. (8.82),

\[
A_v = -g_{m1}(r_{\alpha2} \parallel r_{\alpha1}) = -387.6 \text{ mA/V} \times 5 \text{ k}\Omega = -1938
\]

(b) From Eq. (8.1), we can find the value of \(v_{BE1}\) needed to give \(I_C = 10 \text{ mA}\) for \(I_S = 3.295 \times 10^{-14} \text{ A}\):

\[
10 \text{ mA} = 3.295 \times 10^{-14} \times \exp\left(\frac{v_{BE1}}{25.8 \text{ mV}}\right)
\]

which gives \(v_{BE1} = 0.682 \text{ V}\).

Using Eq. (8.61), we can find the biasing resistor \(R_B:\)

\[
R_B = \frac{\beta_F(V_{CC} - V_{BE})}{I_C} = \frac{173 \times (15 - 0.682)}{10 \times 10^{-3}} = 247.7 \text{ k}\Omega \quad \text{(Use 250 k}\Omega\text{.)}
\]

The CE amplifier with an active load for PSpice simulation is shown in Fig. 8.25.

The PSpice plot of the transfer function \(v_o = V(Q1:C)\) against \(v_S\) is shown in Fig. 8.26. Notice that the operating range of the input voltage is very small—that is, 8.563 mV (709.863 mV - 701.30 mV). Thus, the small-signal gain becomes \(-15 \text{ V/8.563 mV} = -1752\). Details of the .TF analysis are given below. The values obtained from hand calculations are shown in parentheses. The value of \(v_S (=v_{BE1})\) was adjusted to 0.705 V instead of 0.682 V in order to operate in the linear range of the amplifier and to illustrate the benefit of using an active load. Since the voltage gain is very large, any small change in \(v_S\) could drive the amplifier into saturation. Thus, if we build the amplifier and test it in the laboratory with a value of \(v_S = 0.682 \text{ V}\), it might not work; we need to adjust \(v_S\). Using transistor circuit model, we make the range of input voltage from 681 mV to 685 mV.
8.8.2 Resistive-Biased Common-Emitter Amplifier

A CE amplifier with an emitter resistance is shown in Fig. 8.27 where \( C_1 \) and \( C_2 \) are the coupling capacitors to isolate the small-signal input and output voltages from the DC biasing. \( C_E \) will act as a short circuit and bypasses the small AC signal, which results in a higher voltage gain but lowers the small-signal input resistance. Since a DC supply offers zero impedance to an AC signal, \( V_{CC} \) can be short-circuited. That is, one side of both \( R_C \) and \( R_B \) is connected to the ground. The AC equivalent circuit of the amplifier is shown in Fig. 8.28(a), which is similar to Fig. 8.18 except the DC supply \( V_{CC} \) and the capacitors \( C_E, C_1, \) and \( C_2 \) are shorted. Replacing the transistor \( Q_1 \) by its model of Fig. 8.28(c), the small-signal AC equivalent circuit is shown in Fig. 8.28[(b) or (c)], which can be represented by an equivalent voltage amplifier as shown in Fig. 8.28(d). We will consider \( R_L \) as an external element to the amplifier so that the effect of loading can be determined. Thus, \( R_L \) is not included in Fig. 8.28(b).

![Figure 8.27 Common-emitter BJT amplifier with one emitter resistor](image)
We will derive the input resistance $R_i$, the output resistance $R_o$, and the no-load voltage gain $A_{vo}$ of common-emitter BJT amplifier. We will include the biasing resistance $R_B$, which will reduce the value of $R_i$.

**Input Resistance $R_i$**

Using KVL around loop I formed by $r_\pi$ and $R_{E1}$ in Fig. 8.28(c), we have

$$v_b = i_b r_\pi + R_{E1} i_c = i_b (r_\pi + (1 + g_m r_\pi) R_{E1})$$

which gives the resistance $R_x$ at the base of the transistor as

$$R_x = \frac{v_b}{i_b} = r_\pi + (1 + g_m r_\pi) R_{E1} = r_\pi + (1 + \beta_i) R_{E1}$$

Thus, the input resistance of the amplifier is the parallel combination of $R_1$, $R_2$, and $R_x$. That is,

$$R_i = \frac{v_b}{i_b} = R_1 \parallel R_2 \parallel R_x = R_B \parallel R_x$$

where

$$R_B = R_1 \parallel R_2$$
Thus, $R_i$ depends on $R_E$, $R_1$, and $R_2$. Their values can be chosen to give the input resistance required of the amplifier.

**Output Resistance $R_o$**

Output resistance $R_o$, which is Thevenin’s resistance, can be calculated from Fig. 8.28(c) if $v_s$ is shorted and a test voltage $v_x$ is applied across $R_C$. Since $v_s = 0$, the dependent source current will be zero—that is, the circuit will be open. The output resistance will simply be $R_C$. That is,

$$R_o = R_C$$  \hspace{1cm} (8.88)

**Open-Circuit (or No-Load) Voltage Gain $A_{v_o}$**

The open-circuit output voltage from Fig. 8.28(c) is

$$v_o = -R_C i_c = -R_C g_m v_{be}$$  \hspace{1cm} (8.89)

The B-E voltage $v_{be}$, which controls the collector current, can be related to $r_\pi$ by

$$v_{be} = r_\pi i_b$$  \hspace{1cm} (8.90)

Substituting $i_b$ from Eq. (8.84) into Eq. (8.90) yields

$$v_{be} = \frac{r_\pi}{r_\pi + (1 + g_m r_\pi) R_{E1}} v_b$$  \hspace{1cm} (8.91)

Substituting $v_{be}$ from Eq. (8.91) into Eq. (8.89) gives the output voltage

$$v_o = -R_C g_m \frac{r_\pi}{r_\pi + (1 + g_m r_\pi) R_{E1}} v_b$$

which gives the open-circuit voltage gain $A_{v_o}$ as

$$A_{v_o} = \frac{v_o}{v_b} = -\frac{g_m r_\pi R_C}{r_\pi + (1 + g_m r_\pi) R_{E1}} = \frac{-\beta_i R_C}{r_\pi + (1 + \beta_i) R_{E1}}$$  \hspace{1cm} (8.92)

This equation indicates that the voltage gain $A_{v_o}$ can be made large (1) by making $R_{E1} = 0$, (2) by using a transistor with a large value of $g_m$ (or $\beta_i$), and (3) by choosing a high value of $R_C$. For $R_{E1} = R_E - R_{E2} = 0$, Eq. (8.92) gives the maximum open-circuit voltage gain as

$$A_{v_o(max)} = -\frac{g_m r_\pi R_C}{r_\pi} = -g_m R_C = \frac{-\beta_i R_C}{r_\pi}$$  \hspace{1cm} (8.93)

Making $R_{E1} = 0$ will decrease the input resistance $R_x$ in Eq. (8.85), and the amplifier will draw more current from the input source, but the DC-biasing point also depends on $R_C$ and $R_E$. These conflicting constraints—a higher value of $R_{E1}$ for a larger input resistance and a lower value for a larger voltage gain—can be satisfied by using two emitter resistors $R_{E1}$ and $R_{E2}$, as shown in Fig. 8.29. $R_{E1}$ and $R_{E2}$ set the DC-biasing point, and $R_{E1}$ gives the desired AC input resistance or voltage gain. For DC-biasing calculations, however, $R_E (=R_{E1} + R_{E2})$ should be used. It is often necessary to compromise among the design specifications for the biasing point, the input resistance, and the open-circuit voltage gain. It is not always possible to satisfy all the design specifications with one amplifier stage.
EXAMPLE 8.5

Designing a common-emitter BJT amplifier
(a) Design a CE amplifier as shown in Fig. 8.29 to give a voltage gain of $|A_{vo}| = v_o/v_b \geq 20$. Use transistor Q2N2222, for which minimum $\beta_T = 100$, nominal $\beta_T = 173$, $I_S = 3.295 \times 10^{-14}$ A, and $V_A = 100$ V. The operating DC collector current is to be set at $I_C = 10$ mA. The DC power supply is $V_{CC} = 15$ V. Assume $V_{BE} = 0.7$ V.

(b) Use PSpice/SPICE to verify your results in part (a).

SOLUTION
(a) Step 1. Design the biasing circuit. The results of Example 8.3 give $R_C = 500$ $\Omega$, $R_E = 495$ $\Omega$, $R_1 = 12.1$ k$\Omega$, and $R_2 = 8.52$ k$\Omega$.

Step 2. Find the small-signal parameters of the transistor. The results of Example 8.3 give $r_\pi = 258$ $\Omega$, $g_m = 387.6$ mA/V, $\beta_T = 100$, and $r_0 = 20$ k$\Omega$ (which can be ignored for hand calculations).

Step 3. Find the values of $C_1$, $C_2$, $C_E$, and $R_{E1}$. Let us choose $C_1 = C_2 = C_E = 10$ $\mu$F. The worst-case maximum possible gain that we can obtain from transistor Q2N2222 operating at $I_C = 10$ mA can be found from Eq. (8.93):

$$|A_{vo(max)}| = \frac{\beta_T R_C}{r_\pi} = \frac{100 \times 500}{258} = 193.8 \text{ V/V}$$

The desired gain is less than the maximum possible value, so we can proceed with the design. Otherwise, we would need to choose another transistor with a higher value of $\beta_T$. The value of unbypassed emitter resistance $R_{E1}$ in Fig. 8.29 can be found from Eq. (8.92). That is,

$$r_\pi + (1 + \beta_T)R_{E1} = \frac{\beta_T R_C}{|A_{vo}|}$$

which, for $|A_{vo}| = 20$, $\beta_T = 100$, $R_C = 500$ $\Omega$, and $r_\pi = 258$ $\Omega$, gives $R_{E1} = 22.2$ $\Omega$ and $R_{E2} = R_E - R_{E1} = 495 - 22.2 = 472.8$ $\Omega$.

(b) The transistor circuit for PSpice simulation is shown in Fig. 8.30. Note that a signal source resistance $R_s = 0.1$ $\Omega$ is connected at the input side to make the PSpice circuit more general.

The results of .OP analysis obtained from the output file follow. (The values obtained from hand calculations are shown in parentheses.)
The PSpice plots of the base voltage $v_B = V(C1:2)$, the collector voltage $v_C = V(RC:1)$, the input voltage $v_s = V(vs:+)$, and the load voltage $v_o = V(RL:2)$ are shown in Fig. 8.31. Notice that $v_B$ and $v_C$ have a DC value with an AC signal superimposed on them such that $v_B = V_B + v_s$ and $v_C = V_C + v_o$. Capacitor $C_1$ superimposes $v_s$ on $V_B$, whereas capacitor $C_2$ separates the amplified AC voltage (i.e., the output voltage $v_o$) from $V_C$. The voltage gain $v_o/v_s$ is 16.77, which is less than the desired value of 20. Thus, the design calculation should be repeated until the desired gain is obtained. We might try reducing the value of $R_{E1}$ and increasing the value of $R_{E2}$ by the same amount so that the biasing point remains fixed. Even if we modify the design to satisfy the specifications, the results can be expected to differ from those that would be obtained in the laboratory, although not significantly due to the variable value of $\beta_F$.

FIGURE 8.30 Transistor circuit for PSpice simulation

FIGURE 8.31 PSpice plots for Example 8.5
8.9 Emitter Followers

A common-collector amplifier is generally known as an emitter follower because the emitter voltage follows the voltage at the base terminal. Such an amplifier has a low output resistance and a high input resistance. It is commonly used as a buffer stage between a load and the source.

8.9.1 Active-Biased Emitter Follower

This arrangement is similar to the circuit in Fig. 8.19, except that the emitter resistance $R_E$ is replaced by the sinking current source using npn transistors shown in Fig. 8.16(a). This is shown in Fig. 8.32(a). Replacing the transistors by their signal model, the AC equivalent circuit is shown in Fig. 8.32(b), which can be simplified to Fig. 8.32(c).

**Input Resistance $R_i$**

When the load resistance $R_L$ is connected to the amplifier, $R_L$ becomes parallel to $R_E$ and will affect the input resistance. Unlike the case of a CE amplifier, in an emitter follower, $R_L$ should be included in finding the input resistance $R_i$. Using KVL around the B-E loop I in Fig. 8.32(c), we get

$$v_b = i_b r_\pi + i_c (r_{o1} \parallel r_{o2} \parallel R_L) = i_b [r_\pi + (1 + \beta_f) (r_{o1} \parallel r_{o2} \parallel R_L)]$$

which gives the resistance $R_x$ at the base of the transistor as

$$R_x = \frac{v_b}{i_b} = r_\pi + (1 + \beta_f) (r_{o1} \parallel r_{o2} \parallel R_L)$$

(8.95)

(8.96)
Input resistance $R_i$, which is the parallel combination of $R_B$ and $R_x$, is

$$R_i = \frac{v_b}{i_s} = R_B \parallel R_x \quad (8.97)$$

**Open-Circuit (or No-Load) Voltage Gain $A_{v0}$**

The open-circuit output voltage from Fig. 8.32(c) $v_o$ is

$$v_o = i_c R_E = (i_b + g_m v_{be})(r_{o1} \parallel r_{o2}) \quad \text{where } v_{be} = v_{be1}$$

Since $v_{be} = r_n i_b$, we get

$$v_o = (i_b + g_m v_{be}) R_E = (1 + g_m r_n) i_b(r_{o1} \parallel r_{o2}) = (1 + \beta i_b(r_{o1} \parallel r_{o2})$$
Substituting $i_b$ from Eq. (8.95), we get the no-load voltage $v_o$ as

$$v_o = \frac{(1 + \beta_f)(r_{o1} \parallel r_{o2})}{r_\pi + (1 + \beta_f)(r_{o1} \parallel r_{o2})} v_b$$

which gives the open-circuit voltage gain $A_{vo}$ as

$$A_{vo} = \frac{v_o}{v_b} = \frac{(1 + g_m r_\pi)(r_{o1} \parallel r_{o2})}{r_\pi + (1 + \beta_f)(r_{o1} \parallel r_{o2})} = \frac{(1 + \beta_f)(r_{o1} \parallel r_{o2})}{1 + r_\pi/[(1 + \beta_f)(r_{o1} \parallel r_{o2})]}$$  (8.98)

For $r_\pi \ll (1 + \beta_f)(r_{o1} \parallel r_{o2})$, which is usually the case for current sources, Eq. (8.98) can be approximated by $A_{vo} \approx 1$.

**Output Resistance $R_o$**

Output resistance $R_o$ can be calculated by applying a test voltage $v_x$ across the output terminals and shorting the input source $v_s$, as shown in Fig. 8.33. To account for the effect of $r_o$ on $R_o$, we include $r_o$ in Fig. 8.33. The base current $i_b$ flows through $r_\pi$, which is in series with the parallel combination of $R_s$ and $R_B$, so

$$i_b = \frac{-v_x}{r_\pi + (R_s \parallel R_B)}$$  (8.99)

Using KCL at the emitter junction (E) yields

$$i_x = \frac{v_x}{r_{o1}} + \frac{v_x}{r_{o2}} - g_m v_{be} - i_b$$

Since $v_{be} = -i_b r_\pi$, substituting $i_b$ from Eq. (8.97) into the above equation gives

$$i_x = \frac{v_x}{r_{o1}} + \frac{v_x}{r_{o2}} + \frac{g_m r_\pi v_x}{r_\pi + (R_s \parallel R_B)} + \frac{v_x}{r_\pi + (R_s \parallel R_B)} = v_x \left[ \frac{1}{r_{o1}} + \frac{1}{r_{o2}} + \frac{1 + g_m r_\pi}{r_\pi + (R_s \parallel R_B)} \right]$$

**FIGURE 8.33** Equivalent circuit for determining output resistance $R_o$
which gives the output resistance $R_o$ as

$$R_o = \frac{v_x}{i_x} = r_{o1} \parallel r_{o2} \parallel \frac{r_{\pi} + (R_s \parallel R_B)}{1 + g_m r_{\pi}}$$  \hspace{1cm} (8.100)

Thus, $R_o$ is the parallel combination of $r_{o1}$, $r_{o2}$, and $[r_{\pi} + (R_s \parallel R_B)]$ reflected from the $i_b$ branch into the $i_e$ branch. Since $\beta_I >> 1$ and $R_s << R_B$, the output resistance $r_{o2}$ can be approximated by $R_o = (r_{\pi} + R_s)/\beta_I$.

Note from Eqs. (8.96) and (8.98) that the output resistance $r_{o2}$ of the current source should be as high as possible to give a higher input resistance $R_i$ and a higher voltage gain $A_{vo}$. Since $r_{o2}$ is generally high, the active-biased emitter follower gives a higher input resistance and almost close-to-unity voltage gain.

An ideal current source should maintain a constant current at an infinite output resistance under all operating conditions. To achieve this goal, a number of current sources are developed in Chapter 9. The commonly used current sources are the basic current source (which was applied in Secs. 8.8.1 and 8.9.1), the modified basic source, the Widlar current source, the cascaded current source, and the Wilson current source.

**NOTE** For a current source of $I_{reg} = 5 \, \mu A$, a resistor of $R = 5.72 \, M \Omega$ would be required. Resistors of such high values are costly in terms of the die area. Resistors over 50 k\Omega are generally avoided for integrated circuit applications. Thus, this current source is not suitable for generating a current of less than about 0.6 mA at $V_{CC} = 30 \, V$ and 0.3 mA at $V_{CC} = 15 \, V$.

### 8.9.2 Resistive-Biased Emitter Follower

The coupling capacitors $C_1$ and $C_2$ are connected to the circuit in Fig. 8.19 to feed the input signal $v_x$. This arrangement is shown in Fig. 8.34(a) in which $R_B$ sets the biasing base current. The equivalent circuit for DC analysis is shown in Fig. 8.34(b). Replacing the transistor by its model of Fig. 8.10(a), the small-signal AC equivalent circuit of the emitter follower is shown in Fig. 8.34(c). Figure 8.34(d) shows the emitter follower using the two–base resistor biasing as shown in Fig. 8.20.

The analysis of this circuit is similar to that of the circuit in Fig. 8.32(a), and we can use the same equations by substituting $r_{o2}$ by $R_E$. From Eqs. (8.96) and (8.97), we can find the input resistance as given by

$$R_i = R_B \parallel (r_{\pi} + (1 + \beta_I)(r_{o1} \parallel R_E \parallel R_L))$$  \hspace{1cm} (8.101)

From Eq. (8.98), we can find the no-load voltage gain as given by

$$A_{vo} = \frac{(1 + g_m r_{\pi})(r_{o1} \parallel R_E)}{1 + (1 + g_m r_{\pi})(r_{o1} \parallel R_E)} = \frac{(1 + \beta_I)(r_{o1} \parallel R_E)}{1 + (1 + \beta_I)(r_{o1} \parallel R_E)}$$  \hspace{1cm} (8.102)

From Eq. (8.100), we can find the output resistance as given by

$$R_o = (r_{o1} \parallel R_E) \parallel \frac{r_{\pi} + (R_s \parallel R_B)}{1 + \beta_I}$$  \hspace{1cm} (8.103)
EXAMPLE 8.6

Designing an emitter follower

(a) Design an emitter follower with the topology shown in Fig. 8.34(a). Use transistor Q2N2222, for which minimum $\beta_F = 100$, nominal $\beta_F = 173$, $I_S = 3.295 \times 10^{-14}$ A, and $V_A = 200$ V. The operating collector current is set at $I_C = 10$ mA. The DC power supply is $V_{CC} = 15$ V. Assume $V_{BE} = 0.7$ V, $R_L = 5$ kΩ, and $R_s = 250$ Ω.

(b) Use PSpice/SPICE to verify your results in part (a).

SOLUTION

(a) Step 1. Design the biasing circuit. $I_C = 10$ mA, and $V_{CC} = 15$ V. We will design for the worst-case value of $\beta_F = 100$.

$I_E = \frac{(1 + \beta_F)I_C}{\beta_F} = \frac{101 \times 10 \text{ mA}}{100} = 10.1 \text{ mA}$

$I_B = \frac{I_C}{\beta_F} = \frac{10 \text{ mA}}{100} = 0.1 \text{ mA}$
From Table 8.1 for Fig. 8.19, \( V_E = V_{CC}/2 = 15/2 = 7.5 \) V, which gives the value of

\[
R_E = \frac{V_E}{I_E} = \frac{7.5}{10.1 \text{ mA}} = 743 \Omega
\]

The power rating of \( R_E \) is

\[
P_{RE} = I_E^2R_E = (10.1 \text{ mA})^2 \times 743 = 758 \text{ mW}
\]

The base voltage \( V_B \) becomes

\[
V_B = V_E + V_{BE} = 7.5 + 0.7 = 8.2 \text{ V}
\]

The value of \( R_B \) can be found from

\[
R_B = \frac{V_{CC} - V_B}{I_B} = \frac{(V_{CC} - V_B)\beta_F}{I_C}
\]

\[
= \frac{(15 - 8.2) \times 100}{10 \text{ mA}} = 68 \text{ k}\Omega
\]

The power rating of \( R_B \) is

\[
P_{RB} = I_B^2R_B = (0.1 \text{ mA})^2 \times 68 \text{ k}\Omega = 0.68 \text{ mW}
\]

**Step 2.** Find the small-signal parameters of the transistor. The results of Example 8.3 give \( r_m = 258 \) Ω, \( g_m = 387.6 \text{ mA/V}, \beta_F = 100, \) and \( r_{o1} = 20 \text{ k}\Omega. \)

**Step 3.** Find the values of \( C_1 \) and \( C_2. \) Let us choose \( C_1 = C_2 = 10 \mu\text{F}. \)

**Step 4.** Evaluate the values of the input resistance \( R_i, \) the open-circuit voltage gain \( A_{vo}, \) and the output resistance \( R_o. \) From Eq. (8.96),

\[
R_x = r_m + (1 + \beta_F)(R_E \parallel R_L) = 258 + (1 + 100) \times (20 \text{ k} \parallel 743 \parallel 5 \text{ k}) = 65.59 \text{ k}\Omega
\]

From Eq. (8.97),

\[
R_i = R_B \parallel R_x = 68 \text{ k} \parallel 65.59 \text{ k} = 33.4 \text{ k}\Omega
\]

\[
R_m = \frac{V_x}{I_x} = R_i + R_x = 33.4 \text{ k} + 250 = 33.6 \text{ k}\Omega
\]

From Eq. (8.102),

\[
A_{vo} = \frac{(1 + \beta_F)(R_E \parallel r_{o1})}{r_m + (1 + \beta_F)(R_E \parallel r_{o1})}
\]

\[
= \frac{(1 + 100) \times (743 \parallel 20 \text{ k})}{258 + (1 + 100) \times (743 \parallel 20 \text{ k})} = 0.9966
\]

From Eq. (8.103),

\[
R_o = R_E \parallel r_{o1} \parallel \frac{r_m + (R_e \parallel R_{in})}{1 + \beta_F} = 743 \parallel 20 \text{ k} \parallel \frac{258 + 250}{1 + 100} = 5 \text{ \Omega}
\]

The output resistance including \( R_L \) is

\[
R_{out} = R_L \parallel R_o = 5 \text{ k} \parallel 5 \approx 4.99 \text{ \Omega}
\]

(b) The emitter-follower circuit for PSpice simulation is shown in Fig. 8.35.
The PSpice plots, which are shown in Fig. 8.36 for $v_s = 1$ V, give $v_o = 987$ mV and $R_{in} = V_{s(rms)}/I_{s(rms)} = 41.98$ kΩ (expected value is 33.6 kΩ), and the voltage gain is $A_v = v_o/v_s = 0.987$ (expected value is 0.9966). If we run the simulation with a very large value of $R_L$, tending to infinity (say, $R_L = 10$ GΩ), the output voltage will be the maximum $v_{o(max)}$. Then if we connect the normal load (say, $R_L = 5$ kΩ) and run the simulation, the output voltage should drop because of the current flow through the output resistance $R_o$ of the amplifier. PSpice simulation gives $v_o = 987$ mV and $v_{o(max)} = 990$ mV for $R_L = 10$ GΩ. Thus, $R_o$ can be found from

$$\Delta v_o = v_{o(max)} - v_o = R_o I_L = \frac{R_o v_o}{R_L}$$

which gives

$$R_o = \frac{R_L(v_{o(max)} - v_o)}{v_o} = \frac{5 \text{ kΩ} \times (990 - 987) \text{ mV}}{987 \text{ mV}} = 15 \Omega$$
NOTE: This example simulates the steps that would normally be used to measure the output resistance of an amplifier in the laboratory. As expected, the simulated results differ from the design values, and the design calculations should be modified. Even if we modify the design to satisfy the design specifications, the results can be expected to differ from those that would be obtained in the laboratory, although not significantly.

If we ran a PSpice simulation of the linear circuit shown in Fig. 8.34(c), the results would be closer to the expected values, but they would not take into account the nonlinear behavior of the transistor. The results of .TF analysis are as follows: (The values obtained from hand calculations are shown in parentheses to the right.)


<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>INPUT RESISTANCE AT VS=3.310E+04</strong></td>
<td>33.1 kΩ</td>
</tr>
<tr>
<td><strong>OUTPUT RESISTANCE AT V(4)=4.981E+00</strong></td>
<td>4.981 Ω</td>
</tr>
</tbody>
</table>

*** SMALL-SIGNAL CHARACTERISTICS

A common-collector amplifier is generally known as an emitter follower because the emitter voltage follows the voltage at the base terminal with ideally a unity voltage gain. This type of amplifier has low output resistance and a high input resistance. It is commonly used as a buffer stage between a load and the source.

The voltage gain of a resistive-biased emitter follower is close to unity, and an active-biased amplifier with a sinking current source can yield almost unity voltage gain with a very low output resistance.

8.10 Common-Base Amplifiers

In a common-base (CB) amplifier, the input signal is applied to the emitter terminal. That is, the base is common to both the input and the output terminal. Such an amplifier has a low input resistance. There is no change in the phase shift, however, between the input and output signals; that is, the output signal is in phase with the input signal. Similar to other amplifier types, we could use either active or resistive biasing. A CB amplifier with resistive biasing is shown in Fig. 8.37(a). The configuration of this circuit may appear different from that of the common emitter, but it is not. The circuit can be redrawn as shown in Fig. 8.37(b), where the input signal $v_s$ is connected to the emitter terminal via a coupling capacitor $C_1$. Thus, the biasing of this circuit is identical to that of the common emitter, and the technique discussed earlier can be applied to design the DC-biasing circuit.

Let us assume that $C_1$, $C_2$, and output resistance $r_o$ of $Q_1$ are very large, tending to infinity. That is, $C_1 = C_2 = \infty$, and $r_o = \infty$. The small-signal AC equivalent circuit of the amplifier in Fig. 8.37(a) is shown in Fig. 8.38(a), which can be simplified to Fig. 8.38(b). $R_L$ is considered an external element and is not included in Fig. 8.38(a). This amplifier can be represented by the equivalent voltage and transconductance amplifiers shown in Fig. 8.38[(c) and (d)], respectively.
8.10.1 Input Resistance $R_i$

Since $v_e = -i_b(r_n + R_B)$ in Fig. 8.38(b), the voltage $v_e$ can be related to the base–emitter voltage $v_{be}$ by

$$v_{be} = -\frac{r_n v_e}{r_n + R_B}$$  \hspace{1cm} (8.105)

where $R_B = R_1 \parallel R_2$. Using KCL at the emitter junction E of Fig. 8.38(b) and substituting for $v_{be}$, we get

$$i_e \approx -i_b - g_m v_{be} = \frac{v_e}{r_n + R_B} - g_m v_{be} = \frac{v_e}{r_n + R_B} + \frac{g_m r_n v_e}{r_n + R_B} \approx \frac{1 + \beta_f}{r_n + R_B} v_e$$
which gives the resistance $R_x$ at the emitter terminal as

$$R_x = \frac{v_e}{i_e} = \frac{r_\pi + R_B}{1 + g_m r_\pi} = \frac{r_\pi + R_B}{1 + \beta_f}$$

Input resistance $R_i$, which is the parallel combination of $R_E$ and $R_x$, is

$$R_i = \frac{v_e}{i_s} = \frac{R_E}{R_x} = \frac{r_\pi + R_B}{1 + \beta_f}$$  \hspace{1cm} (8.106)

Since $(r_\pi + R_B)/(1 + \beta_f)$ will have a small value, the input resistance $R_i$ is usually low. This is the major disadvantage of a CB amplifier. Thus, $R_{in} = R_i + R_s$.

### 8.10.2 No-Load Voltage Gain $A_{vo}$

The no-load output voltage from Fig. 8.38(b) $v_o$ is

$$v_o = -i_c R_C = -R_C g_m v_{be}$$

Substituting $v_{be}$ from Eq. (8.105), we get

$$v_o = R_C g_m \frac{r_\pi}{r_\pi + R_B} v_e$$

which gives the no-load voltage gain $A_{vo}$ as

$$A_{vo} = \frac{v_o}{v_e} = \frac{g_m r_\pi R_C}{r_\pi + R_B} = \frac{\beta_f R_C}{r_\pi + R_B}$$  \hspace{1cm} (8.107)

The no-load voltage gain $A_{vo}$ can be increased by making $R_B = 0$ if a bypass capacitor $C_B$ is connected between the base and the ground, as shown in Fig. 8.39. Equation (8.107) gives the maximum no-load voltage gain as

$$A_{vo(max)} = \frac{\beta_f R_C}{r_\pi} = g_m R_C$$  \hspace{1cm} (8.108)

### 8.10.3 Output Resistance $R_o$

Assuming that the output resistance of the transistor is very large, tending to infinity (i.e., $r_{o1} \approx \infty$), the output resistance $R_o$ can be found by inspection of Fig. 8.38(b) to be $R_o \approx R_C$.

![Common-base configuration with bypass capacitor $C_B$](image)
Because of the low input resistance, the overall voltage gain $A_v$ is reduced considerably. The collector-base capacitor $C_C$, which has a low capacitance, appears between the output and base terminals, not between the output and input terminals. If a capacitor is connected between the input and output terminals of an amplifier, the capacitance is subject to Miller’s multiplication effect, as discussed in Sec. 2.6. In a CB amplifier, $C_C$ is not subject to Miller’s multiplication effect, and thus CB amplifiers are used for high-frequency applications.

**EXAMPLE 8.7**

**Finding the parameters of a common-base amplifier**  The CB amplifier of Fig. 8.37(a) has $R_1 = 13.16 \, k\Omega$, $R_2 = 8.06 \, k\Omega$, $R_E = 495 \, \Omega$, $R_C = 500 \, \Omega$, $R_L = 5 \, k\Omega$, and $R_s = 250 \, \Omega$. The parameters of the transistor are $r_\pi = 258 \, \Omega$, $\beta_f = 100$, and $r_{o1} = \infty$. Assume that $C_1 = C_2 = \infty$.

(a) Calculate the input resistance $R_{in} (=v_s/i_s)$, the no-load voltage gain $A_{vo} (=v_o/v_e)$, the output resistance $R_o$, the overall voltage gain $A_v (=v_L/v_s)$, and the maximum permissible voltage gain $A_{vo(max)}$.

(b) Use PSpice/SPICE to verify your results in part (a).

**SOLUTION**

(a) We first find $R_B$ and $R_x$.

$$R_B = R_1 \parallel R_2 = 13.16 \, k \parallel 8.06 \, k = 5 \, k\Omega$$

$$R_x = \frac{r_\pi + R_B}{1 + \beta_f} = \frac{258 + 5 \, k}{101} = 52.1 \, \Omega$$

From Eq. (8.106),

$$R_i = R_E \parallel R_x = 495 \parallel 52.1 = 47.1 \, \Omega$$

Thus

$$R_{in} = R_i + R_s = 47.1 + 250 = 297.1 \, \Omega$$

From Eq. (8.107),

$$A_{vo} = \frac{\beta_f R_C}{r_\pi + R_B} = \frac{100 \times 500}{258 + 5 \, k} = 9.5$$

and

$$R_o = R_C = 500 \, \Omega$$

Using Fig. 8.38(c), we find that the overall voltage gain $A_v$ is

$$A_v = \frac{v_L}{v_s} = \frac{A_{vo} R_i R_L}{(R_i + R_s)(R_L + R_o)} = \frac{9.5 \times 47.1 \times 5 \, k}{(47.1 + 250)(5 \, k + 500)} = 1.37$$

The maximum permissible voltage gain from Eq. (8.108) is given by

$$A_{vo(max)} = \frac{\beta_f R_C}{r_\pi} = \frac{100 \times 500}{258} = 194$$

(b) The emitter-follower circuit for PSpice simulation is shown in Fig. 8.40.
The PSpice plots, which are shown in Fig. 8.41 for $v_s = 1\, \text{mV}$, give $v_o = 1.44\, \text{mV}$ and $R_{\text{in}} = V_{\text{rms}}/I_{\text{rms}} = 424\, \Omega$ (expected value is 297\,\Omega), and the voltage gain is $A_v = v_o/v_s = 1.43$ (expected value is 1.37). Thus, the results are close to the expected values.

**KEY POINTS OF SECTION 8.10**

- In a CB amplifier, the input signal is applied to the emitter terminal. That is, the base is common to both the input and the output terminal. Similar to other amplifier types, we could use either active or resistive biasing.
- This type of amplifier has a low input resistance. There is no phase shift, however, between the input and output signals; that is, the output signal is in phase with the input signal.
- The expressions for BJT amplifiers in terms of their parameters, the input resistance $R_i$, the output resistance $R_o$, and the no-load voltage gain $A_{vo}$, are summarized in Table 8.2.

**TABLE 8.2** Summary of expressions for BJT amplifiers

<table>
<thead>
<tr>
<th></th>
<th>CE Amplifier (Fig. 8.29)</th>
<th>Emitter Follower (Fig. 8.34[a])</th>
<th>CB Amplifier (Fig. 8.37[a])</th>
<th>CE Amplifier with Active Load (Fig. 8.24[a])</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_i$ ($\Omega$)</td>
<td>$R_B \parallel [r_\pi + (1 + \beta_f)R_E]$</td>
<td>$R_B \parallel [r_\pi + (1 + \beta_f)(r_{o1} \parallel R_E \parallel R_L)]$</td>
<td>$R_E \parallel \frac{r_\pi + R_B}{\beta_f}$</td>
<td>$r_\pi \parallel R_B$</td>
</tr>
<tr>
<td>$R_o$ ($\Omega$)</td>
<td>$R_C$</td>
<td>$R_E \parallel r_{o1} \parallel \frac{r_\pi + (R_s \parallel R_B)}{1 + \beta_f}$</td>
<td>$R_C$</td>
<td>$r_{o2} \parallel r_{o1}$</td>
</tr>
<tr>
<td>$A_{vo}$ (V/V)</td>
<td>$\frac{-\beta_f R_C}{r_\pi + (1 + \beta_f)R_E}$</td>
<td>$\frac{(1 + \beta_f)(R_E \parallel r_{o1})}{r_\pi + (1 + \beta_f)(R_E \parallel r_{o1})}$</td>
<td>$\frac{\beta_f R_C}{r_\pi + R_B}$</td>
<td>$-g_{m1}(r_{o2} \parallel r_{o1})$</td>
</tr>
</tbody>
</table>
8.11 Multistage Amplifiers

The design requirements of amplifiers normally specify an overall high voltage gain, a high input resistance, and a low output resistance. A single-transistor amplifier rarely satisfies these design requirements, and multistages are often used to satisfy the design specifications. To achieve the design specifications, multiple transistor stages are connected in such a way that the output of one stage is the input to the next stage and so on. The most common types of arrangements are (1) capacitor-coupled cascaded, (2) direct coupled, and (3) cascoded.

8.11.1 Capacitor-Coupled Cascaded Amplifiers

In a capacitor-coupled amplifier, the output of one stage is connected to the input of the next stage via a capacitor as shown in Fig. 8.42(a). The first stage is a common-emitter amplifier that is designed to offer a high input resistance \( R_i \). The emitter follower in the third stage satisfies the requirement of a low output resistance \( R_o \). The second stage is common-emitter amplifier to meet the overall voltage gain requirement \( A_{vo} \). If \( A_{vo1} \) and \( A_{vo3} \) are the voltage gains of first and third stages, respectively, then the required gain for the second stage is \( A_{vo2} = A_{vo}/(A_{vo1} \times A_{vo3}) \). The DC-biasing point of each stage can be determined independently for each stage because the coupling capacitor provides DC isolation between the two stages. The biasing collector current should be low to reduce the power drain from the DC voltage source.

Each stage can be represented by its parameters \( R_1, R_o, \) and \( A_{vo} \) as shown in Fig. 8.42(b). The output resistance of a stage acts as the source resistance of the following stage, and the input resistance of

![A three-stage capacitor-coupled cascaded amplifier](image-url)

**FIGURE 8.42** A three-stage capacitor-coupled cascaded amplifier
a stage is the load resistance of the preceding stage. Thus, there will be a loading effect due to the interaction between stages, and the effective voltage gain will be reduced (see Sec. 2.4). When designing an amplifier, keep in mind the gain reduction due to loading effect and start the design with a voltage gain higher than \( A_{vo} \) to satisfy the design requirement.

### 8.11.2 Direct-Coupled Amplifiers

In *direct-coupled amplifiers*, the output of one stage is directly connected to the input of the next stage. We can make the amplifier in Fig. 8.42(a) a direct-coupled amplifier if we remove the coupling capacitors \( C_1 \) and \( C_2 \) and connect the following stage directly to the preceding stage. In this case, we can also remove the biasing resistances \( R_{21}, R_{22}, \) and \( R_{31} \).

### 8.11.3 Cascoded Amplifiers

The effective output resistance of a transistor can be increased by connecting two transistors in a configuration commonly referred to as a *cascoded amplifier*. The input signal is applied to one transistor whose output is the input to the other transistor, as shown in Fig. 8.43(a). The input signal is applied to the common-emitter amplifier, and the output is obtained at the collector of the common-base amplifier of transistor \( Q_2 \).

#### DC Biasing

The DC equivalent circuit for determining the DC-operating point of the transistors is shown in Fig. 8.43(b). The analysis can be simplified by assuming \( \beta_{F1} = \beta_{F2} = \beta_F >> 1 \). That is, the base current of a transistor is negligible, and the base currents can be ignored in comparison to the currents through the biasing resistances \( I_{B1} \ll I_3 \) and \( I_{B2} \ll I_2 \). Therefore, we can find the DC biasing base voltages as given by

\[
V_{B1} = \frac{R_{B1}}{R_{B1} + R_{B2} + R_{B3}} V_{CC} \quad (8.109)
\]

\[
V_{B2} = \frac{R_{B1} + R_{B2}}{R_{B1} + R_{B2} + R_{B3}} V_{CC} \quad (8.110)
\]

which can be used to find the collector and emitter currents:

\[
I_{E1} \approx I_{C1} = I_{E2} \approx I_{C2} = \frac{V_{B1} - V_{BE}}{R_E} \quad (8.111)
\]

The C-E voltages are given by

\[
V_{CE1} = V_{B2} - V_{BE2} - (V_{B1} - V_{BE1}) = V_{B2} - V_{B1} \quad \text{(for } V_{BE1} = V_{BE2} \text{)} \quad (8.112)
\]

\[
V_{CE2} = V_{CC} - R_C I_{C2} - (V_{B2} - V_{BE2}) \quad (8.113)
\]

#### Small-Signal Voltage Gain

Once we have found the DC-biasing values, we can find the small-signal mode parameters \( r_{\pi}, r_o, \) and \( g_m \) for both transistors. The small-signal equivalent circuit is shown in Fig. 8.43(c). The small-signal input resistance is given by

\[
R_i = \frac{V_s}{I_s} = R_B \parallel r_{\pi1} \quad (8.114)
\]
Since \( i_{c2} = g_{m2}v\text{be}_2 \), \( i_{c2} = i_{c2}/\alpha_F \), and \( v_{o1} = -v\text{be}_2 \), equating the currents \( i_{c1} = i_{c2} \) gives

\[
g_{m1}v\text{be}_1 = i_{c2} = \frac{g_{m2}v\text{be}_2}{\alpha_F} = -\frac{g_{m2}v_{o1}}{\alpha_F}
\]

which gives the voltage gain of the first stage as

\[
A_{vo1} = \frac{v_{o1}}{v\text{be}_1} = -\alpha_F \quad \text{(for} \ g_{m1} = g_{m2})
\]

(8.115)

Since the collector current \( i_{c2} = \alpha_F i_{c2} \), we can find the output voltage as

\[
v_o = -i_{c2}R_C = -R_C\alpha_F i_{c2} = -R_C\alpha_F i_{c1} = -R_C\alpha_F g_{m1}v\text{be}_1
\]

which gives the small-signal overall voltage gain \( A_{vo} \) as

\[
A_{vo} = \frac{v_o}{v_{be1}} = -\alpha_F g_{m1}R_C
\]

(8.116)
Therefore, the voltage gain of the second stage is

\[ A_{v_o2} = \frac{A_{v_o1}}{A_{v_o2}} = -\frac{\alpha_F g_m R_C}{\alpha_F} = -g_m R_C \quad (8.117) \]

**Small-Signal Output Resistance**

The equivalent circuit for determining the output resistance is shown in Fig. 8.43(d). Since \( v_{be1} = 0 \), the current source \( g_{m1} v_{be1} = 0 \) is open-circuited. Using KVL, we can write

\[ v_x = r_{o2}(i_x - g_{m2}v_{be2}) + v_{o1} \]

After substituting \( v_{o1} = i_x(r_{o1} \parallel r_{\pi2}) \) and \( v_{be2} = -v_{vo1} \), we can get the output resistance, after simplification, as given by

\[ R_o = \frac{v_x}{i_x} = (r_{o1} \parallel r_{\pi2}) + r_{o2}[1 + g_{m2}(r_{o1} \parallel r_{\pi2})] \]

\[ \approx r_{o2}(1 + g_{m2}r_{\pi2}) = r_{o2}(1 + \beta_F) \quad (for \ r_{o1} \gg r_{\pi2}) \quad (8.118) \]

which is the same as the output resistance \( R_o \) of the Widlar current source described by Eq. (9.30) in Sec. 9.4 if we substitute \( R_2 \) with \( r_{o1} \).

---

**KEY POINTS OF SECTION 8.11**

- The design requirements of amplifiers normally specify an overall high voltage gain, a high input resistance, and a low output resistance. A single-transistor amplifier rarely satisfies these design requirements and multistages are often used to satisfy the design specifications.
- The output of one stage in a capacitively coupled amplifier is connected to the input of the next stage via a capacitor. In direct-coupled amplifiers, the output of one stage is directly connected to the input of the next stage.
- The effective output resistance of a transistor can be increased by connecting two transistors in a configuration commonly referred to as a cascoded amplifier. The input signal is applied to one transistor whose output is the input to the other transistor.

---

**8.12 The Darlington Pair Transistor**

In all types of biasing circuits, the input resistance, the output resistance, and the voltage gain are dependent on the transistor current gain \( \beta_F \). A compound transistor configuration, known as a *Darlington pair*, is often used to give a much higher input resistance and a much lower input bias current than a single transistor would provide. It consists of two cascaded transistors as shown in Fig. 8.44(a); the internal structure is shown in Fig. 8.44(b). A Darlington pair can be represented as an equivalent single transistor as shown in Fig. 8.45.

The effective B-E voltage is

\[ V_{BE} = V_{BE1} + V_{BE2} = V_T \ln \left( \frac{I_{C1}}{I_{S1}} \right) + V_T \ln \left( \frac{I_{C2}}{I_{S2}} \right) = V_T \ln \left( \frac{I_{C1}I_{C2}}{I_{S1}I_{S2}} \right) \quad (8.119) \]
Since \( I_C = I_{C2} = \beta_F I_{C1} \), Eq. (8.119) becomes

\[
V_{BE} = V_T \ln \left( \frac{I_C^2}{\beta_F I_{S1} I_{S2}} \right)
\]  

(8.120)

Solving for \( I_C \), we get

\[
I_C = \sqrt{\beta_F I_{S1} I_{S2}} \exp \left( \frac{V_{BE}}{2V_T} \right) = I_S \exp \left( \frac{V_{BE}}{V_T} \right)
\]  

(8.121)

where \( I_S = \sqrt{\beta_F I_{S1} I_{S2}} \) = effective saturation current and \( V_T = 2V_T \) = effective thermal voltage.

The collector current \( I_C \) can be related to \( I_{B1} \) by

\[
I_C = I_{C2} = \beta_F I_{B2} = (1 + \beta_F)I_{C1} = \beta_F(1 + \beta_F)I_{B1} \approx \beta_F^2 I_{B1}
\]  

(8.122)

Thus, the effective input resistance of the compound pair is given by

\[
r'_\pi = \frac{V_T}{I_{B1}} = \beta_F^2 \frac{2V_T}{I_C}
\]  

(8.123)

which will be \( 2\beta_F \) times greater than that for a single device. For a single equivalent transistor \( Q_T \), \( r'_\pi = 2\beta_F r_\pi \). Thus, if \( I_C = 200 \, \mu A \), \( \beta_F = 100 \), and \( V_T = 26 \, mV \),

\[
r_\pi = \frac{\beta_F V_T}{I_C} = \frac{100 \times 26 \, mV}{200 \, \mu A} = 13 \, k\Omega
\]

FIGURE 8.44  Darlington pair transistor

FIGURE 8.45  Darlington pair single-transistor equivalent
for a single transistor and
\[ r'_\pi = \frac{100^2 \times 2 \times 26 \text{ mV}}{200 \mu\text{A}} = 2.6 \text{ M\Omega} \]

for a Darlington pair. The input offset voltage \( V_{\text{OS}} \), however, will increase (generally \( \sqrt{2} \) times) as a result of the increase in the effective thermal voltage.

**EXAMPLE 8.8**

**Finding the effective parameters of a Darlington pair**  The Darlington pair shown in Fig. 8.46 is biased in such a way that the collector biasing current \( I_{C2} \) of \( Q_2 \) is 1 mA. The current gains of the two transistors are the same, \( \beta_{F1} = \beta_{F2} = \beta_F = 100 \), and the Early voltage is \( V_A = 75 \text{ V} \). Calculate (a) the effective input resistance \( r_\pi \), (b) the effective transconductance \( g_m \), (c) the effective current gain \( \beta_{F(\text{eff})} \), and (d) the effective output resistance \( r_o \).

**SOLUTION**

The two transistors in Fig. 8.46 can be replaced by an equivalent transistor, shown in Fig. 8.47(a), which may be regarded as the subcircuit of the two transistors and can be modeled by the circuit in Fig. 8.47(b). Replacing each transistor by its model gives the small-signal equivalent circuit of a Darlington pair shown in Fig. 8.47(c).
Let us assume that $r_o \approx \infty$. The two collector and base currents of the transistors will be different. According to Eq. (8.14), $r_\pi$ depends inversely on the collector current. Thus, the small-signal input resistances $r_{\pi 1}$ and $r_{\pi 2}$ of the transistors will be different. Assuming we have $V_T = 25.8 \text{ mV}$,

$$g_{m2} = \frac{I_{C2}}{V_T} = \frac{1 \text{ mA}}{25.8 \text{ mV}} = 38.76 \text{ mA/V}$$

$$r_{\pi 2} = \frac{\beta_F}{g_{m2}} = \frac{\beta_F V_T}{I_{C2}} = \frac{100 \times 25.8 \text{ mV}}{1 \text{ mA}} = 2.58 \text{ k\Omega}$$

$$I_{R2} = I_{E1} = \frac{I_{C2}}{\beta_F} = \frac{1 \text{ mA}}{100} = 10 \text{ \mu A}$$

$$r_{o2} = \frac{V_A}{I_{C2}} = \frac{75 \text{ V}}{1 \text{ mA}} = 75 \text{ k\Omega}$$

$$I_{C1} = \frac{I_{R2} \beta_F}{1 + \beta_F} = \frac{10 \text{ \mu A} \times 100}{1 + 100} = 9.9 \text{ \mu A}$$

$$g_{m1} = \frac{I_{C1}}{V_T} = \frac{9.9 \text{ \mu A}}{25.8 \text{ mV}} = 383.7 \text{ \mu A/V}$$

$$r_{\pi 1} = \frac{\beta_F}{g_{m1}} = \frac{100}{383.7 \text{ \mu A}} = 260.6 \text{ k\Omega}$$

$$r_{o1} = \frac{V_A}{I_{C1}} = \frac{75 \text{ V}}{9.9 \text{ \mu A}} = 7.6 \text{ M\Omega}$$

(a) Figure 8.47(c) is similar to Fig. 8.28(b). Thus, Eq. (8.85) can be applied to Fig. 8.47(c) if $r_\pi$ and $R_E$ are replaced by $r_{\pi 1}$ and $r_{\pi 2}$, respectively. Replacing $R_{E1}$ in Eq. (8.85) with $r_{\pi 2}$ (in parallel with $r_{o1}$) gives the input resistance

$$r_\pi = r_{\pi 1} + (1 + \beta_F)(r_{\pi 2} \parallel r_{o1})$$

$$= 260.6 \text{ k\Omega} + (1 + 100) \times (2.58 \text{ k\Omega} \parallel 7.6 \text{ M\Omega}) = 521.1 \text{ k\Omega}$$

(b) The voltage $v_{be2}$ in Fig. 8.47(c) is identical to $v_o$ in Fig. 8.34(c) provided $r_{\pi 2}$ is substituted for $R_E$. Thus, $v_{be2}$ can be related to the input voltage $v_b$ by Eq. (8.102):

$$v_{be2} = \frac{(1 + \beta_{F1})r_{\pi 2}}{r_{\pi 1} + (1 + \beta_{F1})r_{\pi 2}} v_b$$

The collector current $i_c$ of the second transistor can be found from

$$i_c = g_{m2} v_{be2} = g_{m2} \frac{(1 + \beta_{F1})r_{\pi 2}}{r_{\pi 1} + (1 + \beta_{F1})r_{\pi 2}} v_b$$

which gives the equivalent transconductance $g_m$ as

$$g_m = \frac{i_c}{v_b} = \frac{g_{m2} (1 + \beta_{F1})r_{\pi 2}}{r_{\pi 1} + (1 + \beta_{F1})r_{\pi 2}} = \frac{g_{m2}}{1 + r_{\pi 1}/[(1 + \beta_{F1})r_{\pi 2}]}$$

(8.126)
Since the emitter current $I_{E1}$ of Q1 is equal to the base current $I_{B2}$ of Q2, the biasing base current $I_{B2}$ of Q2 will be related to the biasing base current $I_{B1}$ of Q1 by

$$I_{B2} = I_{E1} = (1 + \beta_{F1})I_{B1}$$

According to Eq. (8.21), $r_{\pi}$ is inversely proportional to the biasing base current $I_{B}$. Thus, $r_{\pi1}$ and $r_{\pi2}$ will be related by $r_{\pi1} = (1 + \beta_{F1})r_{\pi2}$. Therefore, Eq. (8.126) can be simplified to

$$g_m = \frac{g_{m2}}{2}$$

$$= \frac{38.76}{2} = 19.38 \text{ mA/V}$$

(c) The collector current $i_{c2}$ of Q2 can also be written as

$$i_{c} = \beta_{F2}i_{b2}$$

Since $i_{b2} = (1 + \beta_{F1})i_{b1},$

$$i_{c} = \beta_{F2}i_{b2} = \beta_{F2}(1 + \beta_{F1})i_{b1}$$

Thus, the effective current gain $\beta_{F(\text{eff})}$, which is the ratio of $i_{c}$ to $i_{b}$, is

$$\beta_{F(\text{eff})} = \frac{i_{c}}{i_{b1}} = \beta_{F2}(1 + \beta_{F1})$$

$$= \beta_{F2}(1 + \beta_{F1}) = 100 \times (1 + 100) = 10,100$$

(d) The effective output resistance $r_o$ is

$$r_o = r_{o2} = 75 \text{ k}\Omega$$

Thus, a model that can represent the two CC or CE transistors is shown in Fig. 8.47(b).

### KEY POINT OF SECTION 8.12

- A compound transistor configuration known as a Darlington pair, which increases the effective current gain $\beta_F$, is often used to give a much higher input resistance and a much lower input bias current than a single transistor would provide.

## 8.13 DC Level Shifting and Amplifier

In all the amplifiers discussed so far, we used coupling capacitors to superimpose the small AC signal on the DC biasing voltage at the base terminal of the transistors. These capacitors provide DC isolations of each stage from the previous or subsequent stage. The amplified AC signal is superimposed on the DC biasing voltage at the output terminal of the transistors. The coupling capacitors cannot be used in the design of amplifiers that only amplify DC signals. In some cases, it may be necessary to shift the
quiescent voltage of one stage before applying its output to the following stage. Level-shifting circuits can adjust the DC-bias levels between amplification stages. Level shifting is also required to make the output close to zero in the quiescent state with no input signal. The input resistance of the level-shifting stage should be high to prevent loading of the previous gain stage. Also, the output resistance should be low to effectively drive the subsequent stage.

### 8.13.1 Level-Shifting Methods

The emitter-follower configuration is normally used to level shift. The emitter follower has an inherent characteristic of level shifting by $V_{BE} \approx 0.7$ V such that the output voltage is $V_O = V_S - V_{BE}$. Thus, the idea is to create a voltage in the emitter terminal, and it can be accomplished by (1) a potential divider network, (2) a current source, and (3) a zener diode.

#### Potential Divider Level Shifting

This arrangement is shown in Fig. 8.48(a). The voltage shift is

$$v_O - v_S = -(V_{BE} + R_1 i_E)$$

which, after substituting for $i_E = (v_S - V_{BE} + V_{EE})/(R_1 + R_2)$ and simplifying, gives the output voltage as

$$v_O = \frac{R_2}{R_1 + R_2}(v_S - V_{BE}) - \frac{R_1 V_{EE}}{R_1 + R_2} \tag{8.130}$$

Although this circuit provides a voltage shift, it also attenuates the input signal by a factor $R_2/(R_1 + R_2)$.

![Potential divider shift](image1)

![Current source shift](image2)

![Zener shift](image3)

**FIGURE 8.48** Level shifting
**Current Source Level Shifting**

The attenuation of Fig. 8.48(a) can be prevented by replacing $R_2$ with a current source at a constant current $I_O$. This arrangement is shown in Fig. 8.48(b). The voltage shift is

$$v_O - v_S = -(V_{BE} + R_1I_O)$$

which gives the output voltage as

$$v_O = v_S - V_{BE} - R_1I_O \quad (8.131)$$

Since the current through $R_1$ is fixed, the voltage drop across it is also fixed. It is important to note that $v_O$ is independent of the negative DC supply voltage $-V_{EE}$.

**Zener Level Shifting**

Resistance $R_1$ in Fig. 8.48(a) is replaced by a zener diode with zener voltage $V_z$. This arrangement is shown in Fig. 8.48(c). The voltage shift is

$$v_O - v_S = -(V_{BE} + V_z)$$

which gives the output voltage as

$$v_O = v_S - V_{BE} - V_z \quad (8.132)$$

Since the zener voltage $V_z$ is fixed, the voltage drop across it is also fixed.

**8.13.2 Level-Shifted DC Amplifier**

A BJT amplifier using level shifting is shown in Fig. 8.49(a) with four stages [6]. The first stage generates the reference current for the second stage, which acts as the reference current for the third stage. The fourth stage is the emitter follower. Assume that all transistors are matched devices and have equal parameters: the current gains $\beta_{F1} = \ldots = \beta_{F7} = \beta_F$, the Early voltages $V_{A1} = \ldots = V_{A7} = V_A$, and the B-E voltage $V_{BE1} = \ldots = V_{BE7} = V_{BE}$. To simplify the derivations, assume that $\beta_F \gg 1$ and the base currents of all transistors are small as compared to the collector and emitter currents, and these can be neglected.

**Current Mirror Source**

Using KVL in the B-E loops of $Q_1$, $Q_2$, and $Q_3$, $Q_4$, we can find the reference currents $I_1$, $I_2$, and $I_R$ as given by

$$I_1 = \frac{V_1 + V_{EE} - V_{BE1} - V_{BE2}}{R_A + R_B} \quad (8.133)$$

$$I_R = \frac{V_2 + V_{EE} - V_{BE4} - V_{BE5}}{R} \quad (8.134)$$

$$I_2 = I_R - I_{C3} \quad (8.135)$$
Using Eq. (8.58), we can find the biasing current sources:

\[ I_{C3} = \frac{I_1}{1 + 2/\beta_F} \approx I_1 \]  

(8.136)

\[ I_{C6} = \frac{I_2}{1 + 2/\beta_F} \approx I_2 \]  

(8.137)

\[ I_{E1} = I_{E2} \approx I_1 \quad I_{E3} \approx I_{C3} \quad I_{E4} = I_R \quad I_{E5} \approx I_2 \]
**DC Output Voltage**

For $I_{C6} \gg I_{B7}$ and $I_{C6} \approx I_2$, we can find the voltage at the base of the output transistor $Q_7$ as given by

$$V_x = V_{CC} - R_x(I_{C6} + I_{B7}) = V_{CC} - R_xI_6 = V_{CC} - R_xI_2$$

which, after substituting $I_2$ from Eq. (8.135), can be written as

$$V_x = V_{CC} - R_x(I_R - I_{C3}) = V_{CC} - R_x(I_R - I_1) \quad (8.138)$$

which, after substituting $I_1$ from Eq. (8.133) and substituting $I_R$ from Eq. (8.134), gives

$$V_x = V_{CC} - R_x\left(\frac{V_2 + V_{EE} - V_{BE4} - V_{BE5}}{R} + R_x\left(\frac{V_1 + V_{EE} - V_{BE1} - V_{BE2}}{R_A + R_B}\right)\right) \quad (8.139)$$

We can simplify Eq. (8.139) if we choose resistors such that $R_A + R_B = R = R_x$ and the B-E voltage drops are matched. That is, Eq. (8.139) is simplified to

$$V_x = V_{CC} - V_2 + V_1 \quad (8.140)$$

Therefore, the voltage $V_x$ at the emitter of transistor $Q_7$ is shifted by $V_{BE7}$, and the output voltage becomes

$$V_O = V_x - V_{BE7} = V_{CC} + (V_1 - V_2) - V_{BE7} \quad (8.141)$$

If the base of transistor $Q_4$ is connected to $V_{CC}$ as shown in Fig. 8.49, then $V_2 = V_{CC}$. We get the output voltage as given by

$$V_O = V_{CC} - V_{CC} + V_1 - V_{BE7} = V_1 - V_{BE7} \quad (8.142)$$

Therefore, there is a DC offset voltage due to $V_{BE7}$, which can be canceled by decreasing the current $I_{C6}$ through $R_x$ by an amount

$$\Delta I_{C6} = \frac{V_{BE7}}{R_x} \quad (8.143)$$

This can be accomplished by placing a compensating resistor $R_y$ from the base of $Q_4$ to the negative supply ($-V_{EE}$). This will decrease the reference current $I_2$ by an amount

$$\Delta I_2 = \frac{V_{BE5}}{R_y} \quad (8.144)$$

Since the decrease in $I_{C6}$ should be equal to the decrease in $I_2$, we can find the relation between $R_x$ and $R_y$ as

$$\frac{V_{BE7}}{R_x} = \frac{V_{BE5}}{R_y} \quad (8.145)$$

Therefore, $I_2$ in Eq. (8.135) should be corrected as given by

$$I_2 = I_R - I_{C3} - \frac{V_{BE5}}{R_y} \quad (8.146)$$

Under these conditions, the output voltage in Eq. (8.142) becomes

$$V_O = V_1 \quad (8.147)$$

In integrated circuit design, the emitter of $Q_7$ can be scaled to produce a B-E junction drop identical to that of $Q_5$, and $R_y$ can be made equal to $R_x$ in order to obtain the condition in Eq. (8.145).
Small-Signal Voltage Gain

By replacing the transistors by its small-signal model, the small-signal equivalent circuit is shown in Fig. 8.49(b). The capacitor C_A and the batteries (V_{CC} and V_{EE}) are shorted to ground. Using KVL through the resistance R_B loop, we can find the small-signal reference current i_1 and its mirrored current i_{c3} as

\[ i_1 = i_{c3} = \frac{v_1}{r_{e1} + r_{e2} + R_B} \]  \hspace{1cm} (8.148)

Using KVL through the resistance R loop and the relation \( i_2 = i_R - i_{c3} \), we can find the relation for the small-signal reference current i_R as

\[ 0 = (r_{e4} + R + r_{e5})i_R - r_{es}i_{c3} = (r_{e4} + R + r_{e5})(i_{c3} + i_2) - r_{es}i_{c3} \]

which gives i_2 as

\[ i_2 = -\frac{r_{e4} + R}{r_{e4} + r_{e5} + R}i_{c3} \approx -i_{c3} \quad \text{for} \quad R >> r_{e4}, r_{e5} \]  \hspace{1cm} (8.149)

The small-signal current i_2 is mirrored to i_{c6}, which produces the voltage v_x at the base of transistor Q_7 as given by

\[ v_x = -i_{c6}R_x = i_{c3}R_x \]

Assuming a unity-gain emitter follower, we get the small-signal output voltage v_o as

\[ v_o = i_{c3}R_x \]

which, after substituting i_{c3} from Eq. (8.148), gives the output voltage v_o as

\[ v_o = i_{c3}R_x = \frac{R_x}{r_{e1} + r_{e2} + R_B}v_1 \]

which gives the no-load voltage gain A_{vo} as

\[ A_{vo} = \frac{v_o}{v_1} = \frac{R_x}{r_{e1} + r_{e2} + R_B} \]  \hspace{1cm} (8.150)

For \( R_x = R_A + R_B \) and \( R_B >> (r_{e1} + r_{e2}) \), Eq. (8.150) can be simplified to

\[ A_{vo} = \frac{R_A + R_B}{R_B} = 1 + \frac{R_A}{R_B} \]  \hspace{1cm} (8.151)

It is important to note that any increase in v_1 causes the current i_1 to increase, which is mirrored to i_{c3}, which in turn decreases i_2 by the same amount. Current i_{c6}, which is a mirror of i_2, causes the voltage v_x to increase by R_x i_{c6}. The transistors, which act as current mirrors and shift the voltage levels, do not produce any voltage amplification. The voltage gain described by Eq. (8.151) is accomplished by shunting R_A by the capacitor C_A for AC signals. The maximum voltage gain can be obtained by shunting both R_A and R_B by capacitor C_A. For making \( r_{e1} = r_{e2} \) and \( R_B = 0 \), Eq. (8.150) gives the maximum voltage gain as

\[ A_{vo(max)} = \frac{R_x}{2r_{e1}} = \frac{R_xI_1}{2V_T} \]  \hspace{1cm} (8.152)
where the product $R_x I_1$ is the DC voltage drop across $R_x$ and should not exceed half of the DC supply voltage $V_{CC}/2$ in order to minimize signal distortion. Therefore, Eq. (8.152) gives the maximum permissible voltage gain as

$$A_{vo(perm)} = \frac{R_x}{2r_{e1}} = \frac{R_x V_{CC}}{4V_T}$$  \hspace{1cm} (8.153)

The design of this amplifier is very simple. It requires finding only the values of $R_A$, $R_B$, $R_y$, and $R_x = R = R_A + R_B$ to obtain a specific voltage gain $A_{vo}$.

### KEY POINTS OF SECTION 8.13

- Level-shifting circuits, which can adjust the DC bias levels between amplification stages, are often used to make the output close to zero in the quiescent state with no input signal.
- A level-shifted amplifier can also provide a voltage gain. Level shifting can be accomplished by a potential divider, current source, and zener diode.

### 8.14 Frequency Model and Response of Bipolar Junction Transistors

In deriving the small-signal BJT model in Fig. 8.12(a), we assumed that the B-E and C-E junctions had no capacitances. A model that includes these capacitances will represent the frequency characteristic of BJTs. In this section, we develop the frequency and PSpice/SPICE models and then determine the frequency characteristic of BJTs.

#### 8.14.1 High-Frequency Model

An accurate small-signal high-frequency $\pi$ model that includes capacitances and parasitic resistances is shown in Fig. 8.50(a). The resistances $r_b$, $r_c$, and $r_e$ are the series parasitic resistances in the base, collector, and emitter contacts, respectively. The typical values of these resistances are $r_b = 50 \, \Omega$ to $500 \, \Omega$, $r_c = 20 \, \Omega$ to $500 \, \Omega$, and $r_e = 1 \, \Omega$ to $3 \, \Omega$. The value of $r_{b1} = 10 \beta r_o$ is very large, typically $10 \, \text{M}\Omega$.

A change in the input voltage $v_{be}$ will cause a change in the total minority carrier charge $q_e$ in the base. Because of charge-neutrality requirements, there will be an equal amount of change in the total majority carrier charge $q_h$ in the base. Because of the change in the charge, there will be a capacitance involved. This capacitance, known as the base-charging capacitance, is defined by

$$C_b = \frac{q_h}{v_{be}}$$  \hspace{1cm} (8.154)

(Refer to the diffusion capacitance $C_d$ for the diode because the B-E junction is similar to the diode junction.) A certain amount of time, called the base transit time, is required for a minority carrier to
cross the base. If $q_e$ is the charge in transit and $I_C$ is the collector current, the forward base transit time $\tau_F$ is defined by

$$\tau_F = \frac{q_e}{I_C}$$

Thus, $\tau_F$ is the average time each carrier spends crossing the base. The change in minority charge for a change in collector current is

$$\Delta q_e = \tau_F \Delta I_C$$

The change in minority charge must be equal to the change in majority charge $q_h$. That is, $\Delta q_e = \Delta q_h$. Therefore,

$$\Delta q_h = \Delta q_e = \tau_F \Delta I_C$$

In terms of small-signal quantities, we can write

$$q_h = q_e = \tau_F i_C$$

Substituting $q_h$ from Eq. (8.156) into Eq. (8.154) yields

$$C_b = \frac{q_h}{v_{be}} = \frac{\tau_F i_C}{v_{be}}$$

Substituting $i_C = g_{m} v_{be}$ from Eq. (8.42) into Eq. (8.157) yields

$$C_b = \tau_F g_m = \tau_F \frac{I_C}{V_T}$$

Thus, $C_b$ is proportional to the collector biasing current $I_C$.

In addition to the base-charging capacitance, there will be a B-E depletion capacitance $C_{je}$, which is defined by [7]

$$C_{je} = \frac{C_{je0}}{[1 - V_{BE}/V_{je}]^{1/3}}$$

where $C_{je0}$ is the value of $C_{je}$ for $V_{BE} = 0$ and typically is in the range of 0.2 pF to 1 pF.
$V_{je}$, which is the built-in potential across the junction with zero applied voltage, can be shown in Eq. 6.7 to be \(1\)

\[
V_{je} = V_T \ln \left( \frac{N_a N_d}{n_i^2} \right)  
\]

(8.160)

where $N_a$ is the doping density of $p$-type material in atoms per cubic centimeter, $N_d$ is the doping density of $n$-type material in atoms per cubic centimeter, and $n_i$ is the intrinsic carrier concentration in a pure sample of semiconductor. At 25°C (or 300 K), $n_i = 1.5 \times 10^{10}$ for silicon. For $N_a = 10^{15}$, $N_d = 10^{16}$, and $V_T = 25.8$ mV, the built-in potential is

\[
V_{je} = 25.8 \times 10^{-3} \times \ln \left( \frac{10^{15} \times 10^{16}}{(1.5^2 \times 10^{20})} \right) = 632.6 \text{ mV at 25°C}
\]

$C_{je}$ depends on the B-E voltage $V_{BE} (\leq V_{je})$ and the temperature because of $V_T$. Any increase in $V_{BE}$ will cause $C_{je}$ to increase. Thus, a reverse-biased B-E junction will exhibit a lower value of $C_{je}$.

The B-E input capacitance $C_\pi$ is the sum of $C_b$ and $C_{je}$. That is,

\[
C_\pi = C_b + C_{je}
\]

The C-B junction capacitance can be found approximately from

\[
C_\mu = \frac{C_{\mu 0}}{[1 + V_{CB}/V_{je}^{1/3}]^{1/3}}
\]

(8.161)

where $V_{je}$ and $C_{\mu 0}$ is the value of $C_\mu$ for $V_{CB} = 0$ and typically is in the range of 0.2 pF to 1 pF. A higher value of $V_{CB}$ will cause $C_\mu$ to decrease. Thus, a BJT operating as a switch will have a low value of $V_{CB}$, usually less than 0.7 V, and will exhibit a higher value of $C_\mu$ than a BJT operating as an amplifying device with $V_{CB} > 0.7$ V.

There is also a capacitance from the collector to the substrate (body) of the transistor. The substrate is usually connected to the ground. The collector-substrate capacitance can be found approximately from

\[
C_{cs} = \frac{C_{cs 0}}{[1 + V_{CS}/V_{js}^{1/3}]^{1/3}}
\]

(8.162)

where $V_{js}$ and $C_{cs 0}$ is the value of $C_{cs}$ for $V_{CS} = 0$ and typically is in the range of 1 pF to 3 pF.

Since $r_\mu$ is very large and the collector-substrate capacitance $C_{cs}$ is very small, their effects can be neglected. Although the C-B capacitance $C_\mu$ is small, it has a magnified influence on the frequency response as a result of the Miller effect. The simplified equivalent circuit, which neglects $r_\mu$, $C_{cs}$, $r_b$, $r_c$, and $r_e$, is shown in Fig. 8.50(b).

> **NOTE**  Manufacturers specify the common-emitter hybrid ($h$) parameters of a BJT rather than the $\pi$-model parameters. However, the $h$ parameters can be converted to $\pi$-model parameters (see Appendix D).

### 8.14.2 Small-Signal PSpice/SPICE Model

When simulating electronic circuits, PSpice/SPICE first calculates the DC-biasing point and generates the small-signal parameters for AC and transient analysis. The AC equivalent circuit generated by PSpice is shown in Fig. 8.51 [5].
8.14.3 Frequency Response of BJTs

Since the BJT model contains capacitances, the current gain $\beta_f$ will depend on the frequency. The dependency of $\beta_f$ on the frequency is normally given in the data sheet; the value of $C_\pi$ is not usually specified, but it can be determined from the expression for $\beta_f$ as a function of frequency. Let us apply a test current $i_b$ to the base of a BJT and short-circuit the collector terminal for AC signals. This arrangement is shown in Fig. 8.52(a), and the high-frequency AC equivalent circuit is shown in Fig. 8.52(b). The voltage at the base terminal is given by

$$V_{be}(s) = \left[ r_\pi \left/ \left( (C_\pi + C_\mu)s \right) \right. \right] I_b(s) = \frac{r_\pi}{1 + r_\pi(C_\pi + C_\mu)s} I_b(s)$$  \hspace{0.5cm} (8.163)

Since $r_o$ is very large and $C_\mu$ is very small, the current through them will be very small. That is,

$$I_c(s) = (g_m - sC_\mu) V_{be}(s)$$  \hspace{0.5cm} (8.164)

Substituting $V_{be}(s)$ from Eq. (8.163), we get

$$I_c(s) = \frac{(g_m - sC_\mu)r_\pi}{1 + r_\pi(C_\pi + C_\mu)s} I_b(s)$$

At the frequencies at which the model in Fig. 8.52(b) is valid, $g_m >> \omega C_\mu$, and we get the current gain $\beta_f(j\omega)$ in the frequency domain (for $s = j\omega$) as

$$\beta_f(j\omega) = \frac{I_c(j\omega)}{I_b(j\omega)} = \frac{r_\pi g_m}{1 + r_\pi(C_\pi + C_\mu)j\omega} = \frac{\beta_i}{1 + \beta_f(C_\pi + C_\mu)j\omega/g_m}$$  \hspace{0.5cm} (8.165)
which indicates that the current gain will fall as the frequency increases, at a slope of \(-20 \text{ dB/decade}\). This relationship is shown in Fig. 8.52(c) with a 3-dB frequency given by

\[ \omega = \omega_T = \frac{g_m}{C_\pi + C_\mu} \quad \text{(in rad/s)} \]  

(8.167)

or

\[ f_T = \frac{g_m}{2\pi(C_\pi + C_\mu)} \quad \text{(in Hz)} \]  

(8.168)

where \(\omega_T\) or \(f_T\) is called the transition frequency and is a measure of the useful frequency of the transistor when used as an amplifier. \(\omega_T\) is also the unity-gain bandwidth of the transistor because from Eqs. (8.166) and (8.167) we get

\[ \omega_T = \omega_\beta \beta_T \]  

(8.169)

Unity-gain bandwidth is usually specified by the manufacturer, with typical values ranging from 100 MHz to a few gigahertz \(\omega_T\) is normally determined by measuring the frequency \(\omega_\chi\) when \(|\beta_T(j\omega_\chi)| = 10\) or 5. That is, \(\omega_T = \omega_\chi |\beta_T(j\omega_\chi)|\). The transition period \(\tau_T\) corresponding to \(\omega_T\) is given by

\[ \tau_T = \frac{1}{\omega_T} = \frac{C_\pi + C_\mu}{g_m} = \frac{C_b + C_{je} + C_\mu}{g_m} = \frac{C_b + C_{je} + C_\mu}{g_m} \]

(8.170)
which depends on $g_m$, which in turn depends on the collector current $I_C$ through $g_m (= I_C / V_T)$. As $I_C$ decreases, the terms involving $C_{je}$ and $C_{jc}$ dominate, causing $\tau_T$ to rise and $f_T$ to fall, as shown in Fig. 8.52(d). At high values of $I_C$, however, $\tau_T$ approaches the value of transition time $\tau_F$, which increases with current, causing the frequency to decrease.

**EXAMPLE 8.9**

Finding the high-frequency model parameters of a BJT  Use the DC-biasing values of the transistor circuit in Fig. 8.20(a): $I_C = 10$ mA, $V_{CE} = 5$ V, $V_{BE} = 0.7$ V, and $V_{CS} = V_C = 10$ V. This circuit is shown in Fig. 8.53. The parameters of the transistor are as follows: $C_{je0} = 29.6$ pF, $V_{je} = 0.8$ V for determining $C_{je}$, $C_{jc0} = 19.4$ pF, $V_{jc} = 0.8$ V for determining $C_{jc}$, $C_{cs0} = 3$ pF, $V_{js} = 0.8$ V for determining $C_{cs}$, and $\beta_T = 100$. Assume that $V_T = 25.8$ mV and that the substrate is connected to the ground. The transition frequency is $f_T = 300$ MHz at $V_{CE} = 20$ V and $I_C = 20$ mA.

(a) Find transition time $\tau_T$.

(b) Calculate the small-signal capacitances of the high-frequency model in Fig. 8.50(a).

(c) Use PSpice/SPICE to generate the model parameters.

**SOLUTION**

(a) The transition period is $\tau_T = 1 / 2\pi f_T = 1 / (2\pi \times 300$ MHz) = 530.5 ps. The transition frequency $f_T = 300$ MHz is specified at $I_C = 20$ mA. The transconductance $g_m$ (at $I_C = 20$ mA) becomes

$$g_m = \frac{I_C}{V_T} = \frac{20 \text{ mA}}{25.8 \text{ mV}} = 775.2 \text{ mA/V}$$

We know that

$$V_{CB} = V_{CE} - V_{BE} = 20 - 0.7 = 19.3 \text{ V}$$

**FIGURE 8.53** DC-biasing circuit for a BJT
Equation (8.161) gives

\[ C_\mu = \frac{19.4 \text{ pF}}{(1 + 19.3/0.8)^{1/3}} = 6.62 \text{ pF} \]

From Eq. (8.170) we can find the transit time \( \tau_F \):

\[ 530.5 \text{ ps} = \tau_F + \frac{25 \text{ pF}}{0.7752} + \frac{6.62 \text{ pF}}{0.7752} \]

which gives \( \tau_F = 489.7 \text{ ps} \).

(b) We know that

From Eq. (8.159),

\[ C_{je} = \frac{C_{je0}}{(1 - V_{BE}/V_{je})^{1/3}} = \frac{29.6 \text{ pF}}{(1 - 0.7/0.8)^{1/3}} = 59.2 \text{ pF} \]

From Eq. (8.161),

\[ C_{\pi} = \frac{C_{\pi0}}{(1 + V_{CB}/V_{je})^{1/3}} = \frac{19.4 \text{ pF}}{(1 + 4.3/0.8)^{1/3}} = 10.47 \text{ pF} \]

From Eq. (8.162),

\[ C_{cs} = \frac{C_{cs0}}{(1 + V_{CS}/V_{js})^{1/3}} = \frac{3 \text{ pF}}{(1 + 10/0.8)^{1/3}} = 1.26 \text{ pF} \]

From Eq. (8.158),

\[ C_b = \frac{\tau_F C}{V_T} = \frac{489.7 \times 10^{-12} \times 10 \text{ mA}}{25.8 \text{ mV}} = 189.8 \text{ pF} \]

Then

\[ C_\pi = C_b + C_{je} = 189.8 \text{ pF} + 59.2 \text{ pF} = 249 \text{ pF} \]

(c) Since the capacitances of a BJT depend on the junction voltages, PSpice/SPICE parameters for BJTs are specified at the zero-biased conditions. PSpice/SPICE first calculates the biasing voltages by finding the Q-point and then adjusts the values of junction capacitances accordingly. We will add the PSpice zero-biased parameters affecting the capacitances. That is, \( C_{JE} = C_{je0} = 29.6 \text{ pF}, C_{JC} = C_{IC} = 19.4 \text{ pF}, C_{JS} = C_{cs} = 1.26 \text{ pF}, V_{JE} = V_{je} = 0.8 \text{ V}, V_{JC} = V_{jc} = 0.8 \text{ V}, V_{JS} = V_{js} = 0.8 \text{ V}, \) and \( TF = \tau_F = 489.7 \text{ ps} \). The model statement for the transistor Q2N2222 in Example 8.5 is as follows for \( V_{CS} = 0 \):

\[ .MODEL Q2N2222 NPN \ (BF=100 \ IS=3.295E-14 \ VA=200 \ CJE=29.6 \text{pF} + \ CJC=19.4 \text{pF} \ CJS=1.3 \text{pF} \ VJE=0.8 \ VJC=0.8 \ VJS=0.8 \ TF=489.7 \text{ps}) \]

The results of .OP analysis, which are obtained from the output file, are as follows: (The values obtained from hand calculations are shown in parentheses; the results obtained from the PSpice model are shown in the left-hand column.)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GM</td>
<td>3.54E-01</td>
<td>( g_m = 0.354 \text{ A/V} ) \ (0.3876 \text{ A/V})</td>
</tr>
<tr>
<td>RPI</td>
<td>2.90E+02</td>
<td>( r_\pi = 290 \Omega ) \ (258 \Omega)</td>
</tr>
<tr>
<td>RO</td>
<td>2.24E+04</td>
<td>( r_0 = 22.4 \text{ k}\Omega ) \ (20 \text{ k}\Omega)</td>
</tr>
<tr>
<td>CBE</td>
<td>2.19E-10</td>
<td>( C_\pi = 219 \text{ pF} ) \ (243.6 \text{ pF})</td>
</tr>
<tr>
<td>CBC</td>
<td>1.00E-11</td>
<td>( C_\mu = 10 \text{ pF} ) \ (10.47 \text{ pF})</td>
</tr>
<tr>
<td>CJS</td>
<td>1.26E-12</td>
<td>( C_{cs} = 1.26 \text{ pF} ) \ (1.26 \text{ pF})</td>
</tr>
<tr>
<td>FT</td>
<td>2.46E+08</td>
<td>( f_T = 246 \text{ MHz} ) \ (308.49 \text{ MHz})</td>
</tr>
</tbody>
</table>
The maximum useful frequency of a BJT is called the transition frequency, and it is limited by the internal capacitances $C_\pi$ and $C_\mu$ of the BJT. The collector–base capacitance $C_\mu$ is small; however, it has a magnified influence on the frequency response as a result of the Miller effect.

The base transit time is the average time each majority carrier spends crossing the base.

The transition frequency, also known as the unity-gain bandwidth of the transistor, is a measure of the useful frequency and is fixed for a particular transistor.

8.15 Frequency Response of BJT Amplifiers

In Secs. 2.5.4 and 2.5.5, we introduced the short-circuit and zero-value methods. As examples, we will use these methods for determining the frequency response of BJT amplifiers.

8.15.1 Common-Emitter BJT Amplifiers

A common-emitter BJT amplifier is shown in Fig. 8.54(a). The transistor can be replaced by its simple high-frequency $\pi$ model, shown in Fig. 8.54(b). The values of $C_\pi$ and $C_\mu$ are low (on the order of 10 pF), and these capacitors can be considered to be open-circuited at a low frequency. Thus, Fig. 8.54(b) is reduced to Fig. 8.54(c) at a low frequency. If the transistor can be replaced by its small-signal AC model, the frequency response will depend on the time constants of the model’s capacitors. The values of $C_1$, $C_2$, and $C_E$ are generally much larger than those of $C_\pi$ and $C_\mu$. The amplifier will exhibit a midband characteristic. A typical frequency response profile is shown in Fig. 8.54(d), where $f_L$ is the low 3-dB frequency, $f_H$ is the high 3-dB frequency, and $A_{PB}$ is the passband gain. The low break frequencies will depend mostly on $C_1$, $C_2$, and $C_E$; the high break frequencies will depend on $C_\pi$ and $C_\mu$. An extra capacitance $C_x$ is connected between the collector and the base to give the desired high break frequency.

Low Cutoff Frequencies

The common-emitter (CE) amplifier in Fig. 8.54(a) is expected to operate at low frequencies such that $C_\pi$ and $C_\mu$ will have small values and will behave as if they were open-circuited. The low-frequency equivalent circuit shown in Fig. 8.55 has three capacitors—two coupling capacitors $C_1$ and $C_2$ and a bypass capacitor $C_E$. 

NOTE: There are many factors that affect the parameters of BJTs. The hand calculations are expected to give approximate values only. Even the PSpice results will differ from results of measurements on practical BJTs.
Let us consider the effects of $C_1$ only; $C_2$ and $C_E$ are short-circuited. This situation is shown in Fig. 8.56(a). Thevenin’s equivalent resistance presented to $C_1$ is

$$R_{C1} = R_s + R_B \parallel r_\pi$$

(8.171)

where $R_B = (R_1 \parallel R_2)$. Thus, the break frequency due to $C_1$ only is

$$f_{C1} = \frac{1}{2\pi R_{C1}C_1}$$

(8.172)

The equivalent circuit, with $C_1$ and $C_E$ short-circuited, is shown in Fig. 8.56(b). Thevenin’s equivalent resistance is given by

$$R_{C2} = R_C + R_L$$

(8.173)

FIGURE 8.54  Common-emitter BJT amplifier

FIGURE 8.55  Low-frequency AC equivalent circuit of a common-emitter amplifier
The break frequency due to $C_2$ only is

$$f_{C2} = \frac{1}{2\pi R_{C2}C_2} \quad (8.174)$$

The equivalent circuit, with $C_1$ and $C_2$ short-circuited, is shown in Fig. 8.56(c). Dividing a resistance in the base circuit by $(1 + \beta_f) (1 + g_m r_\pi)$ will give the equivalent emitter resistance. Thus, Thévenin’s equivalent resistance can be found by paralleling $R_E$ with the equivalent B-E resistance. That is,

$$R_{CE} = R_E \parallel \frac{r_\pi + (R_s || R_B)}{1 + \beta_f} \quad (8.175)$$

If $R_s < R_B$ and $r_\pi, R_E > 1\ \text{k}\Omega$, and $\beta_f >> 1$, Eq. (8.175) can be approximated by

$$R_{CE} \approx \frac{r_\pi}{\beta_f} = \frac{1}{g_m} \quad (8.176)$$

The break frequency due to $C_E$ only is

$$f_{CE} = \frac{1}{2\pi R_{CE}C_E} \quad (8.177)$$

The low 3-dB frequency $f_L$ is the largest of $f_{C1}, f_{C2}$, and $f_{CE}$. In general, the value of $C_E$ is much larger than that of $C_1$ or $C_2$; the value of $R_{CE}$ is the smallest. Thus, $f_{CE}$ is generally the low 3-dB frequency: $f_L = f_{CE}$.

### High Cutoff Frequencies

At high frequencies, coupling and bypass capacitors offer very low impedances because of their high values and can be assumed to be short-circuited. However, the impedances due to transistor capacitors are comparable to those of other circuit elements and hence affect the voltage gain.

If the transistor of the CE amplifier in Fig. 8.54(a) is replaced by the high-frequency model shown in Fig. 8.54(b), the result is the high-frequency equivalent circuit shown in Fig. 8.57. $C_x$ is the extra
capacitance connected between the collector and the base of the transistor. The equivalent circuit, with \( C_{\pi} \) open-circuited, is shown in Fig. 8.58(a). The resistance faced by \( C_{\pi} \) and \( C_x \) is given by

\[
R_{C\pi} = r_{\pi} \left( R_s \parallel R_B \right)
\]

(8.178)

The equivalent circuit, with \( C_{\pi} \) open-circuited, is shown in Fig. 8.58(b). Let us replace \( C_{\pi} \) by voltage source \( v_x \), as shown in Fig. 8.58(c). Using KVL, we get

\[
v_x = v_{bc} + (R_L \parallel R_C)(i_x + g_{m}v_{be}) = R'_i i_x + (R_L \parallel R_C)(i_x + g_{m}i_x R'_i)
\]

\[
= \left[(R_L \parallel R_C) + R'_i(1 + g_{m}R_L \parallel R_C)\right]i_x
\]

which gives Thevenin’s equivalent resistance faced by \( C_{\pi} \) as

\[
R_{C\mu} = \frac{v_x}{i_x} = R_L \parallel R_C + R'_i[1 + g_{m}(R_L \parallel R_C)]
\]

(8.179)

where \( R'_i = (r_{\pi} \parallel R_B \parallel R_s) \). Thus, the high 3-dB frequency \( f_H \) is given by

\[
f_H = \frac{1}{2\pi[R_{C\pi}C_{\mu} + R_{C\mu}(C_{\mu} + C_x)]}
\]

(8.180)

\[\text{FIGURE 8.57} \quad \text{High-frequency circuit of a common-emitter amplifier}\]

\[\text{FIGURE 8.58} \quad \text{High-frequency equivalent circuits for the zero-value method}\]
EXAMPLE 8.10

Designing a common-emitter amplifier to give a specified frequency response

(a) Design a CE amplifier as shown in Fig. 8.54(a), setting the low 3-dB frequency at \( f_\text{L} = 150 \) Hz and the high 3-dB frequency at \( f_\text{H} = 250 \) kHz. The circuit parameters are \( f_\text{B} = 80 \), \( g_m = 57.14 \mu A/V \), \( r_\pi = 1.4 \) kΩ, \( C_\pi = 15 \) pF, \( C_\mu = 1 \) pF, \( R_1 = 200 \) Ω, \( R_2 = 4.3 \) kΩ, \( R_E = 330 \) Ω, \( R_C = 5 \) kΩ, and \( R_L = 5 \) kΩ.

(b) Determine the passband gain \( A_{PB} \).

(c) Use Miller’s capacitor method to check the high-frequency design.

(d) Use PSpice/SPICE to plot the frequency response from 100 Hz to 1 MHz with decade increments and 100 points per decade.

SOLUTION

Let \( R_B = R_1 \parallel R_2 = 7 \) kΩ \parallel 4.3 \) kΩ = 2.66 kΩ. Then

\[
R'_\pi = r_\pi \parallel R_B \parallel R_s = R_{CE} = 164 \Omega
\]

(a) The design will have two parts: one in which we set the low 3-dB frequency at \( f_\text{L} = 150 \) Hz and one in which we set the high 3-dB frequency at \( f_\text{H} = 250 \) kHz.

The steps to set \( f_\text{L} = 150 \) Hz are as follows:

**Step 1.** Calculate the equivalent resistances \( R_{C1}, R_{C2}, \) and \( R_{CE} \). From Eq. (8.171),

\[
R_{C1} = R_s + R_B \parallel r_\pi = 200 \Omega + 2.66 \) kΩ \parallel 1.4 \) kΩ = 1.12 kΩ
\]

From Eq. (8.173),

\[
R_{C2} = R_C + R_L = 5 \) kΩ + 5 \) kΩ = 10 kΩ
\]

From Eq. (8.175),

\[
R_{CE} = 330 \parallel \left[ \frac{1.4 \) kΩ + 200 \) Ω}{1} \parallel \frac{2.66 \) kΩ}{80} \right] = 18.48 \Omega
\]

**Step 2.** Assume that the frequency corresponding to the lowest resistance is the dominant cutoff frequency \( f_\text{L} \). The low 3-dB frequency can be assigned to any resistance. However, assigning the low 3-dB frequency to the lowest resistance will reduce the values of coupling capacitors. Since \( R_{CE} \) has the lowest value, let \( f_{CE} = f_\text{L} \). That is, \( f_{CE} = f_\text{L} = f_\text{L} = 150 \) Hz.

**Step 3.** Calculate the required value of \( C_E \) from Eq. (8.177):

\[
f_{CE} = \frac{1}{2\pi R_{CE} C_E} = \frac{1}{2\pi \times 18.48 \times C_E} = 150 \) Hz or \( C_E = 57.4 \mu F
\]

**Step 4.** Set the frequency corresponding to the next higher resistance. Let \( f_{C1} = f_{CL} = 150/10 = 15 \) Hz. From Eq. (8.172),

\[
f_{C1} = \frac{1}{2\pi R_{C1} C_1} = \frac{1}{2\pi \times 1.12 \) kΩ \times C_1} = 15 \) Hz or \( C_1 = 9.5 \mu F
\]

**Step 5.** Set the frequency corresponding to the highest resistance. Let \( f_{C2} = f_{CL} = 150/20 = 7.5 \) Hz. From Eq. (8.174),

\[
f_{C2} = \frac{1}{2\pi R_{C2} C_2} = \frac{1}{2\pi \times 10 \) kΩ \times C_2} = 7.5 \) Hz or \( C_2 = 2.1 \mu F
\]
The steps to set $f_H = 250$ kHz are as follows:

**Step 1.** From Eq. (8.178),

$$R_C \pi = r_\pi \parallel (R_s \parallel R_B) = 1.4 \ \text{k}\Omega \parallel (200 \parallel 2.66 \ \text{k}\Omega) = 164 \ \text{\Omega}$$

From Eq. (8.179),

$$R_C \mu = (5 \ \text{k}\Omega \parallel 5 \ \text{k}\Omega) + 164 \ \text{\Omega} \times [1 + 57.14 \ \text{m\Omega} \times (5 \ \text{k}\Omega \parallel 5 \ \text{k}\Omega)] = 26.1 \ \text{k}\Omega$$

**Step 2.** From Eq. (8.180), the high 3-dB frequency $f_H$ is

$$f_H = \frac{1}{2\pi[164 \ \text{\Omega} \times 15 \ \text{pF} + (26.1 \ \text{k}\Omega)(C_\mu + C_\chi)]} 250 \text{ kHz} \quad \text{or} \quad C_\mu + C_\chi = C_{\text{eff}} = 24.3 \ \text{pF}$$

which gives $C_\chi = 24.3 - 1 = 23.3 \ \text{pF}$. This is the value of the additional capacitor $C_\chi$ that is to be connected between the collector and base terminals of the transistor.

(b) From Eq. (2.114), the passband voltage gain is

$$A_{PB} = (-57.14 \times 10^{-3})(5 \ \text{k}\Omega \parallel 5 \ \text{k}\Omega) \times \frac{2.66 \ \text{k}\Omega \parallel 1.4 \ \text{k}\Omega}{200 + (2.66 \ \text{k}\Omega \parallel 1.4 \ \text{k}\Omega)} = -117.3 \ \text{V/V}$$

(c) Applying Eq. (2.98), we have for the effective capacitance between the base and the emitter terminals

$$C_{\text{eq}} = (C_\mu + C_\chi)(1 + g_m(R_L \parallel R_C)) + C_\pi$$

$$= 24.3 \ \text{pF} \times [1 + 57.14 \ \text{m\Omega} \times (5 \ \text{k}\Omega \parallel 5 \ \text{k}\Omega)] + 15 \ \text{pF} = 3.51 \ \text{nF}$$

The equivalent resistance faced by $C_{\text{eq}}$ is $R_{\text{eq}} = r_\pi \parallel R_B \parallel R_s = 164 \ \text{\Omega}$, so

$$f_H = \frac{1}{2\pi C_{\text{eq}} R_{\text{eq}}} = \frac{1}{2\pi \times 3.51 \ \text{nF} \times 164 \ \text{\Omega}} = 276.5 \ \text{kHz}$$

Miller’s capacitor method gives a higher frequency than the zero-value method does. The actual frequency is higher than the one obtained by the zero-value method but lower than the one obtained by Miller’s capacitor method. Thus, design by the zero-value method provides a more conservative estimate, although the SPICE results match the zero-value prediction.

(d) The AC equivalent circuit for PSpice simulation is shown in Fig. 8.59, where $r_\pi$ has been added to include the output resistance of the transistor and also to make the circuit more general.

![FIGURE 8.59 Small-signal equivalent circuit for PSpice simulation](image-url)
8.15.2 Common-Collector BJT Amplifiers

The techniques for determining the frequency response of a common-collector (CC) amplifier are identical to those used with a CE amplifier. A CC amplifier is shown in Fig. 8.61. Given the small-signal AC model of the transistor(s), we will derive expressions for the low and high break frequencies.

![Common-collector amplifier](image)

**FIGURE 8.61** Common-collector amplifier

**NOTE:** The main objective of this PSpice simulation was to verify the design methods used to set the low and high cutoff frequencies. Thus, the small-signal model rather than the actual PSpice transistor model was used in the simulation. If we designed an amplifier with the small-signal model and then ran the simulation with an actual PSpice transistor, we would expect to get an error. But, the results should be close.
Low Cutoff Frequencies

The low-frequency equivalent circuit obtained by replacing the transistor in Fig. 8.61 by its small-signal low-frequency model is shown in Fig. 8.62(a). There will be two low break frequencies corresponding to the coupling capacitances $C_1$ and $C_2$. Assuming that $C_2$ is short-circuited, as shown in Fig. 8.62(b), $R_E$ becomes parallel to $R_L$. Converting the effective emitter resistance ($R_E/\beta$) into the base terminal and using Eq. (8.101), we get the equivalent input resistance

$$R_I = R_B \| \left[ r_\pi + (1 + \beta_f)(R_E \| R_L) \right]$$

(8.181)

where $R_B = R_1 \| R_2$. Thevenin’s equivalent resistance presented to $C_1$ is

$$R_{C1} = R_s + R_i$$

so the 3-dB frequency due to $C_1$ only is

$$f_{C1} = \frac{1}{2\pi R_{C1} C_1}$$

(8.182)

The equivalent circuit, with $C_1$ short-circuited, is shown in Fig. 8.62(c). Converting the effective base resistance [$r_\pi + (R_s \| R_B)$] into the emitter terminal and using Eq. (8.103), we get the equivalent output resistance

$$R_o = \frac{R_E \| r_\pi + (R_s \| R_B)}{1 + \beta_f}$$

(8.183)
If \( R_s << R_B \) and \( r_\pi \), and \( R_E >> 1 \) kΩ, Eq. (8.183) can be approximated by

\[
R_o \approx \frac{r_\pi}{\beta_f} = \frac{1}{g_m} \quad \text{for } \beta_f >> 1
\]  
(8.184)

Thevenin’s equivalent resistance presented to \( C_2 \) is

\[ R_{C2} = R_L + R_o \]

The 3-dB frequency due to \( C_2 \) only is

\[
f_{C2} = \frac{1}{2\pi R_{C2}C_2}
\]  
(8.185)

In a CC amplifier, \( R_{C2} \) is generally much lower than \( R_{C1} \). For the same values of \( C_1 \) and \( C_2 \), \( f_{C2} \) will become the low 3-dB frequency. That is, \( f_L = f_{C2} \).

**High Cutoff Frequencies**

If the transistor in Fig. 8.61 is replaced by its high-frequency model, we get the high-frequency equivalent circuit shown in Fig. 8.63. Since there is no phase reversal between the input and output voltages, Miller’s method cannot be applied.

If we assume \( C_\pi \) is open-circuited and \( v_s = 0 \), the equivalent circuit is shown in Fig. 8.64(a). The resistance looking to the left of \( C_\mu \) is \( (R_B \parallel R_s) \parallel [r_\pi + (1 + \beta_f)(R_E \parallel R_L)] \). Thus, Thevenin’s equivalent resistance presented to \( C_\mu \) is

\[
R_{C\mu} = (R_B \parallel R_s) \parallel [r_\pi + (1 + \beta_f)(R_E \parallel R_L)]
\]  
(8.186)

If we assume \( C_\mu \) is open-circuited, the equivalent circuit is shown in Fig. 8.64(b). To find \( R_{C\pi} \), let us remove \( C_\pi \) and apply a test voltage \( v_x \), as shown in Fig. 8.64(c). Using KVL around the loop formed by \( R_B \) in parallel with \( R_s \) and by \( R_L \) in parallel with \( R_E \), we get

\[
v_x = (R_B \parallel R_s) \left( i_x - \frac{v_x}{r_\pi} \right) + (R_L \parallel R_E) \left( i_x - \frac{v_x}{r_\pi} - g_m v_x \right)
\]

which can be simplified to

\[
i_x(R_B \parallel R_s + R_L \parallel R_E) = v_x \left[ 1 + \frac{R_B \parallel R_s}{r_\pi} + \frac{R_L \parallel R_E}{r_\pi} + g_m(R_L \parallel R_E) \right]
\]

or

\[
\frac{i_x}{v_x} = \frac{1}{R_B \parallel R_s + R_L \parallel R_E} + \frac{1 + g_m(R_L \parallel R_E)}{R_B \parallel R_s + R_L \parallel R_E}
\]

**FIGURE 8.63** High-frequency circuit of a common-collector amplifier
Thus, Thevenin’s equivalent resistance presented to $C_{\pi}$ is

$$R_{C_{\pi}} = \frac{v_x}{i_x} = r_{\pi} \left\| R_B \parallel R_s + R_L \parallel R_E \right\| \frac{1}{1 + g_m(R_L \parallel R_E)}$$

and the high 3-dB frequency is

$$f_H = \frac{1}{2\pi(R_{C_{\pi}}C_{\pi} + R_{C_{\mu}}C_{\mu})}$$

**EXAMPLE 8.11**

**Designing a common-collector amplifier to give a specified frequency response**

(a) Design a CC BJT amplifier as shown in Fig. 8.61 to give a low 3-dB frequency of $f_L = 150$ Hz. Calculate the values of $C_1$ and $C_2$. The circuit parameters of the CC BJT amplifier are $C_{\pi} = 15$ pF, $C_{\mu} = 1$ pF, $g_m = 57.14$ m$\Omega$, $\beta_f = 80$, $R_s = 200$ $\Omega$, $r_{\pi} = 1.4$ k$\Omega$, $R_E = 330$ $\Omega$, $R_1 = 7$ k$\Omega$, $R_2 = 4.3$ k$\Omega$, and $R_L = 5$ k$\Omega$.

(b) Calculate the high 3-dB cutoff frequency $f_H$.

**SOLUTION**

$$R_B = R_1 \parallel R_2 = 7 \text{ k$\Omega$} \parallel 4.3 \text{ k$\Omega$} = 2.66 \text{ k$\Omega$}.$$  

(a) The design steps for the low 3-dB frequency are as follows:
8.15.3 Common-Base BJT Amplifiers

A common-base (CB) amplifier has a higher 3-dB frequency than either a CE or a CC amplifier. A CB amplifier is shown in Fig. 8.65. We will derive expressions for the low and high cutoff frequencies.

Low Cutoff Frequencies

The low-frequency equivalent circuit of the CB amplifier in Fig. 8.65 is shown in Fig. 8.66(a). There are two coupling capacitors, C1 and C2, and one bypass capacitor, CB. If we assume C2 and CB are short-circuited, the equivalent circuit is shown in Fig. 8.66(b). The resistance \( R_t \) representing the current source is

\[
R_t = \frac{v_{be}}{g_m v_{be}} = \frac{1}{g_m} = \frac{r_\pi}{\beta_t}
\]
Thus, the input resistance $R_i$ at the emitter terminal is

$$R_i = R_E \parallel r_\pi \parallel R_t = R_E \parallel r_\pi \parallel \frac{r_\pi}{\beta_f} = R_E \parallel \frac{r_\pi}{1 + \beta_f}$$

(8.189)

and Thevenin’s resistance presented to $C_1$ is

$$R_{C1} = R_s + R_i$$

Thus, the break frequency due to $C_1$ only is

$$f_{c1} = \frac{1}{2\pi R_{C1} C_1}$$

(8.190)
If we consider $C_1$ and $C_B$ to be short-circuited, the equivalent circuit is shown in Fig. 8.65(c). Thevenin’s resistance presented to $C_2$ is given by

$$R_{C2} = R_C + R_L$$  \hspace{1cm} (8.191)

and the break frequency due to $C_2$ only is

$$f_{C2} = \frac{1}{2\pi R_{C2}C_2}$$  \hspace{1cm} (8.192)

The equivalent circuit, with $C_1$ and $C_2$ short-circuited, is shown in Fig. 8.65(d). $R_s \parallel R_E$, which is in the emitter circuit, has to be converted to an equivalent value in the base circuit. To find the resistance presented to $C_B$, we multiply $(R_s \parallel R_E)$ by $(1 + \beta_I)$ and then combine the result in series with $r_\pi$; the combination forms a parallel circuit with $R_B (=R_1 \parallel R_2)$. That is, Thevenin’s resistance becomes

$$R_{CB} = R_B \parallel [r_\pi + (1 + \beta_I)(R_s \parallel R_E)]$$  \hspace{1cm} (8.193)

The break frequency due to $C_B$ only is

$$f_{CB} = \frac{1}{2\pi R_{CB}C_B}$$  \hspace{1cm} (8.194)

The low 3-dB frequency $f_L$ is the largest of $f_{C1}$, $f_{C2}$, and $f_{CE}$. In general, the value of $R_{C1}$ is on the order of a few hundred ohms, whereas $R_{C2}$ and $R_{CB}$ are on the order of kiloohms. To limit the values of the coupling capacitors, $f_{C1}$ is normally chosen as the low 3-dB frequency; that is, $f_{C1} = f_L$.

**High Cutoff Frequencies**

The high-frequency equivalent circuit for the CB BJT amplifier in Fig. 8.65 is shown in Fig. 8.67(a). If we assume $C_\pi$ is open-circuited, the equivalent circuit is shown in Fig. 8.67(b). The current source to the left of $C_\mu$ isolates the resistances to its left. Thevenin’s equivalent resistance presented to $C_\mu$ is

$$R_{C\mu} = R_C \parallel R_L$$  \hspace{1cm} (8.195)

![High-frequency equivalent circuit](image-url)
If we assume $C_{\mu}$ is open-circuited, the equivalent circuit is shown in Fig. 8.67(c). The resistance offered by the current source is $1/g_m$, which forms a parallel circuit with $R_E$, $R_s$, and $r_\pi$. Thevenin’s equivalent resistance presented to $C_{\pi}$ is given by

$$R_{C\pi} = (R_s \parallel R_E) \parallel r_\pi \parallel \frac{1}{g_m}$$

(8.196)

Thus, the high 3-dB frequency is

$$f_H = \frac{1}{2\pi (R_{C\pi}C_{\pi} + R_{C\mu}C_{\mu})}$$

(8.197)

In general, $1/g_m$ has a small value and is much less than $R_s$, $R_E$, or $r_\pi$; that is, $R_{C\pi} \approx 1/g_m$. For $R_{C\mu} \gg R_{C\pi}$, which is normally the case, Eq. (8.197) can be approximated by

$$f_H \approx \frac{1}{2\pi R_{C\mu}C_{\mu}}$$

(8.198)

**NOTE** $f_H$ is independent of the transistor transconductance gain $g_m$, and there is no Miller’s capacitance multiplication effect. CB amplifiers are used for high-frequency applications.

**EXAMPLE 8.12**

**Designing a common-base amplifier to give a specified frequency response**

(a) Design a CB BJT amplifier as shown in Fig. 8.65 to give a low 3-dB frequency of $f_L = 150$ Hz. Calculate the values of $C_1$, $C_2$, and $C_B$. The circuit parameters are $C_{\pi} = 15$ pF, $C_{\mu} = 1$ pF, $g_m = 57.14$ m$\Omega$, $f_t = 80$, $R_s = 200 \Omega$, $r_\pi = 1.4$ k$\Omega$, $R_E = 330 \Omega$, $R_C = 5$ k$\Omega$, $R_1 = 7$ k$\Omega$, $R_2 = 4.3$ k$\Omega$, and $R_L = 5$ k$\Omega$.

(b) Calculate the high 3-dB frequency $f_H$.

**SOLUTION**

$$R_B = R_1 \parallel R_2 = 7 \text{k}\Omega \parallel 4.3 \text{k}\Omega = 2.66 \text{k}\Omega$$

(a) The design steps are as follows:

**Step 1.** Calculate the equivalent resistances $R_{C1}$, $R_{C2}$, and $R_{CB}$.

From Eq. (8.189),

$$R_1 = 330 \Omega \parallel \frac{1.4 \text{k}\Omega}{1 + 80} = 16.4 \Omega$$

and

$$R_{C1} = R_s + R_1 = 200 + 16.4 = 216.4 \Omega$$

From Eq. (8.191),

$$R_{C2} = R_C + R_L = 5 \text{k}\Omega + 5 \text{k}\Omega = 10 \text{k}\Omega$$

From Eq. (8.193),

$$R_{CB} = 2.66 \text{k}\Omega \parallel [1.4 \text{k}\Omega + (1 + 80) (200 \Omega \parallel 330 \Omega)] = 2.16 \text{k}\Omega$$

**Step 2.** At the lowest resistance, the dominant cutoff frequency is $f_{C1} = f_L = 150$ Hz.
**8.15.4 Multistage Amplifiers**

Multistage amplifiers are often used to meet voltage gain, frequency range, input impedance, and/or output impedance requirements. In this section, we apply the short-circuit and zero-value methods to determine the cutoff frequencies of multistage amplifiers. Some equations will be similar to those used in the preceding sections because the equivalent circuits for the amplifiers are similar to those encountered previously.

When a capacitor $C_{\mu}$ is connected between the base (or gate) and the collector (or drain) of a transistor, it greatly influences the high 3-dB frequency.

**EXAMPLE 8.13**

Finding the frequency response of a two-stage CE-CE BJT amplifier  
A two-stage CE-CE BJT amplifier is shown in Fig. 8.68. The circuit parameters are $C_{\pi 1} = C_{\pi 2} = 15 \text{ pF}$, $C_{\mu 1} = C_{\mu 2} = 1 \text{ pF}$, $g_{m1} = g_{m2} = 57.14 \text{ mS}$, $R_1 = 200 \text{ k}\Omega$, $R_{11} = 22 \text{ k}\Omega$, $R_{21} = 47 \text{ k}\Omega$, $R_{C1} = 8 \text{ k}\Omega$, $R_{E1} = 5 \text{ k}\Omega$, $R_{12} = 22 \text{ k}\Omega$, $R_{22} = 47 \text{ k}\Omega$, $R_{C2} = 8 \text{ k}\Omega$, $R_{E2} = 5 \text{ k}\Omega$, $R_L = 5 \text{ k}\Omega$, $r_{\pi 1} = r_{\pi 2} = 1.4 \text{ k}\Omega$, $\beta_{11} = 100$, $\beta_{12} = 150$, $C_1 = 10 \mu\text{F}$, $C_2 = 5 \mu\text{F}$, $C_3 = 10 \mu\text{F}$, $C_{E1} = 50 \mu\text{F}$, and $C_{E2} = 50 \mu\text{F}$.

(a) Calculate the low 3-dB frequency $f_L$.

(b) Calculate the high 3-dB frequency $f_H$. 

---

**Step 3.** Calculate the required value of $C_1$ from Eq. (8.190):

$$f_{C1} = \frac{1}{2\pi R_{C1} C_1} = \frac{1}{2\pi \times 216.4 \times C_1} = 150 \text{ Hz} \quad \text{or} \quad C_1 = 4.9 \mu\text{F}$$

**Step 4.** At the next higher resistance $R_{CB} = 2.16 \text{ k}\Omega$, $f_{C1} = \frac{f_L}{10} = 150/10 = 15 \text{ Hz}$.

**Step 5.** Calculate the required value of $C_B$ from Eq. (8.194):

$$f_{CB} = \frac{1}{2\pi R_{CB} C_B} = \frac{1}{2\pi \times 2.16 \text{ k}\Omega \times C_B} = 15 \text{ Hz} \quad \text{or} \quad C_B = 4.91 \mu\text{F}$$

**Step 6.** At the highest resistance $R_{C2} = 10 \text{ k}\Omega$, $f_{C2} = \frac{f_L}{20} = 150/20 = 7.5 \text{ Hz}$.

**Step 7.** Calculate the required value of $C_2$ from Eq. (8.192):

$$f_{C2} = \frac{1}{2\pi R_{C2} C_2} = \frac{1}{2\pi \times 10 \text{ k}\Omega \times C_2} = 7.5 \text{ Hz} \quad \text{or} \quad C_2 = 2.12 \mu\text{F}$$

(b) From Eq. (8.195),

$$R_{C\mu} = R_C \parallel R_L = 5 \text{ k}\Omega \parallel 5 \text{ k}\Omega = 2.5 \text{ k}\Omega$$

From Eq. (8.196),

$$R_{C\pi} = (200 \text{ } \Omega \parallel 330 \text{ } \Omega) \parallel 1.4 \text{ k}\Omega \parallel \frac{1}{57.14 \text{ m}\Omega} = 15.2 \text{ } \Omega$$

From Eq. (8.197), the high 3-dB frequency is

$$f_H = \frac{1}{2\pi(R_{C\pi} + R_{C\mu} C\mu)} = \frac{1}{2\pi(15.2 \text{ } \Omega \times 15 \text{ pF} + 2.5 \text{ k}\Omega \times 1 \text{ pF})} = 58.34 \text{ MHz}$$
SOLUTION

We have

\[ R_{B1} = R_{11} \parallel R_{21} = 22 \, k\Omega \parallel 47 \, k\Omega = 15 \, k\Omega \]

and \[ R_{B2} = R_{12} \parallel R_{22} = 22 \, k\Omega \parallel 47 \, k\Omega = 15 \, k\Omega \]

(a) The circuit has five external capacitors—three coupling capacitors and two emitter bypass capacitors. The low-frequency AC equivalent circuit is shown in Fig. 8.69(a).

The time constant \( \tau_1 \) due to \( C_1 \) only is

\[ \tau_1 = [R_s + (R_{B1} \parallel r_{\pi 1})]C_1 \]

\[ = [200 \, \Omega + (15 \, k\Omega \parallel 1.4 \, k\Omega)] \times 10 \, \mu F = 14.8 \, ms \]
The time constant \( \tau_2 \) due to \( C_2 \) only is
\[
\tau_2 = [R_{C1} + (R_{B2} \| r_{\pi2})]C_2 \\
= [8 \, k\Omega + (15 \, k\Omega \| 1.4 \, k\Omega)] \times 5 \, \mu F = 46.4 \, ms
\]

The time constant \( \tau_3 \) due to \( C_3 \) only is
\[
\tau_3 = [R_{C2} + R_L]C_3 \\
= [8 \, k\Omega + 5 \, k\Omega] \times 10 \, \mu F = 130 \, ms
\]

The time constant \( \tau_4 \) due to \( C_{E1} \) only is
\[
\tau_4 = \left[ R_{E1} \parallel \frac{r_{\pi1} + (R_s \| R_{B1})}{1 + \beta_{f1}} \right]C_{E1} \\
= \left[ 5 \, k\Omega \parallel \frac{1.4 \, k\Omega + (200 \, \Omega \| 15 \, k\Omega)}{1 + 100} \right] \times 50 \, \mu F = 0.79 \, ms
\]

The time constant \( \tau_5 \) due to \( C_{E2} \) only is
\[
\tau_5 = \left[ R_{E2} \parallel \frac{r_{\pi2} + (R_{C1} \| R_{B2})}{1 + \beta_{f2}} \right]C_{E2} \\
= \left[ 5 \, k\Omega \parallel \frac{1.4 \, k\Omega + (8 \, k\Omega \| 15 \, k\Omega)}{1 + 150} \right] \times 50 \, \mu F = 2.17 \, ms
\]

From Eq. (2.106), the low 3-dB frequency \( f_L \) is
\[
f_L = \frac{1}{2\pi} \left[ \frac{1}{14.8 \, ms} + \frac{1}{46.4 \, ms} + \frac{1}{130 \, ms} + \frac{1}{0.79 \, ms} + \frac{1}{2.17 \, ms} \right] = 290.2 \, Hz
\]

If we consider only the smallest time constant \( \tau_4 = 0.79 \, ms \), we get
\[
f_L = \frac{1}{2\pi \tau_4} = \frac{1}{2\pi \times 0.79 \, ms} = 201.5 \, Hz
\]

(b) Replacing the transistors by their high-frequency model gives the high-frequency equivalent circuit shown in Fig. 8.69(b). If we assume \( r_{\pi1} \) is thevenin’s equivalent resistance faced by \( C_{\pi 1} \) with \( C_{\mu 1}, C_{\mu 2}, \) and \( C_{\pi 2} \)

open-circuited, the time constant \( \tau_{\pi 1} \) can be found from
\[
\tau_{\pi 1} = R_{\pi 1}[C_{\pi 1} = |r_{\pi 1} \parallel (R_s \| R_{B1})|]C_{\pi 1} \\
= [1.4 \, k\Omega \parallel (200 \, \Omega \| 15 \, k\Omega)] \times 15 \, pF = 2.6 \, ns
\]

If \( r_{\pi 2} \) is thevenin’s equivalent resistance faced by \( C_{\pi 2} \) with \( C_{\mu 1}, C_{\mu 2}, \) and \( C_{\pi 1} \)

open-circuited, the time constant \( \tau_{\pi 2} \) can be found from
\[
\tau_{\pi 2} = R_{\pi 2}[C_{\pi 2} = |r_{\pi 2} \parallel R_{B2} \parallel R_{C1}|]C_{\pi 2} \\
= [1.4 \, k\Omega \parallel 15 \, k\Omega \parallel 8 \, k\Omega] \times 15 \, pF = 16.6 \, ns
\]

With \( C_{\mu 2}, C_{\pi 1}, \) and \( C_{\pi 2} \) open-circuited, the effective load resistance of \( C_{\mu 1} \) is
\[
R_{L1(\text{eff})} = r_{\pi 2} \parallel R_{B2} \parallel R_{C1} \\
= 1.4 \, k\Omega \parallel 15 \, k\Omega \parallel 8 \, k\Omega = 1.1 \, k\Omega
\]

and its effective input side resistance of \( C_{\mu 1} \) is
\[
R_{\pi 1(\text{eff})} = r_{\pi 1} \parallel R_s \parallel R_{B1} \\
= 1.4 \, k\Omega \parallel 200 \, \Omega \parallel 15 \, k\Omega = 173 \, \Omega
\]
EXAMPLE 8.14

Finding the frequency response of a two-stage CE-CB BJT amplifier

A two-stage CE-CB amplifier is shown in Fig. 8.70. The circuit parameters are \( C_{\pi 1} = C_{\pi 2} = 15 \, \text{pF}, \ C_{\mu 1} = C_{\mu 2} = 1 \, \text{pF}, \ R_s = 200 \, \Omega, \ R_{11} = 22 \, \Omega, \ R_{32} = 47 \, \Omega, \ R_{C1} = 15 \, \Omega, \ R_{E1} = 9 \, \Omega, \ R_{12} = 22 \, \Omega, \ R_{22} = 47 \, \Omega, \ R_{C2} = 15 \, \Omega, \ R_{E2} = 9 \, \Omega, \ R_L = 10 \, \Omega, \ r_{\pi 1} = r_{\pi 2} = 1.4 \, \text{k}\Omega, \ \beta_{f1} = 100, \ \beta_{f2} = 150, \ g_{m1} = 71.4 \, \text{m}\Omega, \ g_{m2} = 107.1 \, \text{m}\Omega, \ C_1 = 1 \, \mu\text{F}, \ C_2 = 10 \, \mu\text{F}, \ C_3 = 1 \, \mu\text{F}, \ C_E = 10 \, \mu\text{F}, \) and \( C_B = 10 \, \mu\text{F}. \)

(a) Calculate the low 3-dB frequency \( f_L. \)

(b) Calculate the high 3-dB frequency \( f_H. \)

SOLUTION

We have

\[
R_{B1} = R_{11} \parallel R_{21} = 22 \, \text{k}\Omega \parallel 47 \, \text{k}\Omega = 15 \, \text{k}\Omega
\]

\[
R_{B2} = R_{12} \parallel R_{22} = 22 \, \text{k}\Omega \parallel 47 \, \text{k}\Omega = 15 \, \text{k}\Omega
\]

(a) The low-frequency equivalent circuit is shown in Fig. 8.71(a). The time constant \( \tau_1 \) due to \( C_1 \) only is

\[
\tau_1 = [R_s + (R_{B1} \parallel r_{\pi 1})]C_1
\]

\[
= [200 \, \Omega + (15 \, \text{k}\Omega \parallel 1.4 \, \text{k}\Omega)] \times 1 \, \mu\text{F} = 1.48 \, \text{ms}
\]
The time constant \( \tau_2 \) due to \( C_2 \) only is
\[
\tau_2 = \left[ R_{C1} + (R_{E2} \parallel r_{\pi2}) \parallel \frac{1}{g_{m2}} \right] C_2
\]
\[
= \left[ 15 \, \text{k}\Omega + (9 \, \text{k}\Omega \parallel 1.4 \, \text{k}\Omega) \parallel \frac{1000}{107.1 \, \Omega} \right] \times 10 \, \mu\text{F} = 150.1 \, \text{ms}
\]

The time constant \( \tau_3 \) due to \( C_3 \) only is
\[
\tau_3 = [R_{C2} + R_L] C_3
\]
\[
= [15 \, \text{k}\Omega + 10 \, \text{k}\Omega] \times 1 \, \mu\text{F} = 25 \, \text{ms}
\]

The time constant \( \tau_4 \) due to \( C_{E1} \) only is
\[
\tau_4 = \left[ R_{E1} \parallel \frac{r_{\pi1} + (R_s \parallel R_{B1})}{1 + \beta_{f1}} \right] C_{E1}
\]
\[
= \left[ 9 \, \text{k}\Omega \parallel \frac{1.4 \, \text{k}\Omega + 200 \, \Omega \parallel 15 \, \text{k}\Omega}{1 + 100} \right] \times 10 \, \mu\text{F} = 0.16 \, \text{ms}
\]
The time constant $\tau_5$ due to $C_B$ only is

$$\tau_5 = [R_{B2} \parallel r_{\pi 2} + (1 + \beta_2)(R_{C1} \parallel R_{E2})]C_B$$

$$= [15 \, \text{k}\Omega \parallel 1.4 \, \text{k}\Omega + (1 + 150)(15 \, \text{k}\Omega \parallel 9 \, \text{k}\Omega)] \times 10 \, \mu\text{F} = 147.2 \, \text{ms}$$

From Eq. (2.106), the low 3-dB frequency $f_L$ is

$$f_L = \frac{1}{2\pi \tau_4} = \frac{1}{2\pi \times 0.16 \, \text{ms}} = 995 \, \text{Hz}$$

(b) Replacing the transistors by their high-frequency models gives the high-frequency equivalent circuit shown in Fig. 8.71(b). If $R_{\pi 1}$ is Thevenin’s equivalent resistance faced by $C_{\pi 1}$ with $C_{\mu 1}$, $C_{\mu 2}$, and $C_{\pi 2}$ open-circuited, the time constant $\tau_{\pi 1}$ is

$$\tau_{\pi 1} = R_{\pi 1}C_{\pi 1} = (r_{\pi 1} \parallel R_s \parallel R_{B1})C_{\pi 1}$$

$$= (1.4 \, \text{k}\Omega \parallel 200 \, \Omega \parallel 15 \, \text{k}\Omega) \times 15 \, \mu\text{F} = 2.6 \, \text{ns}$$

If $R_{\pi 2}$ is Thevenin’s equivalent resistance faced by $C_{\pi 2}$ with $C_{\mu 1}$, $C_{\mu 2}$, and $C_{\pi 1}$ open-circuited, the time constant $\tau_{\pi 2}$ can be found from

$$\tau_{\pi 2} = R_{\pi 2}C_{\pi 2} = \left[ r_{\pi 2} \parallel R_{E2} \parallel R_{C1} \parallel \frac{1}{g_{m2}} \right]C_{\pi 2}$$

$$= \left[ 1.4 \, \text{k}\Omega \parallel 9 \, \text{k}\Omega \parallel 15 \, \text{k}\Omega \parallel \frac{1000}{107.1 \, \Omega} \right] \times 15 \, \mu\text{F} = 0.14 \, \text{ns}$$

With $C_{\mu 2}$, $C_{\pi 1}$, and $C_{\pi 2}$ open-circuited, the effective load resistance of $C_{\mu 1}$ is

$$R_{L1(\text{eff})} = r_{\pi 2} \parallel R_{E2} \parallel R_{C1} \parallel \frac{1}{g_{m2}}$$

$$= 1.4 \, \text{k}\Omega \parallel 9 \, \text{k}\Omega \parallel 15 \, \text{k}\Omega \parallel \left( \frac{1000}{107.1 \, \Omega} \right) = 9.3 \, \Omega$$

and its effective input side resistance of $C_{\mu 1}$ is

$$R_{\pi 1(\text{eff})} = r_{\pi 1} \parallel R_s \parallel R_{B1}$$

$$= 1.4 \, \text{k}\Omega \parallel 200 \, \Omega \parallel 15 \, \text{k}\Omega = 173 \, \Omega$$

From Eq. (2.116), the time constant presented to $C_{\mu 1}$ is

$$\tau_{\mu 1} = [R_{L1(\text{eff})} + R_{\pi 1(\text{eff})}(1 + g_{m1}R_{L1(\text{eff})})]C_{\mu 1}$$

$$= [9.3 \, \Omega + 173 \, \Omega \times (1 + 71.4 \, \text{m}\Omega \times 9.3 \, \Omega)] \times 1 \, \mu\text{F} = 0.3 \, \text{ns}$$

With $C_{\mu 1}$, $C_{\pi 1}$, and $C_{\pi 2}$ open-circuited, the time constant presented to $C_{\mu 2}$ is

$$\tau_{\mu 2} = (R_L \parallel R_{C2})C_{\mu 2}$$

$$= (10 \, \text{k}\Omega \parallel 15 \, \text{k}\Omega) \times 1 \, \mu\text{F} = 6.0 \, \text{ns}$$


8.16 MOSFETs versus BJTs

An MOSFETs has the following advantages over a BJT:

1. It has an extremely high input resistance, on the order of megaohms.
2. It has no offset voltage when it is used as a switch, whereas a BJT requires a minimum base–emitter voltage \( V_{BE} \).
3. It is relatively immune to ionizing radiation, whereas a BJT is very sensitive because its beta value is particularly affected.
4. It is less “noisy” than a BJT and thus more suitable for input stages of low-level amplifiers. It is used extensively in FM receivers.
5. It provides better thermal stability than a BJT—that is, the parameters of MOSFETs are less sensitive to temperature changes.

MOSFETs have a smaller gain bandwidth than BJTs and are more susceptible to damage in handling. The gain bandwidth is the frequency at which the gain becomes unity.

8.17 Design of Amplifiers

When an amplifier is being analyzed, the components are specified; however, when an amplifier is being designed, the designer must select the values of the circuit components. The design task can be simplified if a simple transistor model is used to find approximate values of the components. After the initial design stage, the next step is to analyze the amplifier with these approximate values and to compare the performance parameters with the desired values. Often the specifications are not met, and it is necessary to modify the component values. An amplifier is normally specified by the input resistance \( R_i \),

\[
\text{From Eq. (2.112), the high cutoff frequency is}
\]

\[
f_H = \frac{1}{2\pi} \left[ \frac{1}{\tau_{\pi 1} + \tau_{\pi 2} + \tau_{\mu 1} + \tau_{\mu 2}} \right] = \frac{1}{2\pi} \left[ \frac{10^9}{2.6 + 14 + 0.3 + 6.0} \right] = 17.6 \text{ MHz}
\]

which is much higher than \( f_H = 694.4 \text{ kHz} \) for the CE-CE BJT amplifier in Example 8.13.
the output resistance $R_o$, and the voltage gain $A_v$. These specifications are normally defined by the following values:

- Source resistance $R_s$
- DC supply voltage $V_{CC}$ for BJTs
- Load resistance $R_L$
- Overall voltage gain $A_v (=v_L/v_s)$ (at a specified $R_L$)
- Input resistance at the base of the transistor $R_i$

After the specifications of an amplifier have been established, the next step is to decide on the type of transistor to be used. In the following analysis, we develop the necessary design conditions and the steps in meeting design specifications.

### 8.17.1 BJT Amplifier Design

Once a decision has been made to design a BJT amplifier, choose a suitable BJT and note its particular current gain $\beta_f (=\beta_f)$ and Early voltage $V_A$ (or assume a typical value of 200 V). Then choose a collector current $I_C$ at the $Q$-point. The manufacturer normally provides curves showing the variations in current gain $h_{fe}$ ($\beta_f$) against the collector current. $I_C$ may be chosen from the manufacturer’s data sheet so that the current gain $\beta_f$ is maximum. Depending on the type of transistor (npn or pnp), $V_{CC}$ and $I_C$ will have positive or negative values; the value of $V_A$ may be specified as a negative number. However, we use only the magnitudes of $V_{CC}$, $I_C$, and $V_A$.

We have noted that the technique of DC analysis differs from that of AC analysis. For DC analysis, the load line is set by the DC resistance $R_{dc}$. That is,

$$R_{dc} = \begin{cases} R_C + R_E & \text{for the CE amplifier of Fig. 8.28(a)} \\ R_E & \text{for the CC amplifier of Fig. 8.34(a)} \end{cases} \tag{8.213}$$

For AC analysis, the load line is set by the AC resistance. That is,

$$R_{ac} = \begin{cases} R_C \parallel R_L & \text{for the CE amplifier of Fig. 8.28(a)} \\ R_E \parallel R_L & \text{for the CC amplifier of Fig. 8.34(a)} \end{cases} \tag{8.214}$$

Under the no-load condition, the load resistance $R_L$ is disconnected; the AC resistance $R_{ac}$ equals $R_C$. Thus, there are two load lines that must be considered in designing an amplifier circuit. So far, we have considered the DC load line only while designing a biasing circuit. The AC and DC load lines for CE amplifiers are shown in Fig. 8.72. The $Q$-point, which is specified for a zero AC input signal, lies on both the AC and the DC load lines. The AC load line passes through the $Q$-point and has a slope of $-1/R_{ac}$. The slope of the AC line is greater in magnitude than that of the DC line. The AC load line may be described by

$$i_C - I_C = \frac{-(v_{CE} - V_{CE})}{R_{ac}}$$

which gives

$$i_C = \frac{v_{CE}}{R_{ac}} + \left(\frac{V_{CE}}{R_{ac}} + I_C\right) \tag{8.215}$$

The maximum AC collector current $I_{C(max)}$, which occurs at $v_{CE} = 0$, can be found from Eq. (8.215):

$$I_{C(max)} = \frac{V_{CE}}{R_{ac}} + I_C \tag{8.216}$$
An amplifier should be designed to accommodate the maximum AC swing along the AC load line, for which $i_{C\text{(max)}}$ must be twice the value of $I_C$, that is,

$$i_{C\text{(max)}} = 2I_C = \frac{V_{CE}}{R_{ac}} + I_C$$

which gives

$$I_C = \frac{V_{CE}}{R_{ac}}$$

(8.217)

Assuming $I_C \approx I_E$, Eq. (8.75) gives the DC load line

$$V_{CC} = V_{CE} + (R_C + R_E)I_C = V_{CE} + R_{dc}I_C$$

Substituting $V_{CE}$ from Eq. (8.217) into the above equation yields

$$V_{CC} = V_{CE} + R_{dc}I_C = R_{ac}I_C + R_{dc}I_C = (R_{ac} + R_{dc})I_C$$

which gives the collector biasing current $I_C$ as

$$I_C = \frac{V_{CC}}{R_{ac} + R_{dc}}$$

(8.218)

Thus, the Q-point is determined by both the AC and the DC resistances, which are dependent on $R_C$, $R_E$, and $R_L$. A higher voltage gain can be obtained at the expense of a low input resistance and a high output resistance. Therefore, BJT amplifiers are normally designed for a specified voltage gain or for a specified input resistance.

**Designing for Specified Voltage Gain**

When the voltage gain $A_v (=v_{L}/v_{s})$ of the amplifier is specified, generally the input resistance $R_i$ is not of major concern. Such an amplifier, shown in Fig. 8.29, acts as the middle stage of a multistage...
amplifier, providing as much gain as possible. The steps required to complete the design objectives are as follows:

**Step 1.** Calculate \( V_E = \frac{|V_{CC}|}{3} \).

**Step 2.** Calculate \( R_E = \frac{V_E}{I_E} = \frac{V_E \beta_F}{(1 + \beta_F)|I_C|} \).

**Step 3.** Calculate the voltage \( V_B \) at the transistor base: \( V_B = V_E + V_{BE} = V_E + 0.7 \).

**Step 4.** Calculate \( R_B = 0.1(1 + \beta_f)R_E \).

**Step 5.** Calculate the values of \( R_1 \) and \( R_2 \):

\[
R_1 = \frac{R_B |V_{CC}|}{|V_B|} \\
R_2 = \frac{R_B}{1 - |V_B/V_{CC}|}
\]

**Step 6.** Calculate the value of \( R_C \) for known values of \( R_L, R_E, V_{CC}, \) and \( I_C \). \( R_{ac} \) is the parallel combination of \( R_C \) and \( R_L \). From Eqs. (8.214) and (8.218), we get

\[
\frac{|V_{CC}|}{|I_C|} = R_{dc} + R_{ac} = R_E + R_C + \frac{R_C R_L}{R_C + R_L}
\]

**Step 7.** Calculate the values of \( r_\pi \) and \( r_o \).

\[
r_\pi = \frac{25.8 \text{ mV}}{|I_B|} = \frac{25.8 \text{ mV}}{|I_C|} \beta_f
\]

\[
r_o \approx \frac{|V_A|}{|I_C|}
\]

**Step 8.** As the first approximation, let the no-load voltage gain \( |A_{vo}| \) (for \( R_L = \infty \)) equal \( |A_v| \). From Eq. (8.92), the no-load voltage gain \( |A_{vo}| \) is given by

\[
|A_{vo}| = \frac{g_m r_\pi R_C}{r_\pi + (1 + \beta_f)R_{E1}} = \frac{\beta_F R_C}{R_x}
\]

from which the resistance \( R_x \) in Eq. (8.85) at the transistor base can be found:

\[
R_x = \frac{\beta_F R_C}{|A_{vo}|}
\]

**Step 9.** Calculate the required value of emitter resistance \( R_{E1} \) from

\[
R_{E1} = \frac{R_x - r_\pi}{1 + \beta_f}
\]

If \( R_{E1} < 0 \), the desired \( |A_v| \) is too large.

**Step 10.** Calculate the value of bypassed emitter resistance \( R_{E2} \):

\[
R_{E2} = R_E - R_{E1}
\]
If $R_{E2} < 0$, the desired $|A_v|$ is too small; choose a transistor of lower current gain $\beta_F$.

**Step 11.** Calculate the output resistance $R_o = R_C$.

**Step 12.** Calculate the voltage gain $A_v = v_L/v_s$:

$$|A_v| = \frac{A_{vo}R_I R_L}{(R_i + R_x)(R_L + R_o)}$$

**Step 13.** If the value of $|A_v|$ in step 12 is not greater than or equal to the desired absolute value of $A_v$, repeat steps 8 through 12 with progressively higher values of $|A_{vo}|$ until you obtain the desired value for overall voltage gain $A_v$ in step 12.

### Designing for Specified Input Resistance

When the input resistance $R_i$ of the amplifier is specified, generally the voltage gain $A_v (= v_L/v_s)$ is not of major concern. Such an amplifier normally acts as the input stage of a multistage amplifier. The first seven steps in completing the design objectives are the same as described above. The next two steps are as follows:

**Step 8.** Knowing, from Eq. (8.86), that the input resistance $R_i$ is given by

$$R_i = R_B \parallel R_x = \frac{R_B R_x}{R_B + R_x}$$

find the required value of resistance $R_x$ at the base of the transistor from

$$R_x = \frac{R_i}{1 - R_i/R_B} \quad \text{for } R_B \geq R_i$$

If $R_x < 0$, the desired $R_i$ is too high; choose a lower value of $R_i$ or a higher value of $R_B$ (by repeating steps 1 to 4 with a transistor of higher current gain $\beta_i$ and a lower collector current $I_C$).

**Step 9.** Calculate the required value of unbypassed emitter resistance $R_{E1}$ from

$$R_x = r_\pi + R_{E1}(1 + \beta_i) \quad \text{or} \quad R_{E1} = \frac{R_x - r_\pi}{1 + \beta_i}$$

If $R_{E1} < 0$, choose a transistor of higher current gain $\beta_F$ and lower biasing current $I_C$.

Steps 10 to 12 are the same as those used in designing for a specified voltage gain.

### KEY POINTS OF SECTION 8.17

- In general, designing involves decision making and an iterative process. The design steps developed in this section will be helpful in finding component values to satisfy specifications.
- Designing an amplifier requires prior knowledge of desired specifications, choice of a BJT, and choice of a $Q$-point.
- Once the type of transistor and the $Q$-point have been chosen, the next step is to choose the biasing circuit and find its component values.
- The small-signal parameters, which are calculated from the values of the $Q$-point, are then used to find the emitter (or source) resistance needed to obtain the desired voltage gain or input resistance.
Bipolar Junction Transistors and Amplifiers

Summary

Bipolar junction transistors (BJTs) are active devices, and they are of two types: npn and pnp. BJTs are current-controlled devices; the output depends on the input current. A BJT can operate in any one of three regions: the cutoff, active, or saturation region. The forward current gain $\beta_F$, which is a very important parameter, is the ratio of the collector current to the base current. The biasing circuit sets the operating point such that the effects of parameter variations are minimized and allows for the superposition of AC signals with minimum distortion. BJTs may be represented by linear or nonlinear models. The linear models, which give approximate results, are commonly used for initial design and analysis. The nonlinear models are normally used for computer-aided design and analysis, especially with PSpice/SPICE.

A common-emitter amplifier is used for voltage amplification. Emitter resistance increases input resistance, but it reduces voltage gain. A compromise is normally required between high input resistance and high voltage gain requirements. A common-collector amplifier, which is known as an emitter follower, offers a high input resistance and a low output resistance, with a gain approaching unity. An amplifier can have two load lines: an AC load line and a DC load line. The AC load line is affected by external load resistance. Designing an amplifier normally requires specifying the input resistance, the output resistance, and the voltage gain.

Bypass and coupling capacitors control the low 3-dB frequency; capacitors of the small-signal transistor models control the high 3-dB frequency. Analysis of low break frequencies can be simplified by the short-circuit method, in which the time constant due to one capacitor is determined by assuming that the other capacitors are effectively short-circuited. This method can be extended to the analysis of multistage amplifiers.

References


Review Questions

1. What are the types of BJTs?
2. What are the differences between npn- and pnp-type BJTs?
3. What are the possible regions of BJT operation?
4. What is a short-circuit amplification factor?
5. What is a forward amplification factor?
6. What are the characteristics of an active region?
7. What are the characteristics of a saturation region?
8. What is the purpose of biasing a BJT?
9. What is a load line?
10. What is the relationship between power dissipation and junction temperature?
11. What are the linear models of BJTs?
12. What is a transistor saturation current?
13. What are the characteristics of CC amplifiers?
14. What is the small-signal current gain of a BJT?
15. What is the Early voltage?
16. What is the small-signal output resistance of a BJT?
17. What is a load line?
18. What are the characteristics of a saturation region?
19. What are the characteristics of an active region?
20. What is the purpose of an emitter-bypassed capacitor?
21. What is a transistor saturation current?
22. What are the advantages of FETs over BJTs?
23. What is the transition frequency of a transistor?
24. What is a DC load line?
25. What is the transit time of a BJT?

Problems

The symbol \( \bigcirc \) indicates that a problem is a design problem. The symbol \( \bigcirc \) indicates that you can check the solution to a problem using PSpice/SPICE or Electronics Workbench.

8.3 Principles of BJT Operation

8.1 Calculate the width of the B-C depletion region of an \( npn \) transistor if the C-B voltage is \( V_{CB} = 20 \) V. The physical parameters are \( N_C = 10^{16} \text{ cm}^{-3}, N_B = 1 \times 10^{15} \text{ cm}^{-3}, T = 25^\circ \text{C}, w_B = 0.7 \mu\text{m}, e_s = 11.7 \times 8.85 \times 10^{-14}, q = 1.6 \times 10^{-19}, \) and \( n_i = 1.5 \times 10^{10} \).

8.2 Calculate the neutral base width of an \( npn \) transistor if the C-B voltage is \( V_{CB} = 12 \) V. The physical parameters are \( N_C = 10^{16} \text{ cm}^{-3}, N_B = 1 \times 10^{15} \text{ cm}^{-3}, T = 25^\circ \text{C}, w_B = 0.7 \mu\text{m}, e_s = 11.7 \times 8.85 \times 10^{-14}, q = 1.6 \times 10^{-19}, \) and \( n_i = 1.5 \times 10^{10} \).

8.3 Plot the width of the B-C depletion region of an \( npn \) transistor if the C-B voltage is varied from \( V_{CB} = 1 \) V to 20 V. The physical parameters are \( N_C = 10^{16} \text{ cm}^{-3}, N_B = 1 \times 10^{15} \text{ cm}^{-3}, T = 25^\circ \text{C}, w_B = 0.7 \mu\text{m}, e_s = 11.7 \times 8.85 \times 10^{-14}, q = 1.6 \times 10^{-19}, \) and \( n_i = 1.5 \times 10^{10} \).

8.4 Calculate the width of the B-C depletion region of an \( npn \) transistor if the C-B voltage is \( V_{CB} = 15 \) V. The physical parameters are \( N_C = N_B = 10^{16} \text{ cm}^{-3}, T = 25^\circ \text{C}, w_B = 0.7 \mu\text{m}, e_s = 11.7 \times 8.85 \times 10^{-14}, q = 1.6 \times 10^{-19}, \) and \( n_i = 1.5 \times 10^{10} \).

8.5 The physical parameters of an \( npn \) transistor are \( N_C = N_B = 10^{16} \text{ cm}^{-3}, T = 25^\circ \text{C}, w_B = 0.7 \mu\text{m}, e_s = 11.7 \times 8.85 \times 10^{-14}, q = 1.6 \times 10^{-19}, \) and \( n_i = 1.5 \times 10^{10} \). Determine the maximum C-B voltage to limit the width of the B-C region to \( \pm 10\% \) of the base width \( w_B \).

8.6 The physical parameters of an \( npn \) transistor are \( N_C = N_B = 10^{16} \text{ cm}^{-3}, T = 25^\circ \text{C}, w_B = 0.7 \mu\text{m}, e_s = 11.7 \times 8.85 \times 10^{-14}, q = 1.6 \times 10^{-19}, \) and \( n_i = 1.5 \times 10^{10} \). Determine the maximum C-B voltage to limit the width of the B-C region to \( \pm 15\% \) of the base width \( w_B \).
8.7 The physical parameters of an npn transistor are \( N_E = 10^{16} \text{cm}^{-3}, N_B = 1 \times 10^{15} \text{cm}^{-3}, n_i = 1.5 \times 10^{10} \text{cm}^{-3}, D_E = 20 \text{cm}^2/\text{s}, D_B = 10 \text{cm}^2/\text{s}, x_B = 0.7 \mu\text{m}, L_E = 0.013 \text{cm}, \tau_B = 8 \times 10^{-6} \text{s}, \text{ and } A_{BE} = 10^{-3} \text{cm}^2. \) Calculate (a) the collector reverse saturation current \( I_{SC}, \) (b) the current gain \( \beta_F, \) and (c) the base saturation current \( I_{SB}. \)

8.8 The physical parameters of an npn transistor are \( N_E = 10^{16} \text{cm}^{-3}, N_B = 1 \times 10^{15} \text{cm}^{-3}, n_i = 1.5 \times 10^{10} \text{cm}^{-3}, D_E = 20 \text{cm}^2/\text{s}, D_B = 10 \text{cm}^2/\text{s}, x_B = 0.7 \mu\text{m}, L_E = 0.013 \text{cm}, \tau_B = 8 \times 10^{-6} \text{s}, \text{ and } A_{BE} = 10^{-3} \text{cm}^2. \)

a. Calculate the emitter doping density to obtain a current gain \( \beta_F = 150. \)

b. Plot the current gain \( \beta_F \) against the doping ratio \( x = N_E/N_B = 1 \) to 10.

8.9 The physical parameters of an npn transistor are \( N_E = 10^{16} \text{cm}^{-3}, N_B = 1 \times 10^{15} \text{cm}^{-3}, n_i = 1.5 \times 10^{10} \text{cm}^{-3}, D_E = 20 \text{cm}^2/\text{s}, D_B = 10 \text{cm}^2/\text{s}, L_E = 0.013 \text{cm}, \tau_B = 8 \times 10^{-6} \text{s}, \text{ and } A_{BE} = 10^{-3} \text{cm}^2. \)

a. Calculate the base width \( x_B \) to obtain a current gain of \( \beta_F = 100. \)

b. Plot the current gain \( \beta_F \) against the base width \( x_B = 0.1 \mu\text{m} \) to 1 \( \mu\text{m}. \)

8.10 The physical parameters of an npn transistor are \( N_E = 10^{16} \text{cm}^{-3}, N_B = 1 \times 10^{15} \text{cm}^{-3}, n_i = 1.5 \times 10^{10} \text{cm}^{-3}, D_E = 20 \text{cm}^2/\text{s}, D_B = 10 \text{cm}^2/\text{s}, x_B = 0.7 \mu\text{m}, L_E = 0.013 \text{cm}, \tau_B = 8 \times 10^{-6} \text{s}, \text{ and } A_{BE} = 10^{-3} \text{cm}^2. \) Calculate the B-E voltage to give a collector current of \( I_C = 1.2 \text{ mA} \) if \( V_T = 25.8 \text{ mV}, V_{CE} = 15 \text{ V}, \text{ and } |V_A| = 200 \text{ V}. \)

8.11 The physical parameters of a pnp transistor are \( N_E = 10^{15} \text{cm}^{-3}, N_B = 5 \times 10^{16} \text{cm}^{-3}, n_i = 1.5 \times 10^{10} \text{cm}^{-3}, D_E = 20 \text{cm}^2/\text{s}, D_B = 10 \text{cm}^2/\text{s}, x_B = 0.7 \mu\text{m}, L_E = 0.013 \text{cm}, \tau_B = 8 \times 10^{-6} \text{s}, \text{ and } A_{BE} = 10^{-3} \text{cm}^2. \) Calculate the E-B voltage to give a collector current of \( I_C = 1.5 \text{ mA} \) if \( V_T = 25.8 \text{ mV}, V_{CE} = -12 \text{ V}, \text{ and } |V_A| = 200 \text{ V}. \)

8.4 Input and Output Characteristics

8.12 The parameters of an npn transistor are \( \alpha_F = 0.9934 \) and \( I_B = 25 \mu\text{A}. \) Determine (a) the forward current gain \( \beta_F, \) (b) the collector current \( i_C, \) and (c) the emitter current \( i_E. \)

8.13 The parameters of the transistor in Fig. P8.13 are \( \beta_F = 150, V_{BE} = 0.7 \text{ V}, \text{ and } V_{CE(sat)} = 0.3 \text{ V}. \) If \( R_C = 1.5 \text{ k}\Omega, \) determine the critical value of \( R_B \) so that the transistor operates (a) in the active (amplifier) region and (b) in the saturation region.

---

**FIGURE P8.13**

![Bipolar Junction Transistors and Amplifiers](image)

8.14 The parameters of the transistor in Fig. P8.14 are \( \beta_F = 150, V_{BE} = 0.7 \text{ V}, \text{ and } V_{CE(sat)} = 0.3 \text{ V}. \) If \( R_C = 1.5 \text{ k}\Omega \) and \( R_E = 500 \text{ } \Omega, \) determine the critical value of \( R_B \) so that the transistor operates (a) in the active (amplifier) region and (b) in the saturation region.
8.15 The parameters of the transistor in Fig. P8.15 are $\beta_F = 150$, $V_{BE} = 0.7$ V, and $V_{CE(sat)} = 0.3$ V. If $R_C = 1.5$ k$\Omega$, determine the critical value of $R_B$ so that the transistor operates (a) in the active (amplifier) region and (b) in the saturation region.

8.7 DC Biasing of Bipolar Junction Transistors

8.16 The parameters of the transistor in Fig. P8.16 are $\beta_F = 150$, $V_{BE} = 0.7$ V, and $V_{CE(sat)} = 0.3$ V. If $V_{CC} = 12$ V, $V_{EE} = 12$ V, and $R_C = 1.5$ k$\Omega$, determine (a) the base current $I_B$, (b) the collector current $I_C$, and (c) the C-E voltage $V_{CE}$.

8.17 The parameters of the transistor in Fig. P8.17 are $\beta_F = 150$, $V_{BE} = 0.7$ V, and $V_{CE(sat)} = 0.3$ V. If $V_{CC} = 12$ V, $R_B = 200$ k$\Omega$, $R_C = 1.5$ k$\Omega$, and $R_E = 500$ $\Omega$, determine (a) the base current $I_B$, (b) the collector current $I_C$, and (c) the C-E voltage $V_{CE}$. 
8.18 The parameters of the transistor in Fig. P8.18 are $\beta_F = 150$, $V_{BE} = 0.7$ V, and $V_{CE(sat)} = 0.3$ V. If $V_{CC} = 15$ V, $V_B = 5$ V, $R_B = 200$ k$\Omega$, $R_C = 1.5$ k$\Omega$, and $R_E = 500$ $\Omega$, determine (a) the base current $I_B$, (b) the collector current $I_C$, and (c) the C-E voltage $V_{CE}$.

8.19 The parameters of the pnp BJT circuit in Fig. P8.19 are $R_C = 10$ k$\Omega$, $R_E = 1$ k$\Omega$, $V_{CC} = 15$ V, $V_{EE} = 5$ V, $V_{EB} = 0.6$ V, and $\alpha_F = 0.992$. Calculate $I_B$, $I_C$, $I_E$, $V_{CE}$, and $V_{CB}$ at the $Q$-point.

8.20 The parameters of the npn transistor circuit in Fig. P8.20 are $R_1 = 100$ k$\Omega$, $R_C = 1$ k$\Omega$, $R_E = 200$ $\Omega$, $V_{BE} = 0.7$ V, and $V_{CC} = 12$ V.

a. Calculate $I_B$, $I_C$, $I_E$, and $V_{CE}$ at the operating point if $\beta_F = 50$ and if $\beta_F = 250$.

b. Repeat part (a) if $R_E = 0$. 
8.21 The parameters of the transistor circuit in Fig. P8.21 are $R_1 = 10 \, \text{k}\Omega$, $R_C = 1 \, \text{k}\Omega$, $R_E = 200 \, \Omega$, $V_{BE} = 0.7 \, \text{V}$, and $V_{CC} = 12 \, \text{V}$.

**a.** Calculate $I_B$, $I_C$, $I_E$, and $V_{CE}$ at the $Q$-point if $\beta_F = 50$ and if $\beta_F = 250$.

**b.** Repeat part (a) if $R_E = 0$.

**FIGURE P8.21**

8.22 Design a biasing circuit as shown in Fig. 8.20(a). Calculate the values and power ratings of $R_E$, $R_C$, $R_1$, and $R_2$ and the total power dissipation $P_T$ of the circuit. The power supply is $V_{CC} = 30 \, \text{V}$. The quiescent values are $I_C = 2 \, \text{mA}$ and $V_{CE} = 12.6 \, \text{V}$. The nominal value of $\beta_F$ is 50. Assume $V_{BE} = 0.5 \, \text{V}$ and $r_E = \infty$.

8.23 The $n$-type $p$-type transistor circuit of Fig. P8.23 has $V_{BE} = 0.5 \, \text{V}$ and $\beta_F = 80$. Determine the value of $R_1$ that gives $I_C = 4 \, \text{mA}$ and the corresponding value of $V_{CE}$.

**FIGURE P8.23**
8.24 The _pnp_ transistor circuit of Fig. P8.24 has \( \beta_T = 100 \) and \( V_{EB} = 0.7 \) V. Calculate \( I_C \) and \( V_{CE} \).

**FIGURE P8.24**

![Diagram of a transistor circuit](image)

8.8–8.10 Common-Emitter Amplifiers, Emitter Followers, and Common-Base Amplifiers

8.25 The parameters of the amplifier circuit in Fig. P8.25 are \( V_{CC} = 5 \) V, \( R_C = 500 \) Ω, \( R_1 = 6.5 \) kΩ, \( R_2 = 2.5 \) kΩ, \( R_E = 450 \) Ω, \( R_s = 500 \) Ω, \( R_L = 5 \) kΩ, and \( C_1 = C_2 = \infty \). Assume \( \beta_T = 100 \) and \( V_A = 200 \) V.

**FIGURE P8.25**

![Diagram of an amplifier circuit](image)

a. Find the _Q_-point defined by \( I_B \), \( I_C \), and \( V_{CE} \).

b. Calculate the small-signal parameters \( g_m \), \( r_\pi \), and \( r_o \) of the transistor.

c. Calculate the input resistance \( R_{in} = v_s / i_s \), the no-load voltage gain \( A_{vo} = v_o / v_b \), the output resistance \( R_o \), the overall voltage gain \( A_v = v_L / v_s \), the current gain \( A_i = i_o / i_s \), and the power gain \( A_p \).

d. Use PSpice/SPICE to plot the instantaneous values of \( v_L \), \( v_C \), \( v_B \), \( i_C \), and \( i_B \).

8.26 The BJT amplifier of Fig. 8.23 has \( R_1 = 5.5 \) kΩ, \( R_2 = 1.5 \) kΩ, \( R_C = 1.5 \) kΩ, \( R_E = 150 \) Ω, \( R_L = 5 \) kΩ, \( R_s = 200 \) Ω, and \( V_{CC} = 18 \) V. Assume \( \beta_T = 100 \) and \( V_A = 200 \) V.

a. Find the _Q_-point defined by \( I_B \), \( I_C \), and \( V_{CE} \).

b. Calculate the small-signal parameters \( g_m \), \( r_\pi \), and \( r_o \) of the transistor.

c. Calculate the input resistance \( R_{in} = v_s / i_s \), the no-load voltage gain \( A_{vo} = v_o / v_b \), the output resistance \( R_o \), the overall voltage gain \( A_v = v_L / v_s \), the current gain \( A_i = i_o / i_s \), and the power gain \( A_p \).

d. Use PSpice/SPICE to plot the instantaneous values of \( v_L \), \( v_C \), \( v_B \), \( i_C \), and \( i_B \).
8.27 The parameters of the CE amplifier with an active current source as shown in Fig. 8.24(a) are \( \beta_F = 173 \), \( I_S = 3.295 \times 10^{-14} \) A, and \( V_A = 200 \) V. The DC power supply is \( V_{CC} = 15 \) V. Assume that \( V_{BE} = 0.7 \) V. If \( R = 14.3 \) k\( \Omega \), (a) determine the operating collector current \( I_C \), (b) determine the small-signal no-load voltage gain \( A_{vo} \), and (c) use PSpice/SPICE to verify your results.

8.28 Design a CE amplifier of Fig. P8.28 with an active current source. Use transistor Q2N2222 and Q2N2907 whose nominal \( \beta_F = 173 \), \( I_S = 3.295 \times 10^{-14} \) A, and \( V_A = 100 \) V. The operating collector current is set at \( I_C = 1 \) mA. The DC power supply is \( V_{CC} = 15 \) V. Assume that \( V_{BE} = 0.7 \) V.

8.29 The parameters of the CE amplifier with an active current source as shown in Fig. P8.28 are \( \beta_F = 173 \), \( I_S = 3.295 \times 10^{-14} \) A, and \( V_A = 200 \) V. The DC power supply is \( V_{CC} = 15 \) V. Assume that \( V_{BE} = 0.7 \) V. If \( R = 14.3 \) k\( \Omega \), (a) determine the operating collector current \( I_C \), (b) determine the small-signal no-load voltage gain \( A_{vo} \), and (c) use PSpice/SPICE to verify your results.

8.30 The emitter follower of Fig. 8.34(a) has \( R_B = 74 \) k\( \Omega \), \( R_E = 750 \) \( \Omega \), \( R_L = 5 \) k\( \Omega \), \( R_s = 200 \) \( \Omega \), \( V_{CC} = 18 \) V, and \( V_{BE} = 0.7 \) V. Assume \( \beta_F = 100 \) and \( V_A = 200 \) V.

(a) Find the \( Q \)-point defined by \( I_B \), \( I_C \), and \( V_{CE} \).
(b) Calculate the small-signal parameters \( g_m \), \( r_m \), and \( r_o \) of the transistor.
(c) Calculate the input resistance \( R_{in} = v_s / i_s \), the no-load voltage gain \( A_{vo} = v_o / v_s \), the output resistance \( R_o \), the overall voltage gain \( A_v = v_L / v_s \), the current gain \( A_i = i_o / i_s \), and the power gain \( A_p \).
(d) Use PSpice/SPICE to plot the instantaneous values of \( v_L \), \( v_B \), \( i_E \), and \( i_B \).

8.31 The emitter follower of Fig. P8.31 has \( R_1 = 100 \) k\( \Omega \), \( R_2 = 150 \) k\( \Omega \), \( R_E = 750 \) \( \Omega \), \( R_L = 20 \) k\( \Omega \), \( R_s = 200 \) \( \Omega \), \( V_{CC} = 15 \) V, and \( V_{BE} = 0.7 \) V. Assume that \( \beta_F = 100 \) and \( V_A = 200 \) V.
a. Find the $Q$-point defined by $I_B$, $I_C$, and $V_{CE}$.

b. Calculate the small-signal parameters of the transistor $g_m$, $r_{\pi}$, and $r_o$.

c. Calculate (i) the input resistance $R_{in}$, (ii) the no-load voltage gain $A_o (=v_o/v_b)$, (iii) the output resistance $R_o$, (iv) the overall voltage gain $A_v (=v_L/v_s)$, (v) the current gain $A_i$, and (vi) the power gain $A_p$.

8.32 An emitter follower is biased by a transistor current source, as shown in Fig. P8.32(a). Assume all transistors are identical, with $\beta_F = 100$, $V_{BE} = 0.7 \, \text{V}$, and $V_A = 200 \, \text{V}$, and assume $R = 5 \, \text{k}\Omega$ and $R_L = 1 \, \text{k}\Omega$.

a. Find the equivalent biasing current $I_O$ and resistance $r_o$, as shown in Fig. P8.32(b).

b. Calculate the small-signal input resistance $R_i$ and the output resistance $R_o$ of the emitter follower.

8.33 The biasing current for the emitter follower in Fig. P8.32(a) can be generated by the circuits in Fig. P8.33. Find the equivalent biasing current $I_O$ and resistance $r_o$. Assume a transistor with $\beta_F = 100$, $V_{BE} = 0.7 \, \text{V}$, and $V_A = 200 \, \text{V}$ and a diode drop of $V_D = 0.7 \, \text{V}$.
8.11–8.12 Multistage Amplifiers and the Darlington Pair Transistor

8.34 The cascoded amplifier circuit shown in Fig. 8.43(a) has $R_{B1} = R_{B2} = R_{B3} = 50 \, \text{k}\Omega$ and $R_C = R_E = 1 \, \text{k}\Omega$. Assume that the coupling capacitances are large tending to infinity and identical transistors of $\beta_F = 173$, $V_A = 200 \, \text{V}$, and $V_{BE1} = V_{BE2} = 0.7 \, \text{V}$.

a. Determine the biasing voltages and currents $I_{B1}, I_{C1}, V_{CE1}, I_{B2}, I_{C2}$, and $V_{CE2}$.

b. Use PSpice/SPICE to verify your results.

8.35 The cascoded amplifier circuit shown in Fig. 8.43(a) has $R_{B1} = R_{B2} = R_{B3} = 50 \, \text{k}\Omega$ and $R_C = R_E = 1 \, \text{k}\Omega$. Assume that the coupling capacitances are large tending to infinity and identical transistors of $\beta_F = 173$, $V_A = 200 \, \text{V}$, and $V_{BE1} = V_{BE2} = 0.7 \, \text{V}$.

a. Determine the small-signal voltage gain $A_v$.

b. Use PSpice/SPICE to verify your results.

c. Design an active biasing circuit using BJTs for the cascoded amplifier circuit shown in Fig. 8.43(a) to replace $R_C$ by producing a DC biasing current of $I_{C2} = 1 \, \text{mA}$. Assume that the coupling capacitances are large tending to infinity and identical transistors of $\beta_F = 173$, $V_A = 200 \, \text{V}$, and $V_{BE1} = V_{BE2} = 0.7 \, \text{V}$. If $R_E = 500 \, \Omega$, (a) determine the small-signal voltage gain $A_v$ and (b) use PSpice/SPICE to verify your results.

d. The collector resistance $R_C$ of the cascoded amplifier circuit shown in Fig. 8.43(a) is replaced by an active current source of $I_{C2} = 1 \, \text{mA}$ at an output resistance $r_o = 20 \, \text{k}\Omega$. Assume that the coupling capacitances are large tending to infinity and identical transistors of $\beta_F = 173$, $V_A = 200 \, \text{V}$, and $V_{BE1} = V_{BE2} = 0.7 \, \text{V}$. If $R_E = 500 \, \Omega$, (a) determine the small-signal voltage gain $A_v$ and (b) use PSpice/SPICE to verify your results.

e. The Darlington pair shown in Fig. 8.44(a) is biased in such a way that the collector bias current $I_{C2}$ of $Q_2$ is $400 \, \mu\text{A}$. The current gains of the two transistors are the same, $\beta_{F1} = \beta_{F2} = 80$, with an Early voltage $V_A = 50 \, \text{V}$. Calculate (a) the effective input resistance $r_m$, (b) the effective transconductance $g_m$, (c) the effective current gain $\beta$, and (d) the effective output resistance $r_o$.

f. The parameters of the amplifier circuit shown in Fig. P8.40 are $V_{CC} = 15 \, \text{V}$, $R_{C1} = R_{C2} = 1 \, \text{k}\Omega$, $R_1 = R_3 = 65 \, \text{k}\Omega$, $R_2 = R_4 = 25 \, \text{k}\Omega$, $R_{E1} = R_{E2} = 400 \, \Omega$, $R_L = 20 \, \text{k}\Omega$, and $C_1 = C_2 = C_3 = C_E \approx \infty$. Assume, identical transistors of $\beta_F = 100$ and $V_A = 200 \, \text{V}$. Calculate (a) the input resistance $R_i$, (b) the no-load voltage gain $A_{v0}(=v_o/v_i)$, (c) the output resistance $R_o$, (d) the overall voltage gain $A_v(=v_o/v_i)$, (e) the current gain $A_i$, and (f) the power gain $A_p$.

FIGURE P8.40

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
8.41 The parameters of the amplifier circuit shown in Fig. P8.41 are $V_{CC} = 15$ V, $R_{C1} = R_{C2} = 1$ kΩ, $R_1 = R_3 = 130$ kΩ, $R_2 = R_4 = 50$ kΩ, $R_{E1} = R_{E2} = 500$ Ω, $R_L = 20$ kΩ, and $C_1 = C_2 = C_3 = C_{E1} = C_{E2} = \infty$. Assume identical transistors of $\beta = 100$ and $V_A = 200$ V. Calculate (a) the input resistance $R_i$, (b) the no-load voltage gain $A_{vo} = v_o/v_s$, (c) the output resistance $R_o$, (d) the overall voltage gain $A_v = v_L/v_s$, (e) the current gain $A_i$, and (f) the power gain $A_p$.

![FIGURE P8.41](image)

8.42 The parameters of the amplifier circuit shown in Fig. P8.42 are $V_{CC} = 15$ V, $R_{C1} = R_{C2} = 1$ kΩ, $R_1 = 65$ kΩ, $R_2 = 25$ kΩ, $R_E = 500$ Ω, $R_L = 20$ kΩ, and $C_1 = C_2 = C_E = \infty$. Assume identical transistors of $\beta = 100$ and $V_A = 200$ V. Calculate (a) the input resistance $R_i$, (b) the no-load voltage gain $A_{vo} = v_o/v_s$, (c) the output resistance $R_o$, (d) the overall voltage gain $A_v = v_L/v_s$, (e) the current gain $A_i$, and (f) the power gain $A_p$.

![FIGURE P8.42](image)

8.43 The parameters of the amplifier circuit shown in Fig. P8.43 are $V_{CC} = 15$ V, $R_1 = 65$ kΩ, $R_2 = 25$ kΩ, $R_E = 500$ Ω, $R_L = 20$ kΩ, and $C_1 = C_2 = \infty$. Assume identical transistors of $\beta = 100$ and $V_A = 200$ V. Calculate (a) the input resistance $R_i$, (b) the no-load voltage gain $A_{vo} = v_o/v_s$, (c) the output resistance $R_o$, (d) the overall voltage gain $A_v = v_L/v_s$, (e) the current gain $A_i$, and (f) the power gain $A_p$. 
8.13 DC Level Shifting and Amplifier

8.44 The potential level-shifting circuit as shown in Fig. 8.48(a) has $R_1 = 2 \, \text{k}\Omega$ and $R_2 = 4 \, \text{k}\Omega$. Determine the voltage shift $V_{sh}$ and the output voltage $V_o$. Assume $V_{BE} = 0.7 \, \text{V}$.

8.45 The potential level-shifting circuit as shown in Fig. 8.48(a) operates at a DC emitter current $I_E = 1 \, \text{mA}$, and the DC voltages are $V_{CE} = -V_{BE} = 15 \, \text{V}$. Determine the values of $R_1$ and $R_2$ to produce a voltage shift of $3 \, \text{V}$ at an output voltage $V_o = -7 \, \text{V}$.

8.46 Determine the current source $I_o$ needed and $R_1$ in Fig. 8.48(b) to produce a voltage shift of $V_{sh} = 4 \, \text{V}$ at an output voltage of $V_o = -8 \, \text{V}$. Assume $V_{BE} = 0.7 \, \text{V}$.

8.47 The BJT level-shifting amplifier as shown in Fig. 8.49(a) has $R = 20 \, \text{k}\Omega$, $R_A = 15 \, \text{k}\Omega$, $R_B = 5 \, \text{k}\Omega$, $R_x = R_y = 20 \, \text{k}\Omega$, and $R_L = 20 \, \text{k}\Omega$. Assume the bypass capacitance $C_A$ is large tending to infinity and all identical transistors of $\beta_F = 173$, $V_A = 200 \, \text{V}$, and $V_{BE} = 0.7 \, \text{V}$.

a. Determine the biasing collector currents $I_{C1}, I_{C2}, I_{C3}, I_{C4}, I_{C5}$, and $I_{C6}$.

b. Determine the DC voltage $V_{B7}$ at the collector of $Q_6$.

c. Use PSpice/SPICE to verify your results.

8.48 The BJT level-shifting amplifier as shown in Fig. 8.49(a) has $R = 20 \, \text{k}\Omega$, $R_A = 15 \, \text{k}\Omega$, $R_B = 5 \, \text{k}\Omega$, $R_x = R_y = 20 \, \text{k}\Omega$, and $R_L = 20 \, \text{k}\Omega$. Assume the bypass capacitance $C$ is large tending to infinity and all identical transistors of $\beta_F = 173$, $V_A = 200 \, \text{V}$, and $V_{BE} = 0.7 \, \text{V}$. (a) Determine the small-signal voltage gain $A_{vo}$ and (b) use PSpice/SPICE to verify your results.

8.49 Design a BJT level-shifting amplifier as shown in Fig. 8.49(a) to produce a voltage gain of $A_{vo} = 100 \, \text{V/V}$ at a DC input signal of $v_s = 1 \, \text{mV}$. Use identical transistors of $\beta_F = 173$, $V_A = 200 \, \text{V}$, and $V_{BE} = 0.7 \, \text{V}$.

8.14 Frequency Model and Response of Bipolar Junction Transistors

8.50 An nnp transistor of type 2N3904 is biased at $I_C = 20 \, \text{mA}$, $V_{CE} = 5 \, \text{V}$, $V_{BE} = 0.7 \, \text{V}$, and $V_{CS} = V_C = 7 \, \text{V}$.

The parameters of the transistor are as follows: $C_{je0} = 8 \, \text{pF}$ at $V_{BE} = 0.5 \, \text{V}$, $C_{je0} = 4 \, \text{pF}$ at $V_{CB} = 5 \, \text{V}$, $C_{se0} = 4 \, \text{pF}$ at $V_{CS} = 8 \, \text{V}$, $\beta_F = 100$, and $h_{oc} = 1/r_o = 5 \, \mu\Omega$ at $V_{CE} = 10 \, \text{V}$, $I_C = 10 \, \text{mA}$. The transition frequency is $f_T = 300 \, \text{MHz}$ at $V_{CE} = 20 \, \text{V}$, $I_C = 10 \, \text{mA}$. Assume $V_T = 25.8 \, \text{mV}$ and $V_{jc} = V_{jc} = V_{js} = 0.8 \, \text{V}$. The substrate is connected to the ground.

a. Calculate the small-signal capacitances of the high-frequency model in Fig. 8.52(b).

b. Find transition time $\tau_T$.

c. Calculate $f_T$ of the transistor at the operating point.

8.51 Repeat Prob. 8.50 for $I_C = 2 \, \text{mA}$, $V_{CE} = 4 \, \text{V}$, $V_{BE} = 0.7 \, \text{V}$, and $V_{CS} = V_C = 3 \, \text{V}$.
8.52 A pnp transistor of type 2N3905 is biased at $I_C = 50$ mA, $V_{CE} = -6$ V, $V_{BE} = -0.7$ V, and $V_{CS} = V_C = -10$ V. The parameters of the transistor are as follows: $C_{je0} = 10$ pF at $V_{BE} = 0.5$ V, $C_{j0} = 4$ pF at $V_{CS} = 8$ V, $\beta_I = 50$, and $h_{oe} = 1/ro = 5 \mu$Ω at $V_{CE} = 10$ V, $I_C = -1$ mA. The transition frequency is $f_T = 200$ MHz at $V_{CE} = -20$ V, $I_C = -10$ mA. Assume $V_T = 25.8$ mV and $V_{je} = V_{jc} = V_{js} = 0.8$ V. The substrate is connected to the ground.

a. Calculate the small-signal capacitances of the high-frequency model in Fig. 8.49(a).

b. Find transition time $\tau_T$.

c. Calculate $f_T$ of the transistor at the operating point.

8.53 Repeat Prob. 8.52 for $I_C = 5$ mA, $V_{CE} = 5$ V, $V_{BE} = 0.7$ V, and $V_{CS} = V_C = 6$ V.

8.15 Frequency Response of BJT Amplifiers

For Probs. 8.54–8.59 involving BJT amplifiers, use transistors whose parameters are $\beta_I = 100$, $C_{je} = 8$ pF at $V_{BE} = 0.5$ V, $C_{j0} = 4$ pF at $V_{CS} = 5$ V, $C_{s0} = 4$ pF at $V_{CS} = 8$ V, $\beta_I = 100$, $V_{jc} = V_{jk} = 0.8$ V, and $h_{oe} = 1/ro = 5 \mu$Ω at $V_{CE} = 10$ V. The transition frequency is $f_T = 300$ MHz at $V_{CE} = 20$ V, $I_C = 10$ mA. The substrate is connected to the ground. Assume $I_C = 5$ mA (unless specified), $V_{CC} = 15$ V, $V_{BE} = 0.7$ V, $R_s = 1$ kΩ, and $R_L = 10$ kΩ. Use PSpice/SPICE to check your design by plotting the frequency response and give an approximate cost estimate.

8.54 Design a CE amplifier as shown in Fig. P8.54 to give a passband gain of $40 \leq |A_{PB}| \leq 50$, a low 3-dB frequency of $f_L \leq 1$ kHz, and a high 3-dB frequency of $f_H = 50$ kHz.

8.55 Design a CB amplifier as shown in Fig. P8.55 to give a passband gain of $20 \leq |A_{PB}| \leq 30$, a low 3-dB frequency of $f_L \leq 1$ kHz, and a high 3-dB frequency of $f_H = 100$ kHz. Assume $R_s = 15$ kΩ and $R_L = 10$ kΩ.
8.56 Design a CE amplifier as shown in Fig. P8.56 to give a passband gain of $50 \leq |A_{PB}| \leq 60$, a low 3-dB frequency of $f_l \leq 1$ kHz, and a high 3-dB frequency of $f_H = 50$ kHz.

**FIGURE P8.56**

8.57 Design a CC-CE amplifier as shown in Fig. P8.57 to give a passband gain of $25 \leq |A_{PB}| \leq 35$, $Z_{in(mid)} \simeq 50$ kΩ, a low 3-dB frequency of $f_l \leq 5$ kHz, and a high 3-dB frequency of $f_H = 50$ kHz.

**FIGURE P8.57**

8.58 Design a CE-CC amplifier as shown in Fig. P8.58 to give a passband gain of $20 \leq |A_{PB}| \leq 30$, $Z_{i1(mid)} \leq 100$ Ω, a low 3-dB frequency of $f_l \leq 1$ kHz, and a high 3-dB frequency of $f_H = 100$ kHz.

**FIGURE P8.58**
8.59 For the CC-CE amplifier in Fig. P8.59, find the passband gain \(|A_{PB}|\), the low 3-dB frequency \(f_L\), and the high 3-dB frequency \(f_H\). Assume \(C_o = 1 \mu F\).

**FIGURE P8.59**

![Diagram of a CC-CE amplifier](image)

8.60 A two-stage amplifier is shown in Fig. P8.60. The parameters are \(R_s = 5 \, k\Omega\), \(R_{11} = 70 \, k\Omega\), \(R_{21} = 45 \, k\Omega\), \(R_C = 5 \, k\Omega\), \(R_E = 1 \, k\Omega\), \(R_{sf} = 2 \, k\Omega\), \(R_{12} = 1 \, M\Omega\), \(R_{22} = 2 \, M\Omega\), \(R_D = 10 \, k\Omega\), \(R_L = 10 \, k\Omega\), \(r_{\pi 1} = 1.4 \, k\Omega\), \(\beta_{f1} = 50\), \(g_{m1} = 35.7 \, m\Omega\), \(g_{m2} = 107.1 \, m\Omega\), \(C_1 = 2 \, \mu F\), \(C_2 = 5 \, \mu F\), \(C_3 = 1 \, \mu F\), \(C_G = 1 \, \mu F\), \(C_E = 10 \, \mu F\), \(C_{\pi} = 15 \, pF\), \(C_{gd} = 2 \, pF\), and \(C_{gs} = 5 \, pF\). Calculate the low 3-dB frequency \(f_L\) and the high cut-off frequency \(f_H\).

**FIGURE P8.60**

![Diagram of a two-stage amplifier](image)

8.61 A two-stage amplifier is shown in Fig. P8.61. The parameters are \(R_s = 500 \, \Omega\), \(R_B = 47 \, k\Omega\), \(R_C = 10 \, k\Omega\), \(R_L = 10 \, k\Omega\), \(r_{\pi 1} = r_{\pi 2} = 1.4 \, k\Omega\), \(\beta_{f1} = \beta_{f2} = 150\), \(C_1 = 10 \, \mu F\), \(C_2 = 10 \, \mu F\), \(C_{pi 1} = C_{pi 2} = 15 \, pF\), and \(C_{mu 1} = C_{mu 2} = 15 \, pF\). Calculate the low 3-dB frequency \(f_L\) and the high cutoff frequency \(f_H\).

**FIGURE P8.61**

![Diagram of a two-stage amplifier](image)
8.17 Design of Amplifiers

8.62 Design a CE amplifier as shown in Fig. 8.29 to give a voltage gain of \( A_v = v_L/v_s = -20 \). Assume \( \beta_f = \beta_F = 100, V_{BE} = 0.7 \, \text{V}, V_A = 200 \, \text{V}, I_C = 10 \, \text{mA}, V_{CC} = 15 \, \text{V}, R_s = 500 \, \Omega, \) and \( R_L = 20 \, \text{kΩ} \).

8.63 Design a CE amplifier as shown in Fig. 8.29 to give an input resistance of \( R_i = v_b/i_s = 4 \, \text{kΩ} \). Assume \( \beta_f = \beta_F = 100, V_{BE} = 0.7 \, \text{V}, V_A = 200 \, \text{V}, I_C = 5 \, \text{mA}, V_{CC} = 15 \, \text{V}, R_s = 250 \, \Omega, \) and \( R_L = 10 \, \text{kΩ} \).

8.64 a. Design a CE amplifier as shown in Fig. 8.29 to give a voltage gain of \( A_v = v_L/v_s = -25 \). Assume \( \beta_f = \beta_F = 150, V_{BE} = 0.7 \, \text{V}, V_A = 200 \, \text{V}, I_C = 15 \, \text{mA}, V_{CC} = 18 \, \text{V}, R_s = 250 \, \Omega, \) and \( R_L = 5 \, \text{kΩ} \).

b. Use PSpice/SPICE to generate the small-signal parameters \( r_m, r_o, \) and \( r_\mu \) of the transistor and to verify your design.

8.65 a. Design a CE amplifier as shown in Fig. 8.29 to give an input resistance of \( R_i = v_b/i_s \geq 3.5 \, \text{kΩ} \). Assume \( \beta_f = \beta_F = 150, V_{BE} = 0.7 \, \text{V}, V_A = 200 \, \text{V}, I_C = 15 \, \text{mA}, V_{CC} = 18 \, \text{V}, \) and \( R_L = 5 \, \text{kΩ} \).

b. Use PSpice/SPICE to generate the small-signal parameters \( r_m, r_o, \) and \( r_\mu \) of the transistor and to verify your design.

8.66 Design an emitter follower as shown in Fig. 8.34(a). Assume \( \beta_f = \beta_F = 100, V_{BE} = 0.7 \, \text{V}, V_A = 200 \, \text{V}, I_C = 5 \, \text{mA}, R_s = 500 \, \Omega, V_{CC} = 15 \, \text{V}, R_L = 1 \, \text{kΩ}, \) and \( A_v \approx 1 \).

8.67 a. Design an emitter follower as shown in Fig. 8.34(a). Assume \( \beta_f = \beta_F = 150, V_{BE} = 0.7 \, \text{V}, V_A = 150 \, \text{V}, I_C = 10 \, \text{mA}, R_s = 500 \, \Omega, V_{CC} = 18 \, \text{V}, R_L = 5 \, \text{kΩ}, \) and \( A_v \approx 1 \).

b. Use PSpice/SPICE to generate the small-signal parameters \( r_m, r_o, \) and \( r_\mu \) of the transistor and to verify your design.

8.68 a. Design an emitter follower as shown in Fig. 8.34(d). Assume \( \beta_f = \beta_F = 150, V_{BE} = 0.7 \, \text{V}, V_A = 150 \, \text{V}, I_C = 10 \, \text{mA}, R_s = 500 \, \Omega, V_{CC} = 18 \, \text{V}, R_L = 5 \, \text{kΩ}, \) and \( A_v \approx 1 \).

b. Use PSpice/SPICE to generate the small-signal parameters \( r_m, r_o, \) and \( r_\mu \) of the transistor and to verify your design.

8.69 a. Design an emitter follower as shown in Fig. 8.34(d) to give an input resistance of \( R_i = v_b/i_s \geq 15 \, \text{kΩ} \). Assume \( \beta_f = \beta_F = 150, V_{BE} = 0.7 \, \text{V}, V_A = 150 \, \text{V}, I_C = 15 \, \text{mA}, R_s = 500 \, \Omega, V_{CC} = 18 \, \text{V}, R_L = 5 \, \text{kΩ}, \) and \( A_v \approx 1 \).

b. Use PSpice/SPICE to generate the small-signal parameters \( r_m, r_o, \) and \( r_\mu \) of the transistor and to verify your design.

8.70 Design a CE amplifier as shown in Fig. 8.24(a) with an active current source. Use transistors for which minimum \( \beta_f = 200, \) nominal \( \beta_F = 250, \) and \( V_A = 200 \, \text{V} \). The operating collector current is set at \( I_C = 1 \, \text{mA} \).

The DC power supply is \( V_{CC} = 10 \, \text{V} \). Assume \( V_{BE} = 0.7 \, \text{V} \).

8.71 The CE amplifier of Fig. 8.71(a) is biased by the current sources shown in parts (b), (c), (d), and (e). Determine the circuit parameters for each of the circuit sources to give \( I_O = 1 \, \text{mA} \). Assume a pnp transistor of \( \beta_f = 100, V_{BE} = -0.7 \, \text{V}, V_A = 200 \, \text{V}, \) and \( V_D = 0.7 \, \text{V} \). The DC power supply is \( V_{CC} = 12 \, \text{V} \). (Note: There is no unique solution.)
8.72 Design a common emitter–common base (CE-CB) amplifier as shown in Fig. P8.72 to give a voltage gain of $A_v = v_L/v_s = -12$. Assume $V_{CC} = 15\,\text{V}$ and $R_s = 250\,\Omega$. Use bipolar transistors of type 2N2222 or 2N3904.

8.73 Design a CE amplifier of Fig. 8.24(a) with an active current source. Use BJT transistors whose nominal $\beta_F = 173$, $I_e = 3.295 \times 10^{-14}$ A, and $V_A = 100$ V. The operating collector current is set at $I_C = 10$ mA. The DC power supply is $V_{CC} = 15$ V. Assume $V_{BE} = 0.7$ V. Use PSpice/SPICE to verify your results.
8.74 Design an emitter follower of Fig. 8.32(a) with an active current source. Use BJT transistors whose nominal \( \beta_F = 173, I_s = 3.295 \times 10^{-14} \text{ A}, \) and \( V_A = 100 \text{ V}. \) The operating collector current is set at \( I_C = 1 \text{ mA}. \) The DC power supply is \( V_{CC} = 15 \text{ V}. \) Assume \( V_{BE} = 0.7 \text{ V}. \) Use PSpice/SPICE to verify your results.

8.75 Design an active-biased amplifier as shown in Fig. P8.75 with an active current source by finding the values of \( R_B \) and \( R_{ref} \) to obtain biasing currents \( I_{C1} \approx I_{ref} = 1 \text{ mA}. \) Find the small-signal voltage gain. Assume identical BJT transistors whose nominal \( \beta_F = 173, I_s = 3.295 \times 10^{-14} \text{ A}, \) and \( V_A = 100 \text{ V}. \) The DC power supply is \( V_{CC} = 15 \text{ V}. \) Assume \( V_{EB3} = V_{EB5} = 0.7 \text{ V}, C_1 = C_2 \approx \infty, \) and \( R_L = 50 \text{ k}\Omega. \) Use PSpice/SPICE to verify your results.

**FIGURE P8.75**

8.76 Design an active-biased amplifier as shown in Fig. P8.76 with an active current source by finding the value of \( R_B \) to obtain biasing currents \( I_{C1} \approx I_{ref} = 1 \text{ mA}. \) Find the small-signal voltage gain. Assume identical BJT transistors whose nominal \( \beta_F = 173, I_s = 3.295 \times 10^{-14} \text{ A}, \) and \( V_A = 100 \text{ V}. \) The DC power supply is \( V_{CC} = 15 \text{ V}. \) Assume \( V_{BB} = 15 \text{ V} C_L \approx \infty \) and \( R_L = 50 \text{ k}\Omega. \) Use PSpice/SPICE to verify your results.

**FIGURE P8.76**
Design a multistage BJT amplifier to meet the following specifications: voltage gain $|A_v| = v_L/v_s = 600 \pm 5\%$ (with load); input resistance $R_i = v_o/i_o \approx 25 \, \text{k} \Omega$; output resistance $R_o \leq 300 \, \Omega$; load resistance $R_L = 25 \, \text{k} \Omega$; source resistance $R_s = 1 \, \text{k} \Omega$; DC supply $V_{CC} = 15 \, \text{V}$; input signal $v_s = 1 \, \text{mV}$ to $5 \, \text{mV}$ (peak sinusoidal), $1 \, \text{kHz}$. Use identical npn Q2N2222 transistors of $I_{C(max)} = 10 \, \text{mA}$, $\beta_F = 256$, $I_S = 14.34 \times 10^{-15} \, \text{A}$, and $V_A = 74 \, \text{V}$. (Hints: Use the first CE stage to meet the input resistance requirement, the third CC stage to meet the output resistance, and the middle CE stage to attain the remaining gain requirement. Set the drain-biasing current at $I_C \leq I_{C(max)/3}$.)

Design a multistage BJT amplifier to meet the following specifications: voltage gain $|A_v| = v_L/v_s = 600 \pm 5\%$ (with load); input resistance $R_i = v_o/i_o \approx 25 \, \text{k} \Omega$; output resistance $R_o \leq 300 \, \Omega$; load resistance $R_L = 25 \, \text{k} \Omega$; source resistance $R_s = 1 \, \text{k} \Omega$; DC supply $V_{CC} = 15 \, \text{V}$; input signal $v_s = 1 \, \text{mV}$ to $5 \, \text{mV}$ (peak sinusoidal), $1 \, \text{kHz}$. Use identical pnp Q2N2907A transistors of $I_{C(max)} = 10 \, \text{mA}$, $\beta_F = 231$, $I_S = 651 \times 10^{-18} \, \text{A}$, and $V_A = 115 \, \text{V}$. (Hints: Use the first CE stage to meet the input resistance requirement, the third CC stage to meet the output resistance, and the middle CE stage to attain the remaining gain requirement. Set the drain-biasing current at $I_C \leq I_{C(max)/3}$.)
Learning Outcomes

After completing this chapter, students should be able to do the following:

- Design, analyze, and evaluate different types of MOSFET constant-current sources for biasing MOSFET amplifiers.
- Design, analyze, and evaluate different types of BJT constant-current sources for biasing BJT amplifiers.
- Describe and analyze the characteristics of differential amplifiers and their DC and small-signal characteristics.
- Identify the parameters influencing the differential and the common-mode gains of differential amplifiers.
- Analyze and evaluate cascode-connected transistors in order to obtain higher differential voltage gains.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_d, A_c$</td>
<td>Small-signal differential and common-mode voltage gains</td>
</tr>
<tr>
<td>CMRR</td>
<td>Common-mode rejection ratio of an amplifier</td>
</tr>
<tr>
<td>$v_{id}, v_{ic}$</td>
<td>Small-signal differential and common-mode signal</td>
</tr>
<tr>
<td>$v_{o1}, v_{o2}$</td>
<td>Small-signal output voltages due to input voltages at inverting and noninverting terminals</td>
</tr>
</tbody>
</table>
9.1 Introduction

Differential amplifiers are commonly used as an input stage in various types of analog ICs, such as operational amplifiers, voltage comparators, voltage regulators, video amplifiers, power amplifiers, and balanced modulators and demodulators [1–3]. A differential amplifier is a very important transistor stage and determines many of the performance characteristics of an IC. In ICs, including differential amplifiers, it is unnecessary to bias transistors by setting the values of biasing resistors. Because of variations in resistor values, power supply, and temperature, the quiescent point of transistors changes. Transistors can be used to generate the characteristics of DC constant-current sources. Transistors can also be used to produce an output voltage source that is independent of its load or, equivalently, of the output current. This chapter covers the operation, analysis, and characteristics of differential amplifiers using BJTs and MOSFETs. It also covers active current sources and voltage sources.

9.2 Internal Structure of Differential Amplifiers

A differential amplifier acts as an input stage; its output voltage is proportional to the difference between its two input voltages \(v_1\) and \(v_2\). It has a high voltage gain and is directly DC coupled to the input voltages and the load. As we will see later in this chapter, the voltage gain of a differential amplifier depends directly on the output resistance of the current source acting as an active load.

In amplifiers with discrete components, passive components such as resistors and capacitors are less expensive than active devices such as transistors (i.e., MOSFETs and BJTs); thus, in multistage amplifiers, interstage coupling is accomplished with capacitors. However, in monolithic circuits, the die area is the principal determining cost factor. Capacitors of the values and sizes used in amplifiers made with discrete components cannot be included in ICs and must be external to the chip. But using external capacitors increases the pin count of the package and the cost of the IC. A DC-coupled circuit is used to eliminate capacitors. The cheapest component in an IC is the one that can be fabricated within the least area, usually the transistor. The optimal IC has as little resistance as possible and more transistors.

9.2.1 Characteristics of Differential Amplifiers

The differential stage can be represented by an equivalent amplifier, as shown in Fig. 9.1(a). If the two input voltages are equal, a differential amplifier gives an output voltage of almost zero. Its voltage gain is very large, so the input voltage is low, typically less than 50 mV. Thus, we can consider the input voltages as small signals with zero DC components. That is, \(v_{G1} = v_{g1}\) and \(v_{G2} = v_{g2}\).
Let us define a differential voltage \( v_{id} \) as
\[
(9.1)
\]
and a common-mode voltage \( v_{ic} \) as
\[
(9.2)
\]
From Eqs. (9.1) and (9.2), the two input voltages can be expressed as
\[
(9.3)
\]
and
\[
(9.4)
\]
By replacing the input signals with the equivalent differential and common-mode signals, we can represent the differential stage by an equivalent amplifier, as shown in Fig. 9.1(b). Let \( v_{o1} \) be the output voltage due to \( v_{g1} \) only, and let \( v_{o2} \) be the output voltage due to \( v_{g2} \) only. Then we can define a differential output voltage \( v_{od} \) as
\[
(9.5)
\]
and a common-mode output voltage as
\[
(9.6)
\]
From Eqs. (9.5) and (9.6), the two output voltages can be expressed as
\[
(9.7)
\]
and
\[
(9.8)
\]
Let $A_1$ be the voltage gain with an input voltage $v_{g1}$ at terminal 1 and terminal 2 grounded (i.e., $v_{g2} = 0$). Let $A_2$ be the voltage gain with an input voltage $v_{g2}$ at terminal 2 and terminal 1 grounded (i.e., $v_{g1} = 0$). The output voltage of the differential stage can be obtained by applying the superposition theorem; that is,

$$\quad v_o = A_1 v_{g1} + A_2 v_{g2} \quad (9.9)$$

Substituting Eqs. (9.3) and (9.4) into Eq. (9.9) yields

$$\quad v_o = A_1 \left( v_{ic} + \frac{v_{id}}{2} \right) + A_2 \left( v_{ic} - \frac{v_{id}}{2} \right)$$

$$\quad = \left( \frac{A_1 - A_2}{2} \right) v_{id} + (A_1 + A_2)v_{ic}$$

$$\quad = A_d v_{id} + A_c v_{ic} \quad (9.10)$$

$$\quad = A_d \left( v_{id} + \frac{A_c}{A_d} v_{ic} \right) \quad (9.11)$$

where $A_d = (A_1 - A_2)/2 = \text{differential voltage gain}$ and $A_c = A_1 + A_2 = \text{common-mode voltage gain}$.

The output voltage $v_o$ in Eq. (9.11) is due to a common-mode input voltage $v_{ic}$ and a differential input voltage $v_{id}$. If $A_d$ is much greater than $A_c$, the output voltage will be almost independent of the common-mode signal $v_{ic}$. A differential amplifier is expected to amplify the differential voltage as much as possible while rejecting (not amplifying) common-mode signals such as noise or other unwanted signals, which will be present in both terminals.

The ability of an amplifier to reject common-mode signals is defined by a performance criterion called the \textit{common-mode rejection ratio} (CMRR), which is defined by

$$\quad \text{CMRR} = \left| \frac{A_d}{A_c} \right| \quad (9.12)$$

$$\quad = 20 \log \left| \frac{A_d}{A_c} \right| \quad (\text{in dB}) \quad (9.13)$$

Substituting Eq. (9.12) into Eq. (9.11) gives the output voltage

$$\quad v_o = A_d \left( v_{id} + \frac{1}{\text{CMRR}} v_{ic} \right) \quad (9.14)$$

which shows that, to reduce the effect of $v_{ic}$ on the output voltage $v_o$—that is, to get $v_{oc}$ to approach zero—the value of CMRR must be very large, tending to infinity for an ideal amplifier. Thus, a differential amplifier should behave differently for common-mode and differential signals. The small-signal equivalent circuit is shown in Fig. 9.1(c). $R_{id}$ and $R_{ic}$ are the input resistances due to the differential and common-mode signals, respectively. The parameters of a differential amplifier are $A_d$ (ideally $\infty$), $A_c$ (0), CMRR ($\infty$), $R_{id}$ ($\infty$), and $R_{ic}$ ($\infty$). In the following sections, we will determine the circuit elements affecting these parameters.
9.2.2 Internal Structure of Differential Amplifiers

A differential amplifier [4] acts as an input stage of an op-amp circuit as shown in Fig. 9.2(a). It serves as a direct DC-coupled differential stage. It may be viewed as consisting of two common-source (CS) amplifiers as shown in Fig. 9.2(b). If the two amplifiers are identical—that is, $R_{D1} = R_{D2}$—their DC biasing drain currents will be the same, and $i_{D1} = i_{D2}$. The DC output voltages are given by

$$v_{o1} = V_{DD} - R_{D1}i_{D1} \quad \text{and} \quad v_{o2} = V_{DD} - R_{D2}i_{D2}$$

The output voltage between the two output terminals of $M_2$ and $M_1$ is

$$v_{o(21)} = v_{o2} - v_{o1} = V_{DD} - R_{D2}i_{D2} - V_{DD} + R_{D1}i_{D1} = R_{D1}(i_{D2} - i_{D1}) \quad (9.15)$$

Therefore, we get $v_{o(21)} = v_{o(12)} = 0$ for identical amplifiers of the condition $i_{D1} = i_{D2}$. The small-signal AC equivalent half-circuit is shown in Fig. 9.2(c), which, after replacing the transistor by its transconductance model, becomes as shown in Fig. 9.2(d). Assuming that the output of the transistor $M_1$ is $r_{o1} >> R_{D1}$ and using Eq. (7.70), we can find the small-signal common-mode voltage gain as given by

$$A_c = \frac{v_{oc}}{v_{ic}} = \frac{g_{m1}R_{D1}}{1 + 2g_{m1}R_{SS}} \quad (9.16)$$

The derivation of Eq. (9.16) assumes that no interaction exists between the two amplifiers, but there is an interaction between them. The source (or emitter) resistance, whose value is $R_3 \parallel R_d(=2R_{SS} \parallel 2R_{SS} = R_{SS})$. 

---

FIGURE 9.2 Basic structure of a differential amplifier
is shared by the two amplifiers as shown in Fig. 9.2(e). We will see in the following sections that Eq. (9.16) gives the expression for the common-mode voltage gain. We can find the differential gain if we consider a small increase in \( v_\text{g1} \) by a small amount \( v_{g1} \), which will cause a corresponding small increase in \( i_{\text{D1}} \) by an amount \( i_{d1} \) such that \( i_{D1} = I_{\text{D1}} + i_{d1} \). Since the biasing current through the source resistance \( R_{\text{SS}} \) must remain constant at \( I_0 = i_{d1} + i_{d2} = I_{\text{D1}} + I_{\text{D2}} \), the drain current \( i_{\text{D2}} \) must decrease by the same amount such that \( i_{D2} = I_{\text{D2}} - i_{d1} \). Therefore, we can find the small-signal voltage gain from Eq. (9.15) as

\[
\begin{align*}
  v_{o(21)} &= v_{o2} - v_{o1} = R_{\text{D1}}(i_{\text{D2}} - i_{d1} - I_{\text{D1}} - i_{d1}) = -2R_{\text{D1}}i_{d1} \\
  A_d &= \frac{v_{od}}{v_{id}} = -g_{m1}(r_{o1} \parallel R_{\text{D1}}) 
\end{align*}
\]

This does not depend on the source resistance \( R_{\text{SS}} \) because the voltage at the source terminals remains constant and does not change for a differential voltage. As a result, we can find the differential voltage gain \( A_d \) from Eq. (9.16) (for \( R_{\text{SS}} = 0 \)) as given by

\[
A_d = \frac{v_{od}}{v_{id}} = -g_{m1}(r_{o1} || R_{\text{D1}})
\]  

where \( r_{o1} \) is the output resistance of MOSFET M₁ or BJT Q₁.

To obtain a low CMRR of a differential amplifier, active current sources that offer high output resistances are generally used to replace \( R_0 \) and \( R_{\text{SS}} \). Therefore, a typical amplifier consists of three parts: (1) a direct-coupled differential pair consisting of transistors M₁ and M₂, (2) a DC biasing active current source or source resistance \( R_{\text{SS}} \), and (3) an active load or load resistances \( R_{\text{D1}} \) and \( R_{\text{D2}} \).

Thus, we can summarize the requirements for a low CMRR as follows:

- Both transistor output resistance \( r_{o1} \) and the load resistance \( R_{\text{D1}} \) should be large values for high differential-mode voltage gain \( A_d \).
- The source resistance \( R \) should be large values for a low common-mode voltage gain \( A_c \).

**KEY POINTS OF SECTION 9.2**

- A differential amplifier consists of an active biasing circuit, an active load, and a differential transistor pair.
- The performance of a differential amplifier is measured by a differential gain \( A_d \) that occurs in response to a differential voltage between two input terminals, a common-mode gain \( A_c \) that occurs in response to a voltage common to both input terminals, and a common-mode rejection ratio CMRR.
- The CMRR is the ratio of the differential gain to the common-mode gain, and it is a measure of the ability of an amplifier to amplify the differential signal and reject common-mode signals.

**9.3 MOSFET Current Sources**

In Secs. 7.8.1 and 8.7.1, we saw the effects of active current sources in increasing the voltage gain of an amplifier. Transistor current sources are widely used in analog ICs both as biasing elements and as loads for amplifying stages. Current sources are less sensitive to variations in DC power supply and temperature. Especially, for a small value of bias current, the current sources are more economical than resistors in terms of die area required for resistors. A current source can be designed by using either MOSFETs or BJTs [5].

MOSFET current sources are analogous to BJT current sources. We can convert a BJT current source to an equivalent MOSFET current source by assuming that the \( \beta_F \) of the BJTs is infinite. Since MOSFETs
do not draw any gate current, there is no need for gate current compensation as there is with BJTs. The choice of a BJT source or a MOSFET source generally depends on the type of integrated circuit involved (e.g., bipolar or MOS). BJT sources have some advantages over MOSFET sources, such as a wider compliance range and a higher output resistance. However, a higher output resistance can be obtained by cascode-like connections of MOSFETs.

Norton’s equivalent of a current source $I_O$ having an output resistance $R_o$ is shown in Fig. 9.3(a), and its Thevenin’s equivalent is in Fig. 9.3(b). An ideal current source should maintain a constant current at an infinite output resistance under all operating conditions. To achieve this goal, a number of current sources are developed. The commonly used current sources are the basic current source, which was applied in Secs. 7.8.1 and 8.7.1, the modified basic source, the cascoded current source, and the Wilson current source.

### 9.3.1 Basic Current Source

A basic MOSFET current source is shown in Fig. 9.3(c). The direction of the current flow in Fig. 9.3(c) is into the current source circuit; this type of constant-current source is often referred to as a current sink. Let us assume that the two transistors $M_1$ and $M_2$ are identical. Since their gate–source voltages are equal, their drain currents will be the same; that is, $I_{D1} = I_{D2}$. Thus, the output current $I_O$ ($=I_{D1}$) will be the mirror of $I_{D2}$. Since $V_{DS2} = V_{GS2}$, $M_2$ will be in saturation. Let $V_{T1}$ and $V_{T2}$ be the threshold voltages of $M_1$ and $M_2$, respectively. For $M_1$ also to be in saturation, $V_{DS1}$, which is greater than or equal to $(V_{GS2} - V_{T2})$,
must be greater than \((V_{GS1} - V_{t1})\). This condition reduces the voltage compliance range of the MOSFET current source and prevents it from operating from a low power supply (say, 1 V for a battery source). Using the equations that define the saturation region of MOS transistors, we can derive the drain current of an enhancement-type MOSFET:

\[
I_D = \frac{WK_x}{2L}(V_{GS} - V_t)^2(1 + \lambda V_{DS})
\]

\[
= \frac{K_m}{2}(V_{GS} - V_t)^2\left(1 + \frac{V_{DS}}{V_M}\right)
\]

\[
K_m = \frac{WK_x}{L} = 2\left(\frac{WK_x}{2L}\right) = 2K_n
\]

where \(K_n = \frac{WK_x}{2L}\) is also an MOS constant such that \(K_m = K_n\) for \(W = 2L\).

**NOTE**  \(V_t\) (lowercase subscript t) is the threshold voltage of a MOSFET, whereas \(V_T\) (capital subscript T) is the thermal voltage.

The output current, which is equal to the drain current of \(M_1\), is given by

\[
I_{D1} = I_O = K_{n1}(V_{GS1} - V_{t1})^2(1 + \lambda V_{DS1})
\]

Drain current \(I_{D2}\), which is equal to the reference current \(I_R\), is given by

\[
I_{D2} = I_R = K_{n2}(V_{GS2} - V_{t2})^2(1 + \lambda V_{DS2})
\]

In practice, all the components of the current source are processed on the same IC, and hence all physical parameters such as \(K_x\) and \(V_t\) are identical for both devices. Thus, the ratio of \(I_O\) to \(I_R\) is given by

\[
\frac{I_O}{I_R} = \frac{K_{n1}(1 + \lambda V_{DS1})}{K_{n2}(1 + \lambda V_{DS2})} = \frac{(W/L)_1}{(W/L)_2} \times \frac{(1 + \lambda V_{DS1})}{(1 + \lambda V_{DS2})}
\]

In practice, \(\lambda V_{DS} \ll 1\). Thus, Eq. (9.24) can be approximated by

\[
\frac{I_O}{I_R} = \frac{(W/L)_1}{(W/L)_2}
\]

By controlling the ratio \(W/L\), therefore, we can change the output current. The gate length \(L\) is usually held fixed, and the gate width \(W\) is varied from device to device to give the desired current ratio \(I_O/I_R\). By choosing identical transistors with \(W_1 = W_2\) and \(L_1 = L_2\), a designer can ensure that the output current \(I_O\) is almost equal to the reference current \(I_R\).

Since \(V_{GS2} = V_{DD} - R_1I_R\) and \(V_{DS2} = V_{GS2}\), the reference current \(I_R\) can be found approximately from Eq. (9.23); that is,

\[
I_R = I_{D2} = K_{n2}(V_{DD} - R_1I_R - V_{t2})^2
\]

can be solved for known values of \(V_{t2}, K_{n2}, V_{DD}\), and \(R_1\). The current source in Fig. 9.3(c) behaves as a current sink rather than a source. A current source equivalent to this current sink can be obtained by using \(p\)-channel MOS (PMOS) transistors as shown in Fig. 9.3(d).
9.3.2 Modified Basic Current Source

The reference resistance \( R_1 \) can be replaced by another MOSFET \( M_3 \), as shown in Fig. 9.4. Transistors \( M_2 \) and \( M_3 \) are used as voltage dividers to control the gate–source voltage of transistor \( M_1 \). If \( M_1 \) and \( M_2 \) are identical, the output current \( I_O \) exactly mirrors the drain current through \( M_2 \) and \( M_3 \). The value of \( V_{GS1} \) should be made as low as possible without taking \( M_1 \) out of the saturation region.

Since \( V_{GS1} = V_{GS2} \), the drain current \( I_{D1} (= I_O) \) is equal to the drain current \( I_{D2} \) and is given by

\[
I_{D1} = I_{D2} = I_R = K_{n1}(V_{GS1} - V_t)^2(1 + \lambda V_{GS1}) \tag{9.27}
\]

Since \( V_{DS2} = V_{GS2} \), the drain current \( I_{D2} \) is equal to the reference current \( I_R \) and is given by

\[
I_{D2} = I_R = K_{n2}(V_{GS2} - V_t)^2(1 + \lambda V_{GS2}) \tag{9.28}
\]

Since \( V_{GS3} = V_{DD} - V_{GS2} \), the drain current of \( M_3 \) is given by

\[
I_{D3} = I_R = K_{n3}(V_{GS3} - V_t)^2(1 + \lambda V_{DS3}) \tag{9.29}
\]

\[
= K_{n3}(V_{DD} - V_{GS2} - V_t)^2[1 + \lambda(V_{DD} - V_{GS2})]
\]

Since \( I_{D2} = I_{D3} = I_R \), from Eqs. (9.28) and (9.29) we get

\[
\frac{K_{n2}(V_{GS2} - V_t)^2(1 + \lambda V_{GS2})}{K_{n3}(V_{DD} - V_{GS2} - V_t)^2[1 + \lambda(V_{DD} - V_{GS2})]} = 1 \tag{9.30}
\]

Thus, by controlling the constants \( K_{n2} \) and \( K_{n3} \), we can obtain the desired value of \( V_{GS2} = V_{GS1} \), which will give the desired output current.

Output Resistance \( R_o \)

The small-signal drain-source resistance \( r_{ds1} \) can be derived from Eq. (9.20):

\[
\frac{1}{r_{ds1}} = \frac{\delta i_{D1}}{\delta v_{DS1}} = \frac{K_{n1}}{V_M}(V_G - V_0)^2 \approx \frac{I_{D1}}{V_M} \tag{9.31}
\]

Thus, the small-signal output resistance of the current source becomes

\[
R_o = r_{ds1} = \frac{V_M}{I_{D1}} = \frac{1}{\lambda I_{D1}} = r_{o1} \tag{9.32}
\]

which is relatively small. This small output resistance is a disadvantage of having only one MOSFET \( M_1 \) at the output side of a current source.
Designing a simple MOSFET current source  The parameters of the MOSFET current source in Fig. 9.4 are $V_t = 1$ V, $I_O = 50$ μA, $I_R = 40$ μA, $V_{DD} = 10$ V, and $V_M = 10$ V. All channel lengths are equal, $L_1 = L_2 = L_3 = L = 10$ μm, and $K = 20$ μA/V$^2$. Calculate the required values of (a) $K_{n1}$, $W_1$, (b) $K_{n2}$, $W_2$, (c) $K_{n3}$, $W_3$, and (d) the output resistance $R_o$ of the current source. Assume $V_{GS1} = 1.5$ V and $V_{DS1} = 5$ V.

**SOLUTION**

(a) From Eq. (9.22),

$$50 \times 10^{-6} = K_{n1}(1.5 - 1)^2 \left(1 + \frac{5}{10}\right)$$

which gives $K_{n1} = 133.3$ μA/V$^2$. From Eq. (9.21),

$$133.3 \times 10^{-6} = \frac{W_1 \times 20 \times 10^{-6}}{10 \times 10^{-6}}$$

which gives $W_1 = 66.65$ μm.

(b) $V_{DS2} = V_{GS2} = V_{GS1} = 1.5$ V. From Eq. (9.28),

$$40 \times 10^{-6} = K_{n2}(1.5 - 1)^2 \left(1 + \frac{1.5}{10}\right)$$

which gives $K_{n2} = 139.1$ μA/V$^2$. From Eq. (9.21),

$$139.1 \times 10^{-6} = \frac{W_2 \times 20 \times 10^{-6}}{10 \times 10^{-6}}$$

which gives $W_2 = 69.55$ μm.

(c) $V_{GS3} = V_{DS3} = V_{DD} - V_{DS2} = V_{DD} - V_{GS1} = 10 - 1.5 = 8.5$ V. From Eq. (9.29),

$$40 \times 10^{-6} = K_{n3}(8.5 - 1)^2 \left(1 + \frac{8.5}{10}\right)$$

which gives $K_{n3} = 0.384$ μA/V$^2$. From Eq. (9.21),

$$0.384 \times 10^{-6} = \frac{W_3 \times 20 \times 10^{-6}}{10 \times 10^{-6}}$$

which gives $W_3 = 0.192$ μm. In practice, because of manufacturing limitations, the minimum value of $L$ or $W$ is 10 μm. Since the value of $W_3$ is smaller than 10 μm, $V_M$ must be increased to make $W_3$ at least 10 μm.

(d) From Eq. (9.32), the output resistance $R_o$ is

$$R_o = r_{ds1} = \frac{V_M}{K_{n1}(V_{GS1} - V_t)^2} = \frac{10}{133.3 \times 10^{-6} \times (1.5 - 1)^2} = 300 \, \text{kΩ}$$


9.3.3 Multiple Current Sources

Since there is no gate current in a MOSFET, a number of MOSFETs can be connected to a single reference MOSFET $M_1$, as shown in Fig. 9.5. Different output currents can be obtained by suitably adjusting the width-to-length ratios of MOSFETs (i.e., $M_2$, $M_3$, and $M_4$). In practice, the gate length $L$ is normally kept constant, and the gate widths ($W$) of $M_2$, $M_3$, and $M_4$ are varied, to give the desired output currents. Thus, for equal $L$, Eq. (9.25) gives the relationship of the output currents $I_2$, $I_3$, and $I_4$ to $I_R$ as

$$I_2 = \left(\frac{W_2}{W_1}\right)I_R$$

$$I_3 = \left(\frac{W_3}{W_1}\right)I_R$$

$$I_4 = \left(\frac{W_4}{W_1}\right)I_R$$

**Output Resistance $R_o$**

The small-signal output resistance of the current source is

$$R_o = r_{ds1} = \frac{V_M}{I_{D1}} = \frac{1}{\lambda I_{D1}} = r_{o1}$$

9.3.4 Cascode Current Source

The output resistance of the basic current source in Fig. 9.3(c) can be increased by adding two more MOSFETs in a cascode-like connection, as shown in Fig. 9.6(a). The analysis of the circuit is straightforward. The small-signal circuit for finding the output resistance is shown in Fig. 9.6(b), and its small-signal equivalent is shown in Fig. 9.6(c); $r_{o2}$ is the output resistance of transistor $M_2$. Using KVL and the relation $v_{gs1} = -r_{o2}i_x$, we get

$$v_x = r_{o1}i_1 + r_{o2}i_x$$

$$= r_{o1}(i_x - g_m v_{gs1}) + r_{o2}i_x$$

$$= r_{o1}(i_x + g_m r_{o2}i_x) + r_{o2}i_x$$

...
which gives the output resistance \( R_o \) of the current source as
\[
R_o = r_{o1}(1 + g_{m1}r_{o2}) + r_{o2}
\] (9.33)

For identical transistors, \( r_{o1} = r_{o2} = r_o \), and \( R_o \) becomes
\[
R_o = r_o(2 + g_{m1}r_o)
\] (9.34)
\[
\approx g_{m1}r_o^2
\] (9.35)

Thus, the output resistance can be significantly increased, to a level comparable to that of a BJT source. However, the voltage compliance range will be reduced because of the two drain-source voltages in series (i.e., \( V_{DS1} + V_{DS2} \)).

### 9.3.5 Wilson Current Source

The MOSFET version of the Wilson current source is shown in Fig. 9.7(a). The equivalent circuit for finding the output resistance \( R_o \) is shown in Fig. 9.7(b). We have
\[
v_{gs3} = \frac{i_x}{g_{m2}}
\]
and
\[
v_{gs1} + v_{gs3} = -g_{m3}v_{gs3}r_{o3}
\]

which can be simplified to relate \( v_{gs1} \) to \( v_{gs3} \) and \( v_{gs2} \) by
\[
v_{gs1} = -(1 + g_{m3}r_{o3})v_{gs3} = -\frac{(1 + g_{m3}r_{o3})i_x}{g_{m2}}
\]

Applying KVL to Fig. 9.7(b), we get
\[
v_x = (i_x - g_{m1}v_{gs1})r_{o1} + \frac{i_x}{g_{m2}}
\]
which, after substituting for \( v_{gs1} \) and simplifying, gives the output resistance \( R_o \) as

\[
R_o = \frac{v_x}{i_x} = r_{o1} + \frac{1}{g_{m2}} + \frac{g_{m1}}{g_{m2}} r_{o1}(1 + g_{m3} r_{o3})
\]

\[
\approx r_{o1} + r_{o1}(1 + g_{m3} r_{o3}) \quad \text{(for} \; g_{m1} = g_{m2} = g_{m3})
\]

where \( r_{o2} \) and \( r_{o1} \) are the output resistances of transistors \( M_2 \) and \( M_1 \), respectively.

The problem with this circuit is that the drain voltages \( V_D1 \) and \( V_D3 \) of \( M_1 \) and \( M_3 \) are unequal. As a result, their drain currents \( I_{D1} \) and \( I_{D3} \) are also unequal. This problem can be solved by adding one diode-connected MOSFET, as shown in Fig. 9.7(c). This modification ensures that \( M_1 \) and \( M_3 \) have equal drain voltages and thus equal drain currents.

### 9.3.6 Design of Active Current Sources

The specifications for designing a current source will include the output current \( I_o \), the output resistance \( R_o \), and the DC supply voltage \( V_{DD} \). The design sequence is as follows:

**Step 1.** Determine the design specifications: output current and output resistance.

**Step 2.** Decide on the type of device to use—either BJTs or MOSFETs.

**Step 3.** Choose the circuit topology best suited to the specifications. Use simple transistor models for hand analysis to find the circuit-level solution, including component values and specifications of BJTs or MOSFETs.

**Step 4.** Use the standard values of components—for example, \( R_1 = 5.6 \, \text{M} \Omega \pm 5\% \) instead of \( 5.72 \, \text{M} \Omega \), \( R_2 = 30 \, \text{k} \Omega \pm 5\% \) instead of \( 29.3 \, \text{k} \Omega \), and \( R_3 = 27 \, \text{k} \Omega \pm 5\% \) instead of \( 27.5 \, \text{k} \Omega \). Evaluate your design and modify the values, if necessary.

**Step 5.** Use PSpice/SPICE verification, employing complex circuit models to calculate the worst-case results due to component and parameter variations. Modify your design, if necessary.
During the past few years, MOS technology has developed considerably. MOS transistors are being used increasingly in analog integrated circuits. It is relatively easy to connect MOS transistors in cascode form to control the drain current and give high output resistance. MOS differential pairs are the building blocks in MOS ICs.

### 9.4 MOS Differential Amplifiers

During the past few years, MOS technology has developed considerably. MOS transistors are being used increasingly in analog integrated circuits. It is relatively easy to connect MOS transistors in cascode form to control the drain current and give high output resistance. MOS differential pairs are the building blocks in MOS ICs.

#### 9.4.1 NMOS Differential Pair

An $n$-channel NMOS pair is shown in Fig. 9.8. The DC biasing is normally done by a MOS current source. Although a resistor $R_D$ is shown as the load in Fig. 9.8, a MOS active current mirror is normally used as the load.
DC Transfer Characteristics

The following analysis is performed for an \( n \)-channel MOSFET pair, but it is equally applicable to a \( p \)-channel pair with appropriate sign changes. The analysis can be simplified by making the following assumptions:

1. The output resistances of the MOSFETs are infinite: \( r_o = \infty \).
2. The MOSFETs are identical and operate in the saturation region. The threshold voltages are the same, \( V_{t1} = V_{t2} = V_t \), and the constants are equal, \( K_{n1} = K_{n2} = K_n \).
3. The output resistance of the transistor current source is infinite: \( R_{SS} = \infty \).

Assuming that the drain current is related to \( v_{GS} \) by the approximate square law relationship in Eq. (7.10), we can write

\[
i_D = K_n(v_{GS} - V_t)^2
\]

(9.37)

Taking the square root of both sides of Eq. (9.37), we can write the square root of the drain currents as

\[
\sqrt{i_{D1}} = \sqrt{K_n(v_{GS1} - V_t)}
\]

(9.38)

\[
\sqrt{i_{D2}} = \sqrt{K_n(v_{GS2} - V_t)}
\]

(9.39)

We can subtract \( \sqrt{i_{D2}} \) from \( \sqrt{i_{D1}} \) to find a relation for the differential voltage \( v_{id} = v_{GS1} - v_{GS2} \):

\[
\sqrt{i_{D1}} - \sqrt{i_{D2}} = \sqrt{K_n(v_{GS1} - V_t)} - \sqrt{K_n(v_{GS2} - V_t)} = \sqrt{K_n(v_{GS1} - v_{GS2})}
\]

(9.40)

\[
= \sqrt{K_n v_{id}}
\]

The sum of \( i_{D1} \) and \( i_{D2} \) must equal \( I_Q \); that is,

\[
I_Q = i_{D1} + i_{D2}
\]

(9.41)

Substituting Eq. (9.41) into Eq. (9.40) and solving the resultant quadratic, we find the drain currents:

\[
i_{D1} = \frac{I_Q}{2} + \sqrt{2K_n I_Q \left( \frac{v_{id}}{2} \right)} \left[ 1 - \frac{(v_{id}/2)^2}{(I_Q/2K_n)} \right]^{1/2}
\]

(9.42)

\[
i_{D2} = \frac{I_Q}{2} - \sqrt{2K_n I_Q \left( \frac{v_{id}}{2} \right)} \left[ 1 - \frac{(v_{id}/2)^2}{(I_Q/2K_n)} \right]^{1/2}
\]

(9.43)

At the quiescent point \( v_{id} = 0 \), we get

\[
i_{D1} = i_{D2} = \frac{I_Q}{2}
\]

\[v_{GS1} = v_{GS2} = V_{GS}\]

\[I_Q = 2I_D = 2K_n(V_{GS} - V_t)^2\]
Substituting these relations into Eqs. (9.42) and (9.43), we can rewrite $i_D = i_{D2}$ as

$$i_{D1} = \frac{I_Q}{2} + \left(\frac{I_Q}{V_{GS} - V_t}\right) \left(\frac{v_{id}}{2}\right) \left[1 - \left(\frac{v_{id}/2}{V_{GS} - V_t}\right)^2\right]^{1/2}$$

(9.44)

$$i_{D2} = \frac{I_Q}{2} - \left(\frac{I_Q}{V_{GS} - V_t}\right) \left(\frac{v_{id}}{2}\right) \left[1 - \left(\frac{v_{id}/2}{V_{GS} - V_t}\right)^2\right]^{1/2}$$

(9.45)

For $v_{id} \ll (V_{GS} - V_t)$, $i_{D1}$ and $i_{D2}$ can be approximated by

$$i_{D1} = \frac{I_Q}{2} + \left(\frac{I_Q}{V_{GS} - V_t}\right) \left(\frac{v_{id}}{2}\right)$$

(9.46)

$$i_{D2} = \frac{I_Q}{2} - \left(\frac{I_Q}{V_{GS} - V_t}\right) \left(\frac{v_{id}}{2}\right)$$

(9.47)

Thus, the change in drain current from the quiescent value of $I_Q/2$ is given by

$$\Delta I_D = \left(\frac{I_Q}{V_{GS} - V_t}\right) \left(\frac{v_{id}}{2}\right)$$

(9.48)

which can be normalized with respect to the maximum value $I_Q/2$ as

$$\frac{\Delta I_D}{I_Q/2} = \frac{v_{id}}{V_{GS} - V_t} = \frac{v_{id}}{\sqrt{I_Q/2K_n}}$$

(9.49)

If $v_{id}$ is sufficiently large, all of the biasing current $I_D$ must flow through only one of the MOSFETs. The range of $v_{id}$ for which both transistors conduct can be found from Eq. (9.49) under the condition $\Delta I_D = I_Q/2$. That is,

$$v_{id} \leq \left(\frac{I_Q}{2K_n}\right)^{1/2}$$

(9.50)

This equation gives the value of $v_{id}$ for which the current $I_Q$ is carried by one of the two transistors. Thus, outside the range defined by Eq. (9.50), currents $i_{D1}$ and $i_{D2}$ will be either zero or $I_Q$. The plots of the normalized currents $i_{D1}$ and $i_{D2}$ against the differential voltage $v_{id}/v_n$ are shown in Fig. 9.9, where $v_n = \sqrt{I_Q/2K_n}$.

The output voltages of a MOSFET pair are as follows:

$$v_{o1} = V_{DD} - i_{D1}R_D$$

(9.51)

$$v_{o2} = V_{DD} - i_{D2}R_D$$

(9.52)
The differential DC output voltage is

\[ v_{od} = v_{O1} - v_{O2} = R_D(i_{D2} - i_{D1}) = -R_D \Delta I_D \]

Substituting for \( \Delta I_D \) from Eq. (9.47), we get

\[ v_{od} = -R_D \left( \frac{I_Q}{V_{GS} - V_t} \right) \left( \frac{v_{id}}{2} \right) \]

\[ = -R_D \left( \frac{K_n I_Q}{2} \right)^{1/2} v_{id} \]

(9.53)

which shows the relation between the output voltage \( v_{od} \) and the differential voltage \( v_{id} \). If \( v_{id} \) is zero, \( v_{od} \) is also zero.

**Small-Signal Analysis**

From Eq. (7.27), the transconductance of the small-signal model is given by

\[ g_m = 2K_n(V_{GS} - V_t) \]

\[ = 2 \frac{2K_n I_Q}{2} \]

(9.54)

which shows that for a MOSFET a higher value of \( g_m \) requires a higher value of the biasing current \( I_Q \).

The half circuit for differential input voltage is shown in Fig. 9.10(a), and its small-signal equivalent circuit is shown in Fig. 9.10(b). The \( (r_{o1} \parallel R_D) \) differential voltage gain \( A_d \) for a single-ended output can easily be derived as

\[ A_d = \frac{v_{od}/2}{v_{id}/2} = -g_m(r_{o1} \parallel R_D) \]

(9.55)

which, for \( r_{o1} \gg R_D \), can be approximated to \( A_d = -g_mR_D \).

**NOTE** If we define \( v_{od} = v_{o2} - v_{o1} \) and \( v_{gd} = v_{G1} - v_{G2} \), then \( A_d = g_m(R_D \parallel r_{o1}) \).
The half circuit for the common-mode input voltage is shown in Fig. 9.11(a), and its small-signal equivalent circuit is shown in Fig. 9.11(b). Using KVL around the input and the gate–source loop, we get

\[ v_{gs} = v_{ic} - v_{gs}g_{m}2R_{SS} \]

which leads to the following relationship between \( v_{gs} \) and \( v_{ic} \):

\[ v_{gs}(1 + g_{m}2R_{SS}) = v_{ic} \]  \hspace{1cm} (9.56)

The common-mode output voltage is given by

\[ v_{oc} = -R_{D}i_{d} = -R_{D}g_{m}v_{gs} \]

Substituting \( v_{gs} \) from Eq. (9.56) into the above equation, we get

\[ v_{oc} = v_{ic}\left[ \frac{-g_{m}R_{D}}{1 + g_{m}2R_{SS}} \right] \]

which gives the common-mode voltage gain \( A_{c} \) (for a single-ended output) as

\[ A_{c} = \frac{v_{oc}}{v_{ic}} = \frac{-g_{m}R_{D}}{1 + g_{m}2R_{SS}} \]  \hspace{1cm} (9.57)

**NOTE**  To simplify the analysis, the effect of transistor output resistance \( r_{o1} \) is not included in Eq. (9.57).
From Eqs. (9.55) and (9.57) for \( r_{o1} \gg R_D \), we can find CMRR as

\[
CMRR = \frac{A_d}{A_c} = 1 + 2g_mR_{SS}
\]

which is valid only for single-ended output; that is, the output is taken out from one of the output terminals.

The common-mode and differential input resistances are given by

\[
R_{ic} = R_{id} = \infty
\]

\[\text{NOTE}\] To simplify the analysis, the effect of transistor output resistance \( r_o \) is not included in Eqs. (9.57) and (9.58).

**EXAMPLE 9.2**

Analyzing a MOS differential pair with an active current source  The parameters of the MOS differential pair in Fig. 9.8 are \( R_{SS} = 50 \, \text{k}\Omega \), \( I_Q = 10 \, \text{mA} \), \( V_{DD} = 30 \, \text{V} \), and \( R_D = 5 \, \text{k}\Omega \). The NMOSs are identical and have \( K_n = 1.25 \, \text{mA/V}^2 \) and \( V_t = 1.0 \, \text{V} \). Assume \( V_M = 100 \, \text{V} \).

(a) Calculate the DC drain currents through the MOSFETs if \( v_{id} = 10 \, \text{mV} \).

(b) Assuming \( I_{D1} = I_{D2} \), calculate \( A_d \), \( A_c \), and CMRR; \( R_{id} \) and \( R_{ic} \); and the small-signal output voltage if \( v_{g1} = 10 \, \text{mV} \) and \( v_{g2} = 20 \, \text{mV} \).

(c) Find the drain voltage \( V_D \).

**SOLUTION**

(a) For \( v_{id} = 10 \, \text{mV} \), Eq. (9.42) gives the DC drain current \( i_{D1} \) for transistor \( M_1 \) as

\[
i_{D1} = \frac{10 \, \text{m}}{2} + \frac{2}{1} \times 1.25 \, \text{m} \times 10 \, \text{m} \times \left(\frac{10 \, \text{m}}{2}\right) \left[1 - \frac{(10 \, \text{m})^2}{10 \, \text{m}(2 \times 1.25 \, \text{m})}\right]^{1/2} = 5.25 \, \text{mA}
\]

(b) We know that \( I_{D1} = I_{D2} = I_Q/2 = 10 \, \text{mA}/2 = 5 \, \text{mA} \). From Eq. (9.54),

\[
g_m = \frac{2}{1} \times K_d I_Q = \frac{2}{1} \times 1.25 \, \text{m} \times 10 \, \text{m} = 5 \, \text{mA/V}
\]

\[
r_{o1} = \frac{V_M}{I_D} = \frac{100}{5 \, \text{m}} = 20 \, \text{k}\Omega
\]

From Eq. (9.55), the single-ended differential voltage gain \( A_d \) is

\[
A_d = -g_m R_D = -5 \, \text{m} \times (5 \, \text{k} \parallel 20 \, \text{k}) = -20 \, \text{V/V}
\]

From Eq. (9.57), the single-ended common-mode voltage gain \( A_c \) is

\[
A_c = \frac{-g_m R_D}{1 + g_m 2 R_{SS}} = \frac{-5 \, \text{m} \times 5 \, \text{k}}{1 + 5 \, \text{m} \times 2 \times 50 \, \text{k}} = -0.0399 \, \text{V/V}
\]
Thus, CMRR = \(|A_d/A_c| = 20/0.0399 = 501\) (or 54 dB).

From Eq. (9.59),
\[ R_{id} = R_{ic} = \infty \]

We know that
\[ v_{id} = v_{g2} - v_{g1} = 20 \text{ mV} - 10 \text{ mV} = 10 \text{ mV} \]
and \[ v_{ic} = \frac{v_{g1} + v_{g2}}{2} = \frac{10 \text{ mV} + 20 \text{ mV}}{2} = 15 \text{ mV} \]

Using Eq. (9.10), we have
\[ v_o = A_d v_{id} + A_c v_{ic} = -20 \times 10 \text{ mV} - 0.0399 \times 15 \text{ mV} = -201 \text{ mV} \]

(c) The DC drain voltage at the drain terminal of a transistor is
\[ V_D = V_{DD} - I_D R_D = 30 \text{ V} - 5 \text{ mA} \times 5 \text{ k}\Omega = 5 \text{ V} \]

Thus, for \(A_d = -20\), the maximum differential voltage will be \(v_{id} = 5/20 = 250 \text{ mV}\). Therefore, \(V_{DD}\) must be greater than \(I_D R_D\) in order to allow output voltage swing due to the input voltages.

---

**EXAMPLE 9.3**

**Analyzing an NMOS differential pair with an active current source** Repeat Example 9.2 if the transistor current source is replaced by the resistance \(R_{SS} = 50 \text{ k}\Omega\); that is, \(I_{SS} = 0\). Assume \(V_M = 100 \text{ V}\).

**SOLUTION**

(a) The DC drain current and the gate–source voltage of the MOSFETs can be determined from the DC common-mode half circuit for \(v_{g1} = v_{g2} = 0\), shown in Fig. 9.12:

\[ V_{GS} + 2I_D R_{SS} = V_{SS} \]

\[ V_{DD} = +15 \text{ V} \]

\[ R_D = 10 \text{ k}\Omega \]

\[ V_{GS} \]

\[ V_{SR} = -15 \text{ V} \]

\[ 2R_{SS} = 100 \text{ k}\Omega \]

**FIGURE 9.12** Common-mode half circuit for DC biasing of MOS pair
Differential Amplifiers

Substituting $i_D = I_D$ and $v_{GS} = V_{GS}$ from Eq. (9.37) into the preceding equation, we get

$$V_{GS} + 2R_{SS}K_n(V_{GS} - V_t)^2 = V_{SS}$$

Substituting the values, we get

$$V_{GS} + 2 \times 50 \text{k} \times (V_{GS} - 1)^2 = 15$$

This quadratic equation yields the solution for the gate–source voltage $V_{GS} = 1.478 \text{ V}$, or $0.514 \text{ V}$. Since $V_{GS} > V_t$, the acceptable value is $V_{GS} = 1.478 \text{ V}$, which gives the drain current as

$$I_{D1} = K_n(V_{GS} - V_t)^2 = 1.25 \text{ m} \times (1.478 - 1)^2 = 285.6 \mu\text{A}$$

Therefore, the voltage at the source with respect to the ground is

$$V_{SR} = -V_{GS} = -1.478 \text{ V} \quad \text{and} \quad I_Q = 2I_D = 2 \times 285.6 \mu\text{A} = 571.2 \mu\text{A}$$

(b) From Eq. (9.54),

$$g_m = 2 \frac{2K_nI_Q}{2 \times 1.25 \text{ m} \times 571.2 \mu\text{A}} = 1.194 \text{ mA/V}$$

$$r_{o1} = \frac{V_M}{I_D} = \frac{100}{285.6 \mu\text{A}} = 350 \text{k}\Omega$$

From Eq. (9.55),

$$A_d = -g_m(R_D \parallel r_{o1}) = -1.194 \text{ m} \times (5 \text{k} \parallel 350 \text{k}) = -5.89 \text{ V/V}$$

From Eq. (9.57),

$$A_c = \frac{-g_mR_D}{1 + g_m \times 2R_{SS}} = \frac{-1.194 \text{ m} \times 5 \text{k}}{1 + 1.194 \text{ m} \times 2 \times 50 \text{k}} = -0.0489$$

Thus, $\text{CMRR} = |A_d/A_c| = 5.89/0.0489 = 120.4$, or $41.61 \text{ dB}$. From Eq. (9.59),

$$R_{id} = R_{ic} = \infty$$

We know that

$$v_{id} = v_{g2} - v_{g1} = 20 \text{ mV} - 10 \text{ mV} = 10 \text{ mV}$$

$$v_{ic} = \frac{v_{g1} + v_{g2}}{2} = \frac{10 \text{ mV} + 20 \text{ mV}}{2} = 15 \text{ mV}$$

Using Eq. (9.10), we have

$$v_o = A_dv_{id} + A_cv_{ic} = -5.887 \times 10 \text{ mV} - 0.489 \times 15 \text{ mV} = -59.6 \text{ mV}$$

Thus, the output voltage and the voltage gain are much lower than with current-source biasing because the biasing current is low.
EXAMPLE 9.4

Designing a MOS differential pair with an active current source

(a) Design a MOS differential pair as shown in Fig. 9.13, in which one input terminal is grounded. The output is taken from the drain of transistor M1. The DC biasing current is $I_Q = 10$ mA, and $V_{DD} = V_{SS} = 15$ V. The MOSFETs are identical and have $K_n = 1.25$ mA/V$^2$ and $V_t = 1$ V. A small-signal voltage gain of $A_1 = 10$ V/V is required. Assume $V_M = \infty$.

(b) Calculate the design values of $A_d$, $A_c$, and CMRR.

(c) Find the maximum permissible value of $R_D$ and the corresponding voltage gain $A_d$.

SOLUTION

(a) $I_Q = 10$ mA, and $I_{D1} = I_{D2} = I_Q/2 = 10$ mA/2 = 5 mA. From Eq. (9.54),

$g_m = \sqrt{2 \times K_n I_Q} = \sqrt{2 \times 1.25 \times 10} = 5$ mA/V

From Eq. (9.37), the DC gate–source voltage is

$V_{GS1} = -\left(\frac{I_D}{K_n} + V_t\right) = -\left(\frac{5 \text{ mA}}{1.25 \text{ m}} + 1\right) = -2.236$ V

Therefore, the voltage at the source terminal with respect to the ground is $V_{SR} = -V_{GS} = -2.236$ V, and

$R_{SS} = \frac{V_{SR} - V_{SS}}{I_{SS}} = \frac{(-2.236 + 15)}{10 \text{ mA}} = 1.3$ kΩ

(b) $v_{G1} = V_{G1} + v_{g1}$, $V_{G1} = 0$, and $v_{G2} = v_{g2} = 0$. Then

$v_{id} = v_{g1} - v_{g2} = v_{g1}$

$v_{ic} = \frac{v_{g1} + v_{g2}}{2} = \frac{v_{g1}}{2}$

$V_{DD} = +15$ V

$V_{SS} = -15$ V

FIGURE 9.13 MOS differential pair with a single input
9.4.2 MOS Differential Pair with Active Load

MOS differential amplifiers are normally used with current mirror active loads. A commonly used configuration is shown in Fig. 9.14. The current mirror consists of transistors M₃ and M₄. Increasing the input voltage to M₁ by \( \frac{v_{id}}{2} \) will cause the drain current of M₁ to increase by an amount \( g_m \cdot \frac{v_{id}}{2} \). This increase will cause a similar increase in the drain current of M₄ due to the current mirror effect and also a decrease in the drain current of M₂. Since M₁ and M₂ are NMOS transistors and their complements (M₃ and M₄) are PMOS types, this configuration is known as a CMOS amplifier; the manufacturing process by which the amplifiers are produced is known as CMOS technology.

The small-signal equivalent of the output side of the CMOS amplifier is shown in Fig. 9.15. The output resistance \( R_o \) is the parallel combination of \( r_{o2} \) and \( r_{o4} \). That is,

\[
R_o = r_{o2} \parallel r_{o4}
\]

From Eq. (9.7),

\[
v_{o1} = v_{oc} + \frac{v_{id}}{2} = A_c v_{ic} + A_d v_{id} = A_c \frac{v_{g1}}{2} + A_d v_{g1} = \frac{v_{g1}}{2} (A_c + 2A_d)
\]

Substituting for \( A_d \) and \( A_c \) from Eqs. (9.55) and (9.57) gives the voltage gain \( A_1 \) as

\[
A_1 = \frac{v_{o1}}{v_{g1}} = \frac{1}{2} (A_c + 2A_d) = -\frac{1}{2} \left[ \frac{g_m R_D}{1 + 2g_m R_{SS}} + 2g_m R_D \right]
\]

Substituting \( A_1 = -10 \), \( g_m = 5 \, \text{mA/V} \), and \( R_{SS} = 1.3 \, \text{kΩ} \) into the above equation gives \( R_D = 1.93 \, \text{kΩ} \).

\[
r_{o1} = \frac{V_M}{I_{D1}} = \infty
\]

From Eq. (9.55),

\[
A_d = -g_m (R_D \parallel r_{o1}) = -5 \, \text{m} \times (1.93 \, \text{k} \parallel 20 \, \text{k}) = -8.8 \, \text{V/V}
\]

From Eq. (9.57),

\[
A_c = \frac{-g_m R_D}{1 + g_m \times 2R_{SS}} = -\frac{5 \, \text{m} \times 1.93 \, \text{k}}{1 + 5 \, \text{m} \times 2 \times 1.7 \, \text{k}} = -0.64 \, \text{V/V}
\]

Thus, CMRR = \(|A_d/A_c| = 8.8/0.64 = 13.76, \) or 22.78 dB.

(c) For \( I_D = I_Q/2 = 5 \, \text{mA} \) and \( V_{DD} = 15 \, \text{V} \), the maximum value of \( R_D \) is

\[
R_{D(max)} = \frac{V_{DD}}{I_D} = \frac{15 \, \text{V}}{5 \, \text{mA}} = 3 \, \text{kΩ}
\]

which gives the maximum value of

\[
A_{d(max)} = -g_m R_{D(max)} = -5 \, \text{mA} \times 3 \, \text{kΩ} = -15 \, \text{V/V}
\]

\[\text{Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.}\]
Using Eq. (9.55), we find the differential voltage gain $A_d$:

$$A_d = \frac{v_o}{v_{id}} = -g_m (r_{o2} || r_{o4})$$

(9.61)

where $g_m$, which is the transconductance of MOSFET $M_2$, is given by Eq. (9.54). Substituting $r_{o4} = r_{o2} = 2V_m/I_Q$ and $g_m = \sqrt{2K_nI_Q}$ and simplifying, we get

$$A_d = -\frac{2K_n}{I_Q} V_m$$

(9.62)

which gives a higher voltage gain for a lower value of $I_Q$.

**EXAMPLE 9.5**

**Designing a CMOS amplifier**

(a) Design the CMOS amplifier shown in Fig. 9.16 by determining the $W/L$ ratios of the MOSFETs and the threshold voltage $V_t$. The differential voltage gain should be $A_d = 60 \text{ V/V}$ at biasing current $I_Q = 10 \mu A$. Assume identical transistors whose channel modulation voltage is $V_m = 20 \text{ V}$, channel constant is $K_x = 20 \mu A/V^2$, and channel length is $L = 10 \mu m$. The $W/L$ ratio of the current source is 2, and $K_x = 10 \mu A/V^2$. Assume $V_{DD} = V_{SS} = 5 \text{ V}$.

(b) Use PSpice/SPICE to find the small-signal differential voltage gain $A_d$ for $v_{G1} = 1 \text{ mV}$ and $v_{G2} = 0$.

**SOLUTION**

(a) $A_d = 60$, $I_Q = 10 \mu A$, $V_m = 20 \text{ V}$, $K_x = 20 \mu A/V^2$, and $L = 10 \mu m$. From Eq. (9.62), we find the MOS constant $K_n$ as

$$K_n = \left(\frac{I_Q}{2}\right)\left(\frac{A_d}{V_m}\right)^2 = \left(\frac{10 \mu A}{2}\right)\left(\frac{60}{20}\right)^2 = 45 \mu A/V^2$$
From Eq. (9.21), we find the $W/L$ ratio as

$$\frac{W}{L} = \frac{K_n}{K_x} = \frac{45 \mu}{20 \mu} = 2.25$$

Since $L = 10 \mu$m, the desired value is $W = 2.25 \times 10 \mu$m = 22.5 $\mu$m. For identical transistors for the current source,

$$V_{GS6} = \frac{V_{DD} + V_{SS}}{3} = \frac{(5 + 5) V}{3} = 3.33 V$$

Since $W = 2L$ for the MOSFETs of the current source, the MOS constant is

$$K_{n6} = \frac{W_6K_x}{2L_6} = \frac{2 \times 10}{2} \mu A/V^2 = 10 \mu A/V^2$$

The biasing current is given by

$$I_Q = K_{n6}(V_{GS6} - V_I)^2$$

which, for $V_{GS6} = 3.33 V$, $I_Q = 10 \mu A$, and $K_{n6} = 10 \mu A/V^2$, gives $V_I = 2.33 V$. Then

$$r_{o2} = r_{o4} = \frac{2V_M}{I_Q} = \frac{2 \times 20}{10 \mu} = 4 M\Omega$$

$$R_o = r_{o2} || r_{o4} = 2 M\Omega$$

$$r_{o5} = \frac{V_M}{I_Q} = \frac{20}{10 \mu} = 2 M\Omega$$

(b) The CMOS amplifier for PSpice simulation is shown in Fig. 9.17. The plot of the output voltage is shown in Fig. 9.18, which gives a voltage gain 81.72 $V/V$ (expected value 60). Note there is no phase shift of $180^\circ$ because the output is taken from the drain terminal of the pair.

The results of simulation (.TF analysis) are as follows. (The expected values are listed on the right.)

**** SMALL-SIGNAL CHARACTERISTICS

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V(4)/V(1)$</td>
<td>8.189E+01 = 81.89</td>
</tr>
<tr>
<td>$V_{G1}$</td>
<td>10 $\mu$V</td>
</tr>
<tr>
<td>$V_{DD}$</td>
<td>5V</td>
</tr>
<tr>
<td>$V_{SS}$</td>
<td>5V</td>
</tr>
<tr>
<td>$I_D1$</td>
<td>5.44E-06 = 5.44 $\mu$A</td>
</tr>
<tr>
<td>$I_D5$</td>
<td>1.09E-05 = 10.9 $\mu$A</td>
</tr>
<tr>
<td>$I_Q$</td>
<td>10 $\mu$A</td>
</tr>
</tbody>
</table>

FIGURE 9.16 CMOS amplifier

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
9.4.3 Cascoded MOS Differential Amplifier

We can notice from Eq. (9.61) that the differential gain increases with the output resistance $R_o$ of the active amplifier load. MOSFETs can be connected in cascode configuration as shown in Fig. 9.6(a) to increase the output resistance and to improve the frequency response. A common modification to Fig. 9.14(a) is shown in Fig. 9.19(a). Transistors $M_1$ and $M_2$ are connected in a common-source configuration and form a common-source differential pair, whereas $M_3$ and $M_4$ are connected in a common-gate configuration and form a common-gate differential stage. $M_6$ and $M_8$ act as the load. For identical MOSFETs, the DC biasing currents are equal, $I_{D1} = I_{D2} = I_{D4} = I_{D6} = I_{D8} = I_Q/2$. All MOSFETs have equal output resistances, $r_{o1} = r_{o2} = r_{o4} = r_{o6} = r_{o8} = V_M/I_{D1}$, and equal transconductances, $g_{m1} = g_{m2} = g_{m4} = g_{m6} = g_{m8} = 2K_n(V_{GS1} - V_t)$. The small-signal half-circuit is shown in Fig. 9.19(b). Using Eq. (9.34), we can find the effective load resistance offered by $M_6$ and $M_8$ as given by

$$r_{o6}' = r_{o6}(2 + g_{m1}r_{o6})$$  \hspace{1cm} (9.63)
Similarly, we can find the effective resistance offered by $M_2$ and $M_4$ as given by

$$r_{o4}' = r_{o4}(2 + g_{m1}r_{o4})$$  \hspace{1cm} (9.64)

Therefore, the effective output resistance at the output terminal as shown in Fig. 9.19(c) is given by

$$R_o = r_{o4}' || r_{o6}'$$  \hspace{1cm} (9.65)

Applying Eq. (9.61) we can find the small-signal differential output voltage as given by

$$A_d = \frac{v_{od}}{v_{id}} = -g_{m1}R_o = -g_{m1}(r_{o4}' || r_{o6}')$$  \hspace{1cm} (9.66)

This should give a much larger differential gain compared to that of Fig. 9.14(a). For example, if $|V_M| = 20 \text{ V}$, $I_{D1} = 5 \mu \text{ A}$, $g_{m1} = 31.5 \mu \text{ A/V}$, $r_{o4} = r_{o6} = 5 \text{ M\Omega}$, $r_{o8}' = r_{o8} = 514 \text{ M\Omega}$, and $R_o = 257 \text{ M\Omega}$, we get $A_d = 8126 \text{ V/V}$.

**EXAMPLE 9.6**

**Analyzing cascoded MOS amplifiers**  The DC biasing current of the MOS amplifier shown in Fig. 9.19(a) is kept constant at $I_Q = 10 \mu \text{ A}$. All MOS transistors are identical: $V_M = 20 \text{ V}$, $K_n = 25 \mu \text{ A/V}^2$, $W = 30 \mu \text{ m}$, and $L = 10 \mu \text{ m}$.

(a) Determine the differential voltage gain $A_d$ for single-ended output at the drain terminal of $M_4$.

(b) Use PSpice to verify the result.
SOLUTION

You are given $V_M = 20\,\text{V}$, $K_n = 25\,\mu\text{A}/\text{V}^2$, $W = 30\,\mu\text{m}$, $L = 10\,\mu\text{m}$, and $I_D = I_Q/2 = 10\,\mu\text{A}/2 = 5\,\mu\text{A}$.

(a) The output resistance of $M_2$ is $r_{o2} = r_{o6} = r_{o8} = 2V_M/I_Q = 2 \times 20\,\text{V}/10\,\mu\text{A} = 4\,\text{M}\Omega$. The transconductance of $M_2$ is $g_{m2} = 2K_nI_Q = 2 \times 25\,\mu\text{A} \times 10\,\mu\text{A} = 22.36\,\mu\text{A}/\text{V}$. The load resistance due to $M_2$ and $M_4$ cascoded combination is

$$r_{o4} = r_{o6} = r_{o4}(2 + g_{m2}r_{o4}) = 4\,\text{M}\Omega \times (2 + 22.36\,\mu\text{A}/\text{V} \times 4\,\text{M}\Omega) = 366\,\text{M}\Omega.$$  

The effective resistance of the active load is $R_o = r_{o4} || r_{o6} = 366\,\text{M}\Omega || 366\,\text{M} = 183\,\text{M}\Omega$.

Thus, the differential voltage becomes $V_{\text{diff}} = A_d = g_{m2}R_o = 22.36\,\mu\text{A} \times 183\,\text{M}\Omega = 4089\,\text{V}/\text{V}$.

(b) The circuit for PSpice simulation is shown in Fig. 9.20 with a sinusoidal signal of $1\,\mu\text{V}$ at $10\,\text{Hz}$. The plot of the output voltage is shown in Fig. 9.21, which gives a voltage gain of $5370\,\text{V}/\text{V}$ (expected value $4089\,\text{V}/\text{V}$). Note there is no phase shift of $180^\circ$ because the output is taken from the drain terminal of the other pair.

KEY POINTS OF SECTION 9.4

- A MOS amplifier exhibits a linear DC characteristic and has a very high input resistance, tending to infinity.
- It is relatively easy to connect MOS transistors in cascode form in order to control the drain current and give high output resistance. A high-voltage gain can be obtained with a cascode connection.

9.5 Depletion MOS Differential Amplifiers

The depletion MOS amplifiers are similar to the enhancement MOSs, except that the gate–source voltage of a depletion MOS can range from a negative value to a positive value. The drain current becomes the maximum $i_{D(\text{max})} = I_{DSS}$ when $v_{GS} = 0$. 

FIGURE 9.20 PSpice schematic for Example 9.6

FIGURE 9.21 PSpice plot of small-signal output voltage for Example 9.6
9.5.1 Depletion MOS Differential Pair with Resistive Load

A depletion NMOS source-coupled pair is shown in Fig. 9.22. Although the DC biasing circuit can be either a simple resistor (in which case the equivalent current generator will be zero) or a transistor current source, a current source is generally used.

DC Transfer Characteristics

The following analysis is performed for an \(n\)-channel MOS pair, but it is equally applicable to a \(p\)-channel pair with appropriate sign changes. The analysis can be simplified by making the following assumptions:

1. The output resistances of the MOSFETs are infinite: \(r_d = \infty\).
2. The MOSFETs are identical and operate in the saturation region. The pinch-down voltages are the same, \(V_{p1} = V_{p2} = V_p\), and the drain currents (with drain \(V_{GS} = 0\)) are equal, \(I_{DSS1} = I_{DSS2} = I_{DSS}\).
3. The output resistance of the transistor current source is infinite: \(R_{SS} = \infty\).

Using KVL around the loop formed by the two input voltages and two gate–source junctions, we get

\[
v_{G1} - v_{GS1} + v_{GS2} - v_{G2} = 0
\]  

(9.67)

Assuming that the drain current is related to \(v_{GS}\) by the approximate square law relationship in Eq. (7.21), we get

\[
i_D = I_{DSS} \left(1 - \frac{v_{GS}}{V_p}\right)^2
\]

from which we can find

\[
\frac{v_{GS}}{V_p} = 1 - \left(\frac{i_D}{I_{DSS}}\right)^{1/2}
\]  

(9.68)
Substituting Eq. (9.68) into Eq. (9.67) yields

\[
-v_{G1} + v_{G2} = -v_{GS1} + v_{GS2}
\]

\[
= - V_p \left[ 1 - \left( \frac{i_{D1}}{I_{DSS}} \right)^{1/2} \right] + V_p \left[ 1 - \left( \frac{i_{D2}}{I_{DSS}} \right)^{1/2} \right]
\]

\[
= V_p \left( \frac{i_{D1}}{I_{DSS}} \right)^{1/2} - V_p \left( \frac{i_{D2}}{I_{DSS}} \right)^{1/2}
\]

which gives

\[
-v_{G1} + v_{G2} = \frac{v_{id}}{V_p} = \left( \frac{i_{D1}}{I_{DSS}} \right)^{1/2} - \left( \frac{i_{D2}}{I_{DSS}} \right)^{1/2}
\]

(9.70)

where \( v_{id} = v_{G2} - v_{G1} \) is the differential DC voltage.

Using KCL at the source nodes of the transistors gives

\[
I_Q = i_{D1} + i_{D2}
\]

(9.71)

Substituting Eq. (9.71) into Eq. (9.70) and solving the resultant quadratic, we get

\[
i_{D1} = \frac{I_Q}{2} + \frac{I_Q}{2} \left( \frac{v_{id}}{V_p} \right) \left[ 2 \left( \frac{I_{DSS}}{I_Q} \right) - \left( \frac{v_{id}}{V_p} \right)^2 \left( \frac{I_{DSS}}{I_Q} \right)^2 \right]^{1/2}
\]

(9.72)

\[
i_{D2} = \frac{I_Q}{2} - \frac{I_Q}{2} \left( \frac{v_{id}}{V_p} \right) \left[ 2 \left( \frac{I_{DSS}}{I_Q} \right) - \left( \frac{v_{id}}{V_p} \right)^2 \left( \frac{I_{DSS}}{I_Q} \right)^2 \right]^{1/2}
\]

(9.73)

If \( v_{id} \) is sufficiently large, all of the biasing current \( I_Q \) must flow through only one of the MOSFETs. The range of \( v_{id} \) for which both transistors conduct can be found from Eq. (9.72) with the condition \( i_{D2} = 0 \); that is,

\[
\frac{I_Q}{2} - \frac{I_Q}{2} \left( \frac{v_{id}}{V_p} \right) \left[ 2 \left( \frac{I_{DSS}}{I_Q} \right) - \left( \frac{v_{id}}{V_p} \right)^2 \left( \frac{I_{DSS}}{I_Q} \right)^2 \right]^{1/2} = 0
\]

which gives

\[
\left| \frac{v_{id}}{V_p} \right| \leq \frac{I_Q}{I_{DSS}}
\]

(9.74)

This equation gives the value of \( v_{id} \) for which the current \( I_Q \) is carried by one of the two transistors. Typical drain currents for various values of \( I_Q \) are shown in Fig. 9.23. The range of \( v_{id} \) for which the circuit exhibits a linear characteristic is approximately equal to the pinch-down voltage \( |V_p| \) (typically 2 V to 5 V), compared to \( V_T \) (26 mV) for BJTs.

The output voltages for a MOS differential pair are

\[
v_{O1} = V_{DD} - i_{D1}R_D
\]

(9.75)

\[
v_{O2} = V_{DD} - i_{D2}R_D
\]

(9.76)
The differential DC output voltage is
\[ v_{od} = v_{O1} - v_{O2} = R_D(i_{D2} - i_{D1}) \]

Substituting \( i_{D1} \) and \( i_{D2} \) from Eqs. (9.72) and (9.73) into the above equation and simplifying, we get
\[
v_{od} = -\frac{I_Q R_D}{V_p} v_{id} \left[ 2 \left( \frac{I_{DSS}}{I_Q} \right) - \left( \frac{v_{id}}{V_p} \right)^2 \left( \frac{I_{DSS}}{I_Q} \right)^2 \right]^{1/2}
\] (9.77)

As with any differential circuit, if \( v_{id} \) is zero, \( v_{od} \) is also zero. A MOS-coupled pair allows direct coupling of cascaded stages without introducing DC offsets.

**Small-Signal Analysis**

From Eq. (7.78), the transconductance is given by
\[
g_m = -\frac{2I_{DSS}}{V_p} \left( 1 - \frac{V_{GS}}{V_p} \right)
\]
\[
= \frac{2}{|V_p|} ||I_{DSS}||^{1/2}
\] (9.78)

The small-signal analyses for differential and common-mode gains are the same as the enhancement MOS pair. From Eqs. (9.55) and (9.57), we get
\[
A_d = -g_m(R_D \parallel r_o)
\] (9.79)
\[
A_c = \frac{-g_m R_D}{1 + g_m 2R_{SS}}
\] (9.80)

**NOTE** To simplify the analysis, the effect of transistor output resistance \( r_o \) is not included in Eq. (9.80).
EXAMPLE 9.7

Analyzing a depletion MOS differential pair with an active current source  The parameters of the depletion MOS differential pair in Fig. 9.22 are $R_{SS} = 50 \, \text{k}\Omega$, $I_Q = 10 \, \text{mA}$, $V_{DD} = 30 \, \text{V}$, and $R_D = 5 \, \text{k}\Omega$. The depletion MOSFETs are identical and have $V_p = -4 \, \text{V}$ and $I_{DSS} = 20 \, \text{mA}$. Assume $V_M = 100 \, \text{V}$.

(a) Calculate the DC drain currents through the MOSFETs if $v_{id} = 10 \, \text{mV}$.

(b) Assuming $I_{D1} = I_{D2}$, calculate $A_d$, $A_c$, and CMRR; $R_{id}$ and $R_{ic}$; and the small-signal output voltage if $v_{g1} = 10 \, \text{mV}$ and $v_{g2} = 20 \, \text{mV}$.

(c) Find the drain voltage $V_D$.

SOLUTION

(a) For $v_{id} = 10 \, \text{mV}$, Eq. (9.72) gives the DC drain current $i_{D1}$ for transistor $M_1$ as

$$i_{D1} = \frac{10 \, \text{m}}{2} \left(1 + \frac{100 \, \text{m}}{-4} \right) \left(\frac{2}{10 \, \text{m}}\right)^2 = 5.25 \, \text{mA}$$

$$i_{D2} = I_Q - i_{D1} = 10 \, \text{mA} - 5.25 \, \text{mA} = 4.75 \, \text{mA}$$

(b) We know that $I_{D1} = I_{D2} = I_Q/2 = 10 \, \text{mA}/2 = 5 \, \text{mA}$. From Eq. (9.78),

$$g_m = \frac{2[I_{D1}I_{DSS}]}{|V_p|} = \left(\frac{2}{4}\right) \times \frac{5 \, \text{mA} \times 20 \, \text{mA}}{5 \, \text{mA}} = 5 \, \text{mA} / \text{V}$$

$$r_{o1} = \frac{V_M}{I_D} = \frac{100}{5 \, \text{m}} = 20 \, \text{k}\Omega$$

From Eq. (9.79), the single-ended differential voltage gain $A_d$ is

$$A_d = -g_m(R_D || r_{o1}) = -5 \, \text{m} \times (5 \, \text{k} || 20 \, \text{k}) = -20 \, \text{V} / \text{V}$$

From Eq. (9.80), the single-ended common-mode voltage gain $A_c$ is

$$A_c = \frac{-g_m R_D}{1 + g_m 2 R_{SS}} = \frac{-5 \, \text{m} \times 5 \, \text{k}}{1 + (5 \, \text{m} \times 2 \times 50 \, \text{k})} = -0.0399 \, \text{V} / \text{V}$$

Thus, $\text{CMRR} = \left|\frac{A_d}{A_c}\right| = 20/0.0399 = 501$ (or 54 dB). From Eq. (9.59),

$$R_{id} = R_{ic} = \infty$$

We know that

$$v_{id} = v_{g2} - v_{g1} = 20 \, \text{mV} - 10 \, \text{mV} = 10 \, \text{mV}$$

and

$$v_{ic} = \frac{v_{g1} + v_{g2}}{2} = \frac{10 \, \text{mV} + 20 \, \text{mV}}{2} = 15 \, \text{mV}$$

Using Eq. (9.10), we have

$$v_o = A_d v_{id} + A_c v_{ic} = -20 \times 10 \, \text{mV} - 0.0399 \times 15 \, \text{mV} = -250.6 \, \text{mV}$$

(c) The DC drain voltage at the drain terminal of a transistor is

$$V_D = V_{DD} - I_{D}R_D = 30 \, \text{V} - 5 \, \text{mA} \times 5 \, \text{k}\Omega = 5 \, \text{V}$$

Thus, for $A_d = -20$, the maximum differential voltage will be $v_{id} = 5/20 = 250 \, \text{mV}$. Therefore, $V_{DD}$ must be greater than $I_{D}R_D$ in order to allow output voltage swing due to the input voltages.
9.5.2 Depletion MOS Differential Pair with Active Load

Like MOS differential amplifiers, depletion MOS differential amplifiers use a current mirror active load to achieve a large voltage gain. The active load can be made either with MOSFETs similar to that in Fig. 9.24 or with BJTs. A depletion MOS differential amplifier with a basic current source as the BJT active load is shown in Fig. 9.24. From Eq. (9.60), we know that the output resistance $R_o$ is the parallel combination of $r_{o2}$ and $r_{o4}$; that is,

$$ R_o = r_{o2} \parallel r_{o4} \quad (9.81) $$

where $r_{o2} = 2V_M/I_Q = $ output resistance of transistor $M_2$ and $r_{o4} = 2V_A/I_Q = $ output resistance of transistor $Q_4$.

Using Eq. (9.61), we find the differential voltage gain $A_d$:

$$ A_d = \frac{v_O}{v_{id}} = -g_m(r_{o2} \parallel r_{o4}) \quad (9.82) $$

where $g_m$, which is the transconductance of MOSFET $M_2$, is given by Eq. (9.78).

---

**KEY POINTS OF SECTION 9.5**

- A MOSFET amplifier has a very high input resistance, in the range of $10^9 \Omega$ to $10^{12} \Omega$.
- An active load considerably increases the differential gain of a MOSFET amplifier.
- The range of $v_{id}$ for which the circuit exhibits a linear DC characteristic is much higher for MOSFETs than for BJTs. For MOSFETs, this range is approximately equal to the pinch-down voltage $\left| V_p \right|$ (typically 2 V to 5 V), compared to $V_T$ (26 mV) for BJTs.
9.6 BJT Current Sources

The BJT current sources are similar to MOS sources. Since BJTs have a higher voltage gain as compared to MOSFETs, BJT sources can offer higher output resistances. The commonly used current sources are the basic current sources (which were applied in Secs. 7.8.1 and 8.7.1), the modified basic current source, the Widlar current source, the cascode current source, and the Wilson current source.

9.6.1 Basic Current Source

The simplest current source consists of a resistor and two transistors as shown in Fig. 9.25(a). Transistor Q1 is diode connected, and its C-B voltage is forced to zero: $v_{CB} = 0$. Thus, the C-B junction is off, and Q1 will operate in the active region. Transistor Q2 can be in the active region as well as in the saturation region.

Let us assume that Q1 and Q2 are identical transistors whose leakage currents are negligible and whose output resistances are infinite. Since the two transistors have the same B-E voltages (i.e., $V_{BE1} = V_{BE2}$), the collector and base currents are equal: $I_{C1} = I_{C2}$ and $I_{B1} = I_{B2}$. Applying KCL at the collector of Q1 gives the reference current:

$$I_R = I_{C1} + I_{B1} + I_{B2} = I_{C1} + 2I_{B1}$$

Since $I_{C1} = \beta_F I_{B1}$,

$$I_R = I_{C1} + 2I_{B1} = I_{C1} + \frac{2I_{C1}}{\beta_F}$$
which gives the collector current $I_{C1}$ as

$$I_{C1} = I_{C2} = \frac{I_R}{1 + 2/\beta_F} \quad (9.83)$$

$$= \frac{V_{CC} - V_{BE1}}{R_1} \times \frac{1}{1 + 2/\beta_F} \quad (9.84)$$

If the DC current gain $\beta_F >> 2$, which is usually the case, Eq. (9.84) is reduced to

$$I_{C1} = I_{C2} \approx I_R$$

Thus, for two identical transistors, the reference and output currents are almost equal. $I_{C2}$, which is the mirror image of $I_{C1}$, is known as the mirror current of $I_{C1}$. For transistors with small values of $\beta_F$, the current ratio will not be unity. In practice, however, the transistors may not be identical, and the two collector currents will have a constant ratio. The equivalent circuit for the ideal current source is shown in Fig. 9.25(b).

For a transistor with finite output resistance, the effect of the Early voltage $V_A$ should be taken into account, and the collector current in Eq. (8.8) can be modified to

$$I_C = I_S \left[ \exp \left( \frac{V_{BE}}{V_T} \right) - 1 \right] \left( 1 + \frac{V_{CE}}{V_A} \right) \quad (9.85)$$

If we take into account the variation in the collector current due to the C-E voltage, the ratio of the two collector currents can be found from

$$\frac{I_{C2}}{I_{C1}} = \frac{1 + V_{CE2}/V_A}{1 + V_{CE1}/V_A} \quad (9.86)$$

**Output Resistance $R_o$**

The small-signal AC equivalent circuit for determining the output resistance is shown in Fig. 9.25(c). Output resistance $R_o$ is the same as $r_{o2}$; that is,

$$R_o = \frac{v_x}{i_x} = r_{o2} = \frac{V_A}{I_{C2}} \quad (9.87)$$

Thevenin’s equivalent voltage is given by

$$V_{Th} = I_OR_o = I_{C2}R_o = I_{C2} \frac{V_A}{I_{C2}} = V_A \quad (9.88)$$

Norton’s and Thevenin’s equivalents of the current source are shown in Fig. 9.25[(d) and (e)]. If the output of the current source is open-circuited, a voltage of $-V_{Th}$ is expected to appear across transistor Q1. However, this will not actually happen because transistor Q1 will saturate when the voltage across the current source (i.e., the C-E voltage of Q1) reaches zero.

The current source in Fig. 9.25(a) behaves as a current sink rather than a source. A current source equivalent to this current sink can be obtained by using pnp transistors and a negative power supply. This arrangement is shown in Fig. 9.26.
EXAMPLE 9.8

Designing a simple basic current source

(a) Design the basic current source in Fig. 9.25(a) to give an output current of \( I_O = 5 \mu A \). The transistor parameters are \( \beta_F = 100 \), \( V_{CC} = 30 \text{ V} \), \( V_{BE1} = V_{BE2} = V_{CE1} = 0.7 \text{ V} \), and \( V_A = 150 \text{ V} \).

(b) Calculate the output resistance \( R_o \), Thevenin’s equivalent voltage \( V_{Th} \), and the collector current ratio if \( V_{CE2} = 20 \text{ V} \).

SOLUTION

\( V_{BE1} = V_{BE2} = V_{CE1} = 0.7 \text{ V} \), and \( V_A = 150 \text{ V} \).

(a) From Eq. (9.83),

\[
I_O = I_{C2} = I_{C1} = \frac{I_R}{1 + 2/\beta_F}
\]

which, for \( I_O = 5 \mu A \), gives \( I_R = (5 \mu A)(1 + 2/\beta_F) = 5.1 \mu A \). From Eq. (9.84),

\[
R_1 = \frac{V_{CC} - V_{BE1}}{I_R} = \frac{30 \text{ V} - 0.7 \text{ V}}{5.1 \mu A} = 5.75 \text{ M} \Omega
\]

(b) From Eq. (9.87),

\[
R_o = \frac{V_A}{I_{C2}} = \frac{150 \text{ V}}{5 \mu A} = 30 \text{ M} \Omega
\]

From Eq. (9.88),

\[
V_{Th} = V_A = 150 \text{ V}
\]

From Eq. (9.86),

\[
\frac{I_{C2}}{I_{C1}} = \frac{1 + V_{CE2}/V_A}{1 + V_{CE1}/V_A} = \frac{1 + 20/150}{1 + 0.7/150} = 1.128
\]

Thus, \( I_{C2} = 1.128 \times I_{C1} = 1.128 \times 5 \mu A = 5.64 \mu A \), which agrees, with a degree of error, with the desired value of \( I_O = 5 \mu A \).
1. For a current source of 5 \( \mu \text{A} \), a resistor of 5.75 M\( \Omega \) would be required. Resistors of such high value are very costly in terms of the die area. Resistors over 50 k\( \Omega \) are generally avoided for IC applications. Thus, this current source is not suitable for generating a current of less than about 0.6 mA at \( V_{\text{CC}} = 30 \text{ V} \) and 0.3 mA at \( V_{\text{CC}} = 15 \text{ V} \).

2. If the output of the current source is open-circuited, a voltage of \( V_{\text{BE1}} = -150 \text{ V} \) will not appear across transistor \( Q_1 \). Rather, the voltage will be \( V_{\text{CE1(sat)}} = 0.2 \text{ V} \).

### 9.6.2 Modified Basic Current Source

Notice from Eq. (9.83) that the collector current \( I_{C2} (= I_{C1}) \) differs from the reference current \( I_R \) by a factor of \( (1 + 2/\beta_F) \). For low-gain transistors (especially \( pnp \) types), \( I_{C2} \) can differ significantly from \( I_R \). The error can be reduced by adding another transistor so that \( I_{C2} \) becomes less dependent on the transistor parameter \( \beta_F \). This type of circuit is shown in Fig. 9.27(a). Applying KCL at the emitter of transistor \( Q_3 \) gives

\[
I_{E3} = I_{B1} + I_{B2} = \frac{I_{C1}}{\beta_F} + \frac{I_{C2}}{\beta_F} \tag{9.89}
\]

Since \( V_{\text{BE1}} = V_{\text{BE2}} \), it follows that \( I_{C1} = I_{C2} \). Thus, Eq. (9.89) becomes

\[
I_{E3} = \frac{2}{\beta_F} I_{C2} \tag{9.90}
\]

The base current of \( Q_3 \) is related to \( I_{E3} \) by

\[
I_{B3} = \frac{I_{E3}}{1 + \beta_F} = \frac{2}{\beta_F(1 + \beta_F)} I_{C2} \tag{9.91}
\]

Using KCL at the collector of \( Q_1 \) gives

\[
I_R = I_{C1} + I_{B3} = I_{C1} + \frac{2}{\beta_F(1 + \beta_F)} I_{C2} \tag{9.91}
\]
Since $I_{C1} = I_{C2}$, Eq. (9.91) gives the output current $I_O$ as

$$I_O = I_{C2} = \frac{I_R}{1 + 2/(\beta_F^2 + \beta_F)} \quad (9.92)$$

which indicates that the reference current $I_R$ is related to the output current $I_O$ by a factor of only $[1 + 2/(\beta_F^2 + \beta_F)]$. The reference current can be found from

$$I_R = \frac{V_{CC} - V_{BE1} - V_{BE3}}{R_1} \quad (9.93)$$

In the derivation of Eq. (9.92), the output resistances of the transistors are ignored. However, for a finite transistor output resistance, the collector current ratio in Eq. (9.86) is applicable.

**Output Resistance $R_o$**

The small-signal equivalent circuit for determining the output resistance is shown in Fig. 9.27(b). The output resistance $R_o$ is the same as $r_{o2}$; that is,

$$R_o = \frac{v_x}{i_x} = r_{o2} = \frac{V_A}{I_{C2}} \quad (9.94)$$

Thevenin’s equivalent voltage is given by

$$V_{Th} = I_oR_o = I_{C2}R_o = I_{C2}\frac{V_A}{I_{C2}} = V_A \quad (9.95)$$

**EXAMPLE 9.9**

**Designing a simple modified basic current source**

(a) Design the modified basic current source in Fig. 9.27(a) to give an output current $I_O = 5 \mu A$. The transistor parameters are $\beta_F = 100$, $V_{CC} = 30$ V, $V_{BE1} = V_{BE2} = V_{BE3} = 0.7$ V, and $V_A = 150$ V.

(b) Calculate the output resistance $R_o$, Thevenin’s equivalent voltage $V_{Th}$, and the collector current ratio if $V_{CE2} = 20$ V.

**SOLUTION**

$I_O = I_{C2} = 5 \mu A$, $V_{BE1} = V_{BE2} = V_{BE3} = V_{CE1} = 0.7$ V, and $V_A = 150$ V.

(a) From Eq. (9.92),

$$I_O = I_{C2} = \frac{I_R}{1 + 2/(\beta_F^2 + \beta_F)}$$

which, for $I_O = 5 \mu A$, gives $I_R = (5 \mu A)[1 + 2/(\beta_F^2 + \beta_F)] = 5 \mu A$. From Eq. (9.93),

$$R_1 = \frac{V_{CC} - V_{BE1} - V_{BE3}}{I_R} = \frac{30 - 0.7 - 0.7}{5 \mu A}$$
9.6.3 Widlar Current Source

Biasing currents of low magnitudes, typically on the order of 5 \( \mu \text{A} \), are required in a variety of applications. Currents of low magnitude can be obtained by inserting a resistance of moderate value in series with the emitter \( Q_2 \) in Fig. 9.25(a). A circuit with this modification, as shown in Fig. 9.28(a), is known as a Widlar current source. As a result of the addition of \( R \) into the circuit, \( I_{C2} \) is no longer equal to \( I_{R} \), and the value of \( I_{C2} \) can be made much smaller than that of \( I_{C1} \). This circuit can give currents in the microampere range, with acceptable circuit resistance values of less than 50 k\( \Omega \).

Using KVL around the B-E loop in Fig. 9.28(a) gives

\[ V_{BE1} - V_{BE2} - (I_{C2} + I_{B2})R_2 = 0 \]

Since \( I_{C2} >> I_{B2} \),

\[ V_{BE1} - V_{BE2} - I_{C2}R_2 \left( 1 + \frac{1}{\beta_F} \right) = 0 \] \hspace{1cm} (9.96)

Assuming that the transistors have infinite output resistances, \( V_A = \infty \), Eq. (9.85) becomes

\[ I_C = I_S \exp \left( \frac{V_{BE}}{V_T} \right) \]

which gives

\[ V_{BE} = V_T \ln \left( \frac{I_C}{I_S} \right) \] \hspace{1cm} (9.97)

Applying \( V_{BE} \) from Eq. (9.97) in Eq. (9.96), we get

\[ V_T \ln \left( \frac{I_{C1}}{I_{S1}} \right) - V_T \ln \left( \frac{I_{C2}}{I_{S2}} \right) - I_{C2}R_2 = 0 \hspace{1cm} \text{(for} \beta_F >> 1) \]
which, for identical transistors with $I_{S1} = I_{S2}$, can be simplified to

$$V_t \ln \left( \frac{I_{C1}}{I_{S1}} \times \frac{I_{S2}}{I_{C2}} \right) - I_{C2}R_2 = 0$$

This equation gives the relation of $I_{C2}$ to $I_{C1}$ and $R_2$ as

$$V_t \ln \left( \frac{I_{C1}}{I_{C2}} \right) = I_{C2}R_2 \quad (9.98)$$

The reference current $I_R$ can be found from

$$I_R = \frac{V_{CC} - V_{BE1}}{R_1} \quad (9.99)$$

which is also related to the base and collector currents; that is,

$$I_R = I_{C1} + I_{B1} + I_{B2}$$

$$= I_{C1} \left( 1 + \frac{1}{\beta_F} \right) + \frac{I_{C2}}{\beta_F} \quad (9.100)$$
Substituting $I_{C1}$ from Eq. (9.98) into the preceding equation, we get $I_R$ in terms of $I_{C2}$ and $R_2$ as

$$I_R = \frac{1 + \beta_F}{\beta_F} I_{C2} \exp \left( \frac{I_{C2} R_2}{V_T} \right) + \frac{I_{C2}}{\beta_F}$$  \hspace{1cm} (9.101)

Thus, $I_{C2}$ is a nonlinear function of $I_R$ and $R_2$. To find $I_{C2}$, we must solve this transcendental equation by trial and error, using known values of $I_R$ and $R_2$. However, for design purposes, $I_{C2}$ and $I_R$ are known, and it is necessary only to find the value of $R_2$.

**Output Resistance $R_o$**

The small-signal equivalent circuit for determining the output resistance is shown in Fig. 9.28(b). This circuit can be reduced to Fig. 9.28(c), where $R_e$ is the parallel equivalent of $\frac{r_\pi}{1 + g_{m1}}$, $r_{o1}$, and $R_1$; that is,

$$R_e = r_{\pi 1} \parallel \frac{1}{g_{m1}} \parallel R_1 \parallel r_{o1}$$  \hspace{1cm} (9.102)

Using Eq. (8.42), we can relate the input resistance $r_{\pi 1}$ to the transconductance $g_{m1}$ and the small-signal current gain $\beta_f (\approx \beta_F)$:

$$r_{\pi 1} = \frac{\beta_f (\approx \beta_F)}{g_{m1}}$$

Since $\beta_f \gg 1$, $r_{\pi 1} \gg \frac{1}{g_{m1}}$, and in general $R_1 \gg \frac{1}{g_{m1}}$, Eq. (9.102) can be approximated by

$$R_e \approx \frac{1}{g_{m1}}$$  \hspace{1cm} (9.103)

The collector current flows through the parallel combination of $R_2$ and $(r_{\pi 2} + R_e)$, so

$$v_3 = i_x [R_2 \parallel (r_{\pi 2} + R_e)]$$  \hspace{1cm} (9.104)

The voltage $v_3$ is related to $v_2$ by

$$v_2 = \frac{v_3 r_{\pi 2}}{r_{\pi 2} + R_e} = \frac{i_x r_{\pi 2} [R_2 \parallel (r_{\pi 2} + R_e)]}{r_{\pi 2} + R_e}$$  \hspace{1cm} (9.105)

The current $i_1$ thus becomes

$$i_1 = i_x - g_{m2} v_2$$

The test voltage $v_x$ is

$$v_x = v_3 + r_{o2} i_1 = v_3 + r_{o2} i_x - r_{o2} g_{m2} v_2$$

$$= v_3 + i_x r_{o2} + \frac{r_{o2} g_{m2} v_2 r_{\pi 2}}{r_{\pi 2} + R_e}$$  \hspace{1cm} (9.106)

Substituting $v_3$ from Eq. (9.104) into Eq. (9.106) and simplifying, we get the resistance at the collector of the transistor:

$$R_o = \frac{v_x}{i_x} = R_2 \parallel (r_{\pi 2} + R_e) + r_{o2} \left[ 1 + \frac{g_{m2} r_{\pi 2}}{r_{\pi 2} + R_e} R_2 \parallel (r_{\pi 2} + R_e) \right]$$  \hspace{1cm} (9.107)
Since \( r_{o2} \) is large, the first term is much smaller than the second one. If the first term is neglected, Eq. (9.107) can be reduced to

\[
R_o \approx r_{o2}\left[1 + \frac{g_{m2}r_{\pi2}}{r_{\pi2} + R_e} (r_{\pi2} + R_e)\right]
\]  

(9.108)

From Eq. (8.42), we get

\[
\frac{1}{g_{m1}} = \frac{1}{I_{C1}/V_T} = \frac{V_T}{I_{C1}}
\]  

(9.109)

\[
r_{\pi2} = \frac{\beta_F}{g_{m2}} = \frac{\beta_F}{I_{C2}/V_T} = \frac{V_T}{I_{C2}}\beta_F = \frac{V_T}{I_{C1}} \times \frac{I_{C1}}{I_{C2}} = \frac{\beta_F}{g_{m1}} \times \frac{I_{C1}}{I_{C2}}
\]  

(9.110)

Since \( I_{C1} \gg I_{C2} \), \( r_{\pi2} \gg \) and \( r_{\pi2} \gg R_e \), that is,

\[
R_e + r_{\pi2} \approx r_{\pi2}
\]  

(9.111)

Substituting \( R_e \) from Eq. (9.103) into Eq. (9.107) gives

\[
R_o \approx r_{o2}[1 + g_{m2}(R_2 || r_{\pi2})]
\]  

(9.112)

Since \( \beta_F \gg 1 \) and \( r_{\pi2} = \beta_F/g_{m2} \), Eq. (9.112) can be rewritten as

\[
R_o = r_{o2}\frac{g_{m2}R_2(1 + \beta_F) + \beta_F}{g_{m2}R_2 + \beta_F} = r_{o2}\left[1 + \frac{g_{m2}R_2}{1 + \frac{g_{m2}R_2}{\beta_F}}\right]
\]  

(9.113)

Generally, \( \beta_F \gg g_{m2}R_2 \), so Eq. (9.113) can be approximated by

\[
R_o \approx r_{o2}(1 + g_{m2}R_2)
\]  

(9.114)

\[
\approx r_{o2}\left(1 + \frac{I_{C2}R_2}{V_T}\right)
\]  

(9.115)

Thus, \( R_o \) depends on \( I_{C2}R_2 \), which is the DC voltage drop across \( R_2 \). The larger this drop is made, the higher the output resistance becomes. For the Widlar source, \( I_{C2}R_2 \) is limited to several hundred millivolts for practical current ratios, and the corresponding value of \( V_{Th} \) is limited to about \( 10V_A \).

**EXAMPLE 9.10**

**Designing a Widlar current source**

(a) Design the Widlar current source in Fig. 9.28(a) to give \( I_O = 5 \) \( \mu A \) and \( I_R = 1 \) mA. The parameters are \( V_{CC} = 30 V \), \( V_{BE1} = 0.7 V \), \( V_T = 26 mV \), \( V_A = 150 V \), and \( \beta_F = 100 \).

(b) Calculate the output resistance \( R_o \) and Thevenin’s voltage \( V_{Th} \). Assume \( V_T = 26 mV \).
9.6.4 Cascode Current Source

The emitter resistance $R_2$ in the Widlar current source in Fig. 9.28(a) can be replaced by a basic current source consisting of two transistors $Q_3$ and $Q_4$. This arrangement, shown in Fig. 9.29, will give a larger output resistance. In a cascodelike connection, two or more transistors are connected in series so that their collector biasing currents are almost identical (e.g., transistors $Q_1$ and $Q_3$), whereas in a cascode-like connection, the transistors operate in parallel fashion so that one transistor drives the other (e.g., $Q_3$ and $Q_4$). According to Eq. (9.55), the larger the output resistance, the greater the voltage gain of an amplifier.
becomes. Substituting the output resistance $r_{o4}$ of transistor Q4 for $R_2$ in Eq. (9.112) gives the output resistance of this cascode source:

$$R_o \approx r_{o2}[1 + g_{m2}(r_{o4}||r_{n2})] = r_{o2}(1 + g_{m2}r_{n2}) = r_{o2}(1 + \beta_F) \quad (9.116)$$

### 9.6.5 Wilson Current Source

A Wilson current source, shown in Fig. 9.30(a), also gives a high output resistance. However, the output current is approximately equal to the reference current. Base current $I_{B2}$, which is the difference between the reference current $I_R$ and the collector current $I_{C1}$, is multiplied by $(1 + \beta_F)$ to give $I_{E2}$, which flows in the diode-connected transistor Q3 and causes a collector current of the same magnitude to flow in Q1.

There is a feedback path that regulates $I_{C1}$, which is approximately equal to $I_{E2}$ and $I_{C2}$. Thus, $I_{C2}$ remains very nearly equal to $I_{C1}$ and nearly constant, giving a high output resistance:

$$I_{E2} = (1 + \beta_F)I_{B2} = (I_R - I_{C1})(1 + \beta_F)$$

Assuming that the transistors have infinite output resistances, $V_A = \infty$, and that the transistors are identical so that $I_{C1} = I_{C3}$, we can write

$$I_{E2} = I_{C3} + I_{B3} + I_{B1} = I_{C3}\left(1 + \frac{1}{\beta_F}\right) + \frac{I_{C1}}{\beta_F}$$

$$= I_{C3}\left(1 + \frac{2}{\beta_F}\right) \quad (9.117)$$

Using Eq. (9.117), we can relate $I_{C2}$ to $I_{E2}$ and $I_{C3}:

$$I_{C2} = \frac{I_{E2}}{1 + \beta_F} = I_{C3}\left(1 + \frac{2}{\beta_F}\right)\frac{\beta_F}{1 + \beta_F} = I_{C3}\frac{2 + \beta_F}{1 + \beta_F}$$

which gives

$$I_{C3} = I_{C2}\frac{1 + \beta_F}{2 + \beta_F} \quad (9.118)$$
$I_{C1}$ is related to the reference current $I_R$ by

$$I_{C1} = I_R - I_{B2} = I_R - \frac{I_{C2}}{\beta_F}$$  \hspace{1cm} (9.119)

Since $I_{C3} = I_{C1}$, equating Eq. (9.118) to Eq. (9.119) yields

$$I_{C1} = I_{C3} = I_{C2} \frac{1 + \beta_F}{2} = I_R - \frac{I_{C2}}{\beta_F}$$

which gives the output current $I_O = I_{C2}$ as

$$I_O = I_{C2} = I_R \left[ \frac{(2 + \beta_F)\beta_F}{\beta_F^2 + 2\beta_F + 2} \right] = I_R \left[ 1 - \frac{2}{\beta_F^2 + 2\beta_F + 2} \right]$$  \hspace{1cm} (9.120)

Since $\beta_F >> 1$, Eq. (9.120) can be approximated by

$$I_O \approx I_R$$  \hspace{1cm} (9.121)

Thus, the output current almost equals the reference current and is less sensitive to the current gain $\beta_F$, which varies in response to temperature changes.
Output Resistance \( R_0 \)

The small-signal AC equivalent circuit for determining the output resistance is shown in Fig. 9.30(b), which can be reduced to Fig. 9.30(c), where \( R_e \) is the parallel equivalent of \( r_{o1}, \frac{1}{g_{m1}}, \) and \( R_1 \) and where \( R_3 \) is the parallel equivalent of \( r_{\pi1}, r_{\pi3}, \frac{1}{g_{m3}}, \) and \( r_{o3} \). In general, \( \frac{1}{g_{m1}} \) is much smaller than \( r_{o1} \) and \( R_1 \), and \( \frac{1}{g_{m3}} \) is much smaller than \( r_{\pi1}, r_{\pi3}, \) and \( r_{o3} \). Thus,

\[
R_e = \frac{r_{o1}}{1} \parallel R_1 \tag{9.122}
\]

and

\[
R_3 = \frac{r_{\pi1}}{1} \parallel r_{\pi3} \parallel \frac{1}{g_{m3}} \parallel \frac{1}{r_{o3}} = \frac{1}{g_{m3}} \tag{9.123}
\]

Applying KVL to the circuit left of \( r_{\pi1} \), we get

\[
v_1 = i r_{\pi2} + (r_{o1} \parallel R_1)(i - g_{m1} v_1)
\]

which gives the equivalent resistance \( R'_e \) as

\[
R'_e = \frac{v_1}{i} = \frac{r_{\pi2} + (r_{o1} \parallel R_1)}{1 + (r_{o1} \parallel R_1) g_{m1}} \tag{9.124}
\]

Since \( r_{o1} \gg R_1 \),

\[
R'_e = \frac{r_{\pi2} + R_1}{1 + g_{m1} R_1}
\]

which, for \( R_1 \gg r_{\pi2} \) and \( g_{m1} R_1 \gg 1 \), can be simplified to

\[
R'_e \approx \frac{1}{g_{m1}}
\]

The output resistance can be found from Eq. (9.107) by substituting \( R_3 \) for \( R_2 \) and \( R'_e \) for \( (r_{\pi2} + R_e) \). That is,

\[
R_o = \frac{v_x}{i_x} = R_3 \parallel R'_e + r_{o2} \left[ 1 + \frac{g_{m2} r_{\pi2}}{R'_e} (R_2 \parallel R'_e) \right] \tag{9.125}
\]

Since \( r_{o2} \) is large, the first term is much smaller than the second one and can be ignored. Equation (9.124) can be approximated by

\[
R_o \approx r_{o2} \left[ 1 + \frac{g_{m2} r_{\pi2}}{R'_e} (R_3 \parallel R'_e) \right] \tag{9.126}
\]

Assuming \( (R_3 \parallel R'_e)/R'_e = 1/2 \), \( R'_e = 1/g_{m1} \), and \( g_{m1} = g_{m3} \), Eq. (9.126) becomes

\[
R_o \approx r_{o2} \left( 1 + \frac{g_{m2} r_{\pi2}}{2} \right) \approx r_{o2} \left( 1 + \frac{B_F}{2} \right) \tag{9.127}
\]
EXAMPLE 9.11

Designing a Wilson current source
(a) Design the Wilson current source in Fig. 9.30(a) to give $I_O = 5 \mu A$. The parameters are $V_{CC} = 30 V$, $V_{BE1} = V_{BE2} = V_{BE3} = 0.7 V$, $V_T = 26 mV$, $V_A = 150 V$, and $\beta_F = 100$. Assume that all transistors are identical.

(b) Calculate the output resistance $R_o$ and thevenin’s equivalent voltage $V_{Th}$.

(c) Use PSpice/SPICE to calculate the output current, the output resistance, and the reference current for $\beta_F = 100$ and 400. Assume that all transistors are identical, and $V_{CE} = 10 V$.

SOLUTION

$I_{C2} = 5 \mu A$, and $V_T = 26 mV$.

(a) From Eq. (9.120),

$$5 \mu A = I_R \left[ 1 - \frac{2}{(100^2 + 2 \times 100 + 2)} \right]$$

so $I_R = 5 \mu A$. The reference current is

$$I_R = \frac{V_{CC} - V_{BE1} - V_{BE2}}{R_1}$$

That is,

$$5 \mu A = \frac{30 - 0.7 - 0.7}{R_1}$$

which gives $R_1 = 5.72 M\Omega$.

(b) From Eq. (9.118),

$$I_{C3} = I_{C1} = \frac{5 \mu A \times (1 + 100)}{2 + 100} = 4.95 \mu A$$

Then $r_{o1} = r_{o3} = \frac{V_A}{I_{C1}} = \frac{150}{4.95 \mu A} = 30.3 M\Omega$

$$r_{o2} = \frac{V_A}{I_{C2}} = \frac{150}{5 \mu A} = 30 M\Omega$$

Since $1/g_{m1} = 1/g_{m3} = V_T/I_{C1} = 26 mV/4.95 \mu A = 5.25 k\Omega$,

$$g_{m1} = g_{m3} = 190.4 \mu A/V$$

Since $1/g_{m2} = 26 mV/5 \mu A = 5.2 k\Omega$,

$$g_{m2} = 192.3 \mu A/V$$
From Eq. (9.124),
\[ R'_o = \frac{520 \, \text{k} \Omega + (30.3 \, \text{M} \Omega \parallel 5.72 \, \text{M} \Omega)}{[1 + (30.3 \, \text{M} \Omega \parallel 5.72 \, \text{M} \Omega) \times 190.4 \, \mu \text{A/V}]} \]
\[ = 5.81 \, \text{k} \Omega \]
From Eq. (9.123),
\[ R_3 = 525 \, \text{k} \Omega \parallel 525 \, \text{k} \Omega \parallel 30 \, \text{M} \approx 5.15 \, \text{k} \Omega \]
From Eq. (9.126),
\[ R_o = (30 \, \text{M}) \left[ 1 + \frac{192.3 \, \mu \text{A} \times 520 \, \text{k} \Omega}{5.81 \, \text{k} \Omega} \times 5.15 \, \text{k} \Omega \parallel 5.81 \, \text{k} \Omega \right] = 1.44 \, \text{G} \Omega \]
and \( V_{th} = 1.44 \, \text{G} \Omega \times 5 \, \mu \text{A} = 7.2 \, \text{kV} \)

NOTE: If we use Eq. (9.127),
\[ R_o \approx (30 \, \text{M}) \left( 1 + \frac{100}{2} \right) = 1.53 \, \text{G} \Omega \]

(e) The Wilson current source for PSpice simulation is shown in Fig. 9.31. We will use parametric sweep for the model parameter \( \beta_F \) of the transistors. The voltage source \( V_y \) acts as an ammeter for the output current.

The results of simulation (.TF analysis) are as follows. (The hand calculations are shown in parentheses.)

For \( \beta_F = 100 \), the simulation gives

**VOLTAGE SOURCE CURRENTS**

<table>
<thead>
<tr>
<th>NAME</th>
<th>CURRENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>VCC</td>
<td>-5.022E-06</td>
</tr>
</tbody>
</table>

**FIGURE 9.31** Wilson current source for PSpice simulation
9.6.6 Multiple Current Sources

A DC reference current can be generated in one location and reproduced in another location for biasing amplifier circuits in ICs. A group of current sources with only one reference current is shown in Fig. 9.32. This is an extension of the modified basic current source in Fig. 9.25. Transistor Q1 and resistor R1 serve as the reference for current-sink transistors Q3 through Q6. Transistor Q2 supplies the total base currents for the transistors and makes the collector current of Q1 almost equal to the reference current $I_R$; that is,
I_{R1} = I_{C1}. The collector currents \( I_1 \) and \( I_2 \) will be mirrors of current \( I_R \). Since two transistors \( Q_5 \) and \( Q_6 \) are connected in parallel, \( I_3 \) will be two times \( I_R \) (i.e., \( I_3 = 2I_R \)). The parallel combination of \( Q_5 \) and \( Q_6 \) should be equivalent to a single transistor whose E-B junction has double the area of \( Q_1 \). Therefore, the emitter areas of transistors can be scaled in ICs so as to provide multiples of the reference current simply by designing the transistors so that they have an area ratio equal to the desired multiple.

**TABLE 9.1** Comparison of BJT current sources

<table>
<thead>
<tr>
<th>Current Source Type</th>
<th>Output Resistance ( R_o )</th>
<th>Beta ( (\beta_F) ) Dependency of ( I_o )</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic source</td>
<td>( r_o )</td>
<td>(-2/\beta_F)</td>
<td>Not suitable for low currents (i.e., typically less than 0.3 mA)</td>
</tr>
<tr>
<td>Modified source</td>
<td>( r_o )</td>
<td>(-2/(\beta_F^2 + \beta_F))</td>
<td>Not suitable for low currents (i.e., typically less than 0.3 mA)</td>
</tr>
<tr>
<td>Widlar source</td>
<td>( r_o(1 + g_m R_2) )</td>
<td>Nonlinear</td>
<td>Current as low as 5 ( \mu )A; higher output resistance</td>
</tr>
<tr>
<td>Cascode source</td>
<td>( r_o(1 + \beta_F) )</td>
<td>(-2/\beta_F)</td>
<td>Not suitable for low currents; higher output resistance</td>
</tr>
<tr>
<td>Wilson source</td>
<td>( r_o(1 + \beta_F/2) )</td>
<td>(-2/(\beta_F^2 + 2\beta_F + 2))</td>
<td>Not suitable for low currents; higher output resistance</td>
</tr>
</tbody>
</table>

**KEY POINTS OF SECTION 9.6**

- Transistors can generate the characteristics of a constant-current source. An ideal current source has a very high output resistance, and its output current is not sensitive to the transistor parameter \( \beta_F \).
- Table 9.1 compares various BJT current sources. The output resistances for different BJT sources are also summarized in the table.

**9.7 BJT Differential Amplifiers**

The BJT differential pairs are analogous to MOS differential pairs. Since BJTs, in general, have a higher transconductance compared to identical MOSFETs, BJT differential pairs can give a higher voltage gain, but they have limitations of a lower input resistance.

**9.7.1 BJT Differential Pair with Resistive Load**

An emitter-coupled pair, as shown in Fig. 9.33, is commonly used in a differential amplifier. The biasing current should be such that the transistors operate in the active regions. The DC biasing circuit, which is shown as a constant-current source, can be either a simple resistor, in which case the equivalent current generator will be zero, or a transistor current source, which is generally used in ICs.
DC Transfer Characteristics

The DC transfer characteristic, which gives the relation between the input and output voltages, can be determined from the large-signal analysis, and it should be linear over a wide range. The analysis can be simplified by making the following assumptions:

1. The output resistances of the transistors are infinite: \( r_o = \infty \).
2. The output resistance of the transistor current source is infinite: \( R_{EE} = \infty \).

Using KVL around the loop formed by the two input voltages and the two B-E junctions, we get

\[ v_{B1} - v_{BE1} + v_{BE2} - v_{B2} = 0 \]  \hspace{1cm} (9.128)

Assuming \( v_{BE1}, v_{BE2} \gg V_T \) and equal leakage currents \( I_{S1} = I_{S2} = I_S \) and using the transistor current equations, we get

\[ v_{BE1} = V_T \ln \left( \frac{i_{c1}}{I_S} \right) = V_T \ln \left( \frac{i_{c1}}{I_S} \right) \]  \hspace{1cm} (9.129)

\[ v_{BE2} = V_T \ln \left( \frac{i_{c2}}{I_S} \right) = V_T \ln \left( \frac{i_{c2}}{I_S} \right) \]  \hspace{1cm} (9.130)

Substituting Eqs. (9.129) and (9.130) into Eq. (9.128) gives

\[ v_{B1} - V_T \ln \left( \frac{i_{c1}}{I_S} \right) + V_T \ln \left( \frac{i_{c2}}{I_S} \right) - v_{B2} = 0 \]

That is,

\[ v_{B1} - v_{B2} = V_T \left[ \ln \left( \frac{i_{c1}}{I_S} \right) - \ln \left( \frac{i_{c2}}{I_S} \right) \right] \]
which can be simplified to

\[
\frac{i_{C1}}{i_{C2}} = \exp\left(\frac{v_{B1} - v_{B2}}{V_T}\right) = \exp\left(\frac{v_{id}}{V_T}\right)
\]  

(9.131)

where \(v_{id} = v_{B1} - v_{B2}\) is the differential DC input voltage.

Using KCL at the emitter terminal of the transistors, we get

\[
i_Q = \frac{1}{\alpha} (i_{C1} + i_{C2})
\]

(9.132)

where \(\alpha = \beta_E/(1 + \beta_F) \approx 1\). Solving Eqs. (9.131) and (9.132) for \(i_{C1}\) and \(i_{C2}\), we get

\[
i_{C1} = \frac{\alpha i_Q}{1 + \exp(-v_{id}/V_T)}
\]

\[
\approx \alpha i_Q \quad \text{for } v_{id} >> V_T
\]

(9.133)

(9.134)

\[
i_{C2} = \frac{\alpha i_Q}{1 + \exp(v_{id}/V_T)}
\]

\[
\approx 0 \quad \text{for } v_{id} >> V_T
\]

(9.135)

(9.136)

Thus, if \(i_{C1}\) increases, \(i_{C2}\) decreases such that \(i_{C1} + i_{C2} = \alpha i_Q = \alpha i_{EE}\) remains constant.

The plots of the two collector currents are shown as a function of \(v_{id}\) in Fig. 9.34(a). Notice that, for \(v_{id} >> V_T\), \(i_{C1}\) and \(i_{C2}\) become independent of \(v_{id}\) and all the currents flow through one of the transistors. For \(v_{id} \leq V_T\), \(i_{C1}\) and \(i_{C2}\) have an approximately linear relation. The differential voltage change \(\Delta v_{id}\) required to shift the current distribution from \(i_{C1} = 0.9i_Q\) and \(i_{C2} = 0.1i_Q\) to the opposite case, \(i_{C1} = 0.1i_Q\) and \(i_{C2} = 0.9i_Q\), is called the transition voltage; it will have a value of approximately \(2V_T = 52.6\) mV.

\[\text{FIGURE 9.34} \quad \text{Transfer characteristics of emitter-coupled pair}\]
The DC output voltages are

\[ v_{O1} = V_{CC} - iC_1R_C \]  \hspace{1cm} (9.137)

\[ v_{O2} = V_{CC} - iC_2R_C \]  \hspace{1cm} (9.138)

The differential DC output voltage is

\[ v_{od} = v_{O1} - v_{O2} = R_C(iC_2 - iC_1) \]

Substituting Eqs. (9.133) and (9.135) into the above equation and simplifying yields

\[ v_{od} = \alpha I_{EE}R_C \tanh \left(-\frac{v_{id}}{2V_T}\right) \]  \hspace{1cm} (9.139)

Since \( \tanh x \approx x \) for a small value of \( x \), Eq. (9.139) can be approximated by

\[ v_{od} \approx -\alpha I_{EE}R_C \left(\frac{v_{id}}{2V_T}\right) \]  \hspace{1cm} (9.140)

The plot of \( v_{od} \) as a function of \( v_{id} \) is shown in Fig. 9.34(b). If \( v_{id} \) is zero, \( v_{od} \) is also zero, and this feature allows direct coupling of cascoded stages without introducing DC offsets. Thus, the amplifier is a true differential, or difference, amplifier, responding only to the difference in the voltages applied to the two input terminals. If \( v_{B1} = -v_{B2} \), then \( v_{ic} = (v_{B1} + v_{B2})/2 \) is zero and there will be only a differential voltage. If, on the other hand, \( v_{B1} = v_{B2} \), then \( v_{id} \) is zero and there will be a pure common-mode voltage (i.e., no output voltage). However, the range of the differential input voltage \( v_{id} \) over which the emitter-coupled pair exhibits a linear characteristic is very small, typically two or three times \( V_T \). This range can be extended by inserting emitter degeneration resistors, as shown in Fig. 9.35(a); in this case, the range over which the characteristic is linear is approximately equal to \( I_QR_E \). The factor by which the voltage gain is reduced is approximately the same as the factor by which the input range is increased. The variations of \( v_{od} \) with several values of \( R_E \) are shown in Fig. 9.35(b).

---

**FIGURE 9.35** Emitter-coupled pair with degeneration resistors
Small-Signal Analysis

In studying a BJT differential amplifier, it is often of interest to examine the small-signal behavior for small DC differential voltages near zero, when the amplifier operates in the linear portion of the transfer characteristic. Circuit properties such as differential voltage gain \( A_d \), common-mode gain \( A_c \), common-mode rejection ratio (CMMR), common-mode input resistance \( R_{ic} \), and differential mode input resistance \( R_{id} \) can be determined from the small-signal analysis.

Small Differential Signal  Let us assume that the common-mode signal is zero, \( v_{ic} = 0 \), and only the differential input voltage \( v_{id} \) is applied. This situation is shown in Fig. 9.36(a). If the DC differential biasing is removed, we get the small-signal equivalent circuit shown in Fig. 9.36(b). The input voltage to one transistor is \( \frac{v_{id}}{2} \), and that to the other transistor is \( -\frac{v_{id}}{2} \). Assuming that the two transistors are identical and the circuit is balanced, the increase in voltage at the emitter junction due to \( \frac{v_{id}}{2} \) will be compensated for by an equal decrease in voltage due to \( -\frac{v_{id}}{2} \). As a result, the voltage at the emitters of the transistors will not vary at all. The emitter junction, which experiences no voltage variation, can be regarded as the ground potential. Thus, the resistor \( R_{EE} \) may be replaced by a short circuit, as shown in Fig. 9.36(b), which shows two identical sides.

The characteristic of a balanced amplifier can be determined from only one side of the amplifier. This simplified circuit, shown in Fig. 9.37(a), is known as a differential-mode half circuit; its small-signal equivalent circuit is shown in Fig. 9.37(b). The output voltage \( v_{od} \) is given by

\[
\frac{v_{od}}{2} = -g_m R_C \frac{v_{id}}{2}
\]

which gives the differential voltage gain \( A_d \) as

\[
A_d = \frac{v_{od}}{v_{id}} = -g_m R_C
\]  

(9.141)

NOTE  If we include the output resistance \( r_{o1} \) of \( Q_1 \), Eq. (9.141) becomes \( A_d = -g_m (R_C \parallel r_{o1}) \).
High values of $R_C$ and $g_m$ are required to give a high value of $A_d$. An active load (discussed in Sec. 9.6) rather than a discrete resistance ensures a high value of $R_C$ and hence of $A_d$. From Fig. 9.37(b), we can write

$$\frac{v_{id}}{2} = i_{bl}r_\pi$$

which gives the differential input resistance $R_{id}$ as

$$R_{id} = \frac{v_{id}}{i_{bl}} = 2r_\pi \quad (9.142)$$

A high value of $r_\pi (=V_T/I_C)$—that is, a low collector biasing current ($I_{C1} = I_{C2} = I_Q/2$)—is required to achieve a higher value of $R_{id}$.

**Small Common-Mode Signal** An emitter-coupled pair with only common-mode input $v_{ic}$ is shown in Fig. 9.38(a). If the DC common-mode biasing is removed, the result is the small-signal equivalent circuit shown in Fig. 9.38(b). Assuming that the two transistors are identical, the collector currents must be identical, and the voltage at the emitter junction will increase by the same amount in response to inputs at both transistors. Since the voltage across $R_{EE}$ will be the same for both inputs, the resistor $R_{EE}$ can be split into two parallel resistors, each of value $2R_{EE}$, as shown in Fig. 9.38(c). As a result of symmetry, no current will flow through the lead that connects the two sides, and $i_x = 0$. Thus, this lead can be disconnected without affecting the circuit behavior; as shown in Fig. 9.38(d), the two half circuits may be considered to be completely independent. The common-mode behavior can be determined from only one side, as shown in Fig. 9.39(a). The small-signal equivalent circuit is shown in Fig. 9.39(b), from which we get

$$v_{ic} = i_{b1}r_\pi + i_b(1 + \beta_F)2R_{EE} \quad (9.143)$$

which gives the common-mode input resistance $R_{ic}$ as

$$R_{ic} = \frac{v_{ic}}{i_{b1}} = r_\pi + (1 + \beta_F)2R_{EE} \quad (9.144)$$

Equation (9.144) will give a high value for $R_{ic}$, so $R_{ic}$ should be calculated using $r_\mu$ in the BJT model. The current through $r_\mu$ can be found from

$$i_\mu = \frac{v_{ic} - v_o}{r_\mu} = \frac{v_{ic} + \beta_F R_{ic}i_{b1}}{r_\mu}$$
(a) Common-mode circuit

(b) Small-signal equivalent circuit

(c) Small-signal equivalent circuit with split $R_{EE}$

(d) Independent half circuits

FIGURE 9.38  Emitter-coupled pair with common-mode input

(a) Common-mode half circuit

(b) Small-signal equivalent circuit

FIGURE 9.39  Common-mode half circuit
which, after substitution of $i_{b1} = v_{ic}/(r_\pi + (1 + \beta_F)2R_{EE})$, gives the effective resistance as

$$R_{i\mu} = \frac{r_\mu}{1 + \beta_F R_C/(r_\pi + (1 + \beta_F)2R_{EE})}$$

Thus, the common-mode resistance $R_{ic}$ becomes

$$R_{ic} = [r_\pi + 2R_{EE}(1 + \beta_F)]\frac{r_\mu}{r_\pi + (1 + \beta_F)2R_{EE}}$$

The common-mode output voltage $v_{oc}$ is

$$v_{oc} = -R_C i_{c1} = -R_C \beta_F i_{b1} = v_{ic}\left[\frac{-\beta_F R_C}{r_\pi + (1 + \beta_F)2R_{EE}}\right]$$

which gives the common-mode voltage gain $A_c$ as

$$A_c = \frac{v_{oc}}{v_{ic}} = \frac{-\beta_F R_C}{r_\pi + (1 + \beta_F)2R_{EE}}$$
$$A_c = \frac{-g_m R_C}{1 + 2g_m R_{EE}(1 + 1/\beta_F)}$$

**Small-Signal CMRR** From Eqs. (9.141) and (9.146), we can find the CMRR as

$$CMRR = \left|\frac{A_d}{A_c}\right| = 1 + 2g_m R_{EE}\left(1 + \frac{1}{\beta_F}\right)$$

$$\approx 1 + 2g_m R_{EE} \text{ for } \beta_F >> 1$$

which indicates that a high output resistance $R_{EE}$ on the biasing current source will improve the CMRR; that is, the value of $R_{EE}$ should be as large as possible. To obtain a high value of $g_m (=I_C/V_T)$, the collector biasing current ($I_{C1} = I_{C2} = I_Q/2 = I_{EE}/2$) should be made large.

The small-signal input currents, which will flow when both $v_{id}$ and $v_{ic}$ are applied, can be found by superposition. Since $R_{ic}$ is common to both $i_{b1}$ and $i_{b2}$, we get

$$i_{b1} = \frac{v_{ic}}{2R_{ic}} + \frac{v_{id}}{R_{id}}$$
$$i_{b2} = \frac{v_{ic}}{2R_{ic}} + \frac{v_{id}}{R_{id}}$$

The input resistance can thus be represented by the $\pi$-equivalent circuit of Fig. 9.40(a), where $R_{ic}$ is assumed to be much larger than $R_{id}$. The T-equivalent circuit is shown in Fig. 9.40(b), and its values are shown in Fig. 9.40(c).
EXAMPLE 9.12

Finding the performance parameters of an emitter-coupled pair  The parameters of the emitter-coupled pair in Fig. 9.36(a) are $\beta_F = 100$, $R_{EE} = 50 \text{k}\Omega$, $I_Q = 1 \text{mA}$, $V_{CC} = 15 \text{V}$, and $R_C = 10 \text{k}\Omega$.
(a) Calculate the DC collector currents through the transistors if $v_{id} = 5 \text{mV}$.
(b) Assuming $i_{C1} = i_{C2}$, calculate $A_d$, $A_c$, and CMRR; $R_{id}$ and $R_{ic}$; and the small-signal output voltage if $v_{B1} = 20 \text{mV}$ and $v_{B2} = 10 \text{mV}$. Assume $V_T = 26 \text{mV}$.

SOLUTION

$\alpha = \beta_F/(1 + \beta_F) = 100/(1 + 100) = 0.99$.

(a) From Eq. (9.133),
\[
    i_{C1} = \frac{\alpha I_Q}{1 + \exp(-v_{id}/V_T)} = \frac{0.99 \times 1 \text{mA}}{1 + \exp(-5 \text{mV}/26 \text{mV})} = 0.543 \text{mA}
\]
and
\[
    i_{C2} = I_Q - i_{C1} = 1 \text{mA} - 0.543 \text{mA} = 0.457 \text{mA}
\]

(b) We know that $i_{C1} = i_{C2} = 1 \text{mA}/2 = 0.5 \text{mA}$. Thus,
\[
    g_m = i_{C1}/V_T = \frac{0.5 \text{mA}}{26 \text{mV}} = 19.23 \text{mA/V}
\]

From Eq. (9.141),
\[
    A_d = -g_m R_C = -19.23 \text{mA} \times 10 \text{k}\Omega = -192.3 \text{V/V}
\]

From Eq. (9.146),
\[
    A_c = \frac{-g_m R_C}{1 + 2g_m R_{EE}(1 + 1/\beta_F)}
    = \frac{-19.23 \text{mA/V} \times 10 \text{k}\Omega}{1 + 2 \times 19.23 \text{mA/V} \times 50 \text{k}\Omega \times (1 + 1/100)} = -0.099 \text{V/V}
\]
Thus, CMRR = \( |A_d/A_c| = 192.3/0.099 = 1942.4 \) (or 65.77 dB). From Eq. (9.148),
\[
\text{CMRR} \approx 1 + 2g_mR_{\text{EE}} = 1 + 2 \times 19.23 \text{ m} \times 50 \text{ k} = 1924
\]

We know that \( r_\pi = \beta_F/g_m = 100/19.23 \text{ m} = 5.2 \text{ k} \). From Eq. (9.142),
\[
R_{\text{id}} = 2r_\pi = 2 \times 5.2 \text{ k} = 10.4 \text{ k} \Omega
\]

From Eq. (9.144),
\[
R_{\text{ic}} = r_\pi + (1 + \beta_F)2R_{\text{EE}} = 5.2 \text{ k} + (1 + 100) \times 2 \times 50 \text{ k} = 10.1 \text{ M} \Omega
\]

We know that \( v_{\text{id}} = 20 - 10 = 10 \text{ mV} \), and \( v_{\text{ic}} = (20 + 10)/2 = 15 \text{ mV} \). From Eq. (9.10),
\[
v_{\text{o}} = A_d v_{\text{id}} + A_c v_{\text{ic}} = -192.3 \times 10 \text{ mV} - 0.099 \times 15 \text{ mV} = -1924.5 \text{ mV}
\]

**EXAMPLE 9.13**

**Designing an emitter-coupled pair**

(a) Design an emitter-coupled pair as shown in Fig. 9.41 in which one input terminal is grounded. The output is taken from the collector of transistor Q1. The biasing current is \( I_{\text{EE}} = 1 \text{ mA} \), and \( V_{\text{CC}} = V_{\text{EE}} = 15 \text{ V} \). The transistors are identical. Assume \( V_{\text{BE}} = 0.7 \text{ V} \), \( V_T = 26 \text{ mV} \), \( V_A = \infty \), and \( \beta_F = 100 \). A small-signal voltage gain of \( A_1 = -250 \text{ V/V} \) is required.

(b) Calculate the design values of \( A_d, A_c, \) and CMRR.

**SOLUTION**

(a) \( I_{\text{EE}} = 1 \text{ mA} \). Since \( v_{B2} = 0 \), we can write
\[
R_{\text{EE}}I_{\text{EE}} + V_{\text{BE}} = V_{\text{EE}} = 15 \text{ V}
\]
Thus,
\[
R_{\text{EE}} = \frac{15 - 0.7}{1 \text{ m}} = 14.3 \text{ k} \Omega
\]

We have
\[
i_{\text{C1}} = i_{\text{C2}} = \frac{I_{\text{EE}}}{2} = \frac{1 \text{ mA}}{2} = 0.5 \text{ mA}
\]
and \( g_m = \frac{i_C}{V_T} = \frac{0.5 \text{ mA}}{26 \text{ mV}} = 19.23 \text{ mA/V} \)

Since \( v_{B2} = V_{B2} + v_{b2} = 0 \), \( v_{b2} = 0 \). Then

\[
V \equiv v_{id} = v_{b1} - v_{b2} = v_{b1}
\]

and

\[
v_{ic} = \frac{v_{b1} + v_{b2}}{2} = \frac{v_{b1}}{2}
\]

From Eq. (9.10), we get

\[
v_{o1} = A_cv_{ic} + \frac{A_dv_{id}}{2} = A_c \frac{v_{b1}}{2} + \frac{A_dv_{b1}}{2} = \frac{v_{b1}}{2}(A_c + A_d)
\]  \( (9.151) \)

Substituting for \( A_d \) and \( A_c \) from Eqs. (9.141) and (9.146) gives the voltage gain \( A_1 \) as

\[
A_1 = \frac{v_{o1}}{v_{b1}} = \frac{1}{2}(A_c + A_d) = \frac{1}{2} \left[ g_m R_C \left( 1 + \frac{1}{\beta_F} \right) + g_m R_C \right]
\]  \( (9.152) \)

Substituting \( A_1 = -250 \), \( g_m = 19.23 \text{ mA/V} \), \( R_{EE} = 14.3 \text{ k\Omega} \), and \( \beta_F = 100 \) into Eq. (9.152) gives \( R_C = 25.95 \text{ k\Omega} \).

(b) From Eq. (9.141),

\[
A_d = -g_m R_C = -19.23 \text{ mA} \times 25.95 \text{ k\Omega} = -499.01
\]

From Eq. (9.146),

\[
A_c = \frac{-19.23 \text{ mA/V} \times 25.95 \text{ k\Omega}}{1 + 2 \times 19.23 \text{ mA/V \times 14.3 k\Omega \times (1 + 1/100)}} = -0.897
\]

Thus, CMRR = \( \left| A_d/A_c \right| = 499.01/0.897 = 556.3 \) (or 54.91 dB).
9.7.2 BJT Differential Amplifiers with Basic Current Mirror Active Load

We saw in Sec. 9.7.1 that the differential gain of a differential pair with a resistive load $R_C$ is $-g_m R_C = -R_C I_C / V_T = -R_C I_Q / 2V_T$. In differential amplifiers, a very small value of biasing current $I_Q$, in the microampere range, is often used. As a result, a very large value of $R_C$, on the order of megaohms, will be required to give a substantial voltage gain. However, a large value of $R_C$ will cause a large DC voltage drop, reducing the collector voltage to $V_{CC} - R_C I_Q / 2$, which will be substantially less than $V_{CC}$. This low collector voltage will reduce the allowable input voltage range of the amplifier.

Active devices such as transistors occupy much less silicon area than medium-size or large resistors. In practical amplifiers, the load resistor $R_C$ is normally replaced by a constant-current source, which offers a very high load resistance to the amplifier and hence can give a high voltage gain. This type of load, known as an active load, has a small voltage drop, typically 0.7 V, and hence allows a wider input voltage range.

A differential amplifier with a basic current source as the active load is shown in Fig. 9.42. The active load consists of transistors $Q_3$ and $Q_4$. Since their B-E voltages are the same, their collector currents will be equal. That is, $i_{C3} \approx i_{C4}$. Thus, the current through $Q_4$ will be the mirror of the current through $Q_3$. Under quiescent conditions, the differential amplifiers will be balanced such that $I_{C1} = I_{C2}$. Since $I_{C1} = I_{C2}$ and $I_{C3} = I_{C4}$, we can find the quiescent load current:

$$I_O = I_{B3} + I_{B4} = \frac{I_{C3} + I_{C4}}{\beta_F} \approx \frac{I_{C1} + I_{C2}}{\beta_F} = \frac{\alpha I_Q}{\beta_F} = \frac{I_Q}{1 + \beta_F}$$

Since $\beta_F \gg 1$, $I_O$ will be very small.

![Figure 9.42](image_url)
**Small-Signal Analysis**

**Small Differential Signal** If the input voltages change by a small differential amount $v_{id}$, the collector current of Q1 will change by a small amount $g_m v_{id}/2$, and the collector current of Q3 will change by the same amount. Since $i_{C4}$ is a mirror of $i_{C3}$, the collector current of Q4 will change by $g_m v_{id}/2$. The B-E voltage of Q2 will decrease by $v_{id}/2$, causing its collector current to change by an amount $-g_m v_{id}/2$. The equivalent half circuit is shown in Fig. 9.43(a), and its small-signal equivalent appears in Fig. 9.43(b).

Using KCL at the collector of transistors Q2 and Q4, we can find the output voltage $v_o$ in terms of $v_{id}$:

$$-\frac{2g_m v_{id}}{2} = \frac{v_o}{r_{o2}} + \frac{v_o}{r_{o4}} + \frac{v_o}{R_L}$$

After simplification, we can find the differential voltage $A_d$ as

$$A_d = \frac{v_o}{v_{id}} = -g_m (\frac{1}{r_{o2}} + \frac{1}{r_{o4}} + \frac{1}{R_L})$$

(9.153)

For $r_{o2} = r_{o4} = r_o$ and the no-load condition ($R_L = \infty$), Eq. (9.153) becomes

$$A_d = -\frac{g_m r_o}{2}$$

(9.154)

Substituting $g_m = I_C/V_T$ and $r_o = V_A/I_C$ into Eq. (9.154), we get

$$A_d = -\left(\frac{I_C}{V_T}\right)\left(\frac{V_A}{2I_C}\right) = -\frac{V_A}{2V_T}$$

(9.155)

which is a constant for a given transistor. For typical values of $V_A = 100$ V and $V_T = 25.8$ mV, the differential gain becomes $A_d = -100 \text{ V}/(2 \times 25.8 \text{ mV}) = -1938$. Thus, we can see that with an active load, a very large voltage gain can be obtained with only a single amplifier stage. Also, the gain $A_d$ depends only on the physical parameters $V_A$ and $V_T$. Since $V_T$ is temperature dependent, $A_d$ will be temperature dependent too.
The differential input resistance $R_{id}$ is given by

$$R_{id} = \frac{v_{id}}{i_{d}} = 2r_{\pi} \quad \text{(for } I_C > 0) \quad (9.156)$$

Equation (9.156) is not valid for the no-load condition—that is, when $I_C = 0$. Substituting $r_{\pi} = \beta_F V_T/I_C$ into Eq. (9.156) gives

$$R_{id} = 2\frac{\beta_F V_T}{I_C} = 2 \times \frac{\beta_F V_T}{I_Q/2} = \frac{4\beta_F V_T}{I_Q} \quad (9.157)$$

which indicates that a lower biasing current $I_Q$ will give a higher value of $R_{id}$ while still maintaining a high voltage gain $A_d$. A very low value of $I_Q$, however, will affect the frequency and transient responses of the amplifier, which is undesirable. If a low biasing current is desired to achieve a high input resistance, then a MOSFET differential amplifier is preferable because the amplifier can be operated at a relatively higher value of biasing current without affecting the frequency and transient responses.

The output resistance $R_o$ is the parallel combination of $r_{o2}$ and $r_{o4}$; that is,

$$R_o = r_{o2} \parallel r_{o4} = \frac{r_o}{2} \quad (9.158)$$

(since $r_{o2} = r_{o4}$).

A differential amplifier is normally followed by other stages. The input resistance $R_L$ of the next stage acts as a load of the amplifier and hence influences the overall voltage gain. The amplifier is often represented as a transconductance amplifier so that the short-circuit current and the effect of load resistance on the output voltage can be determined easily. This arrangement is shown in Fig. 9.43(c). The total current is $2g_m v_{id}/2 = g_m v_{id}$, which gives the effective $G_m$ as

$$G_m = g_m = \frac{I_C}{V_T} = \frac{I_Q}{2V_T} \quad (9.159)$$

**Small Common-Mode Signal** The common-mode input resistance $R_{ic}$ can be found from Eq. (9.144). The approximate value of the common-mode gain $A_c$ can be found from Eq. (9.145) by replacing $R_C$ by $R_{o4}$. For practical amplifiers, $A_c$ is generally very small and can be ignored in finding the output voltage.

### 9.7.3 Differential Amplifier with Modified Current Mirror

Let us use the modified basic current shown in Fig. 9.27(a) as the active load. This arrangement is shown in Fig. 9.44. The active load consists of transistors $Q_3$, $Q_4$, and $Q_5$. The addition of $Q_5$ makes the ratio $i_{C4}/i_{C3}$ independent of current gain $\beta_F$, and $i_{C4}$ approximates a true mirror of $i_{C3}$. Transistors $Q_6$ and $Q_7$ belong to the second stage and act as the load of the differential amplifier. $Q_6$ and $Q_7$ form a compound transistor (Darlington pair), in which the emitter current of $Q_6$ becomes the base current of $Q_7$. As a result, the effective current gain becomes $\beta_F^2$ such that $i_{C7} = \beta_F^2 i_{B6}$. 

---

*Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.*
Under quiescent conditions with identical transistors, $I_{C1} = I_{C2} = I_Q$, $I_{C1} = I_{C2}$, and $I_{C3} = I_{C4}$. Thus, $I_{B5} = I_{B6}$, so $I_{E5} = I_{E6}$; that is,

$$I_{B7} = I_{E6} = I_{E5} = I_{B3} + I_{B4} = \frac{I_{C3} + I_{C4}}{\beta_F} = \frac{I_Q}{\beta_F}$$

Thus, the collector current of $Q_7$ becomes $I_{C7} = \beta_F I_{B7} = I_Q$. This current mirror causes the quiescent current of the next stage to have the same value as the differential amplifier while having the same performance as the amplifier of Fig. 9.42.

**EXAMPLE 9.14**

Analyzing a BJT differential amplifier with a current mirror active load  The parameters of the differential amplifier in Fig. 9.44 are $\beta_F = 100$, $I_Q = 20 \mu A$, and $V_{CC} = 15 V$. Calculate $A_d$, $R_{id}$, $R_o$, and the overall voltage gain with load $A_{d\text{(load)}}$. Assume $V_T = 26 mV$ and $V_A = 100 V$.

**SOLUTION**

We have

$$I_{C1} = I_{C2} = I_{C4} = \frac{20 \mu A}{2} = 10 \mu A$$

$$I_{B6} = \frac{I_Q}{\beta_F \beta_{F8}} = \frac{20 \mu A}{100 \times 100} = 2 nA$$
9.7.4 Cascode Differential Amplifier

Notice from Eq. (9.154) that the differential gain increases with the output resistance \( R_o \) of the differential amplifier. Transistors are often connected in cascode configurations to increase the output resistance and also to improve the frequency response. A common modification to Fig. 9.44 is shown in Fig. 9.45. Transistors Q5 and Q6 are connected in a common-base configuration and form a common-base differential stage. Note that Q5 and Q6 must be biased by connecting the base terminals of Q5 and Q6 to the DC supply \( V_{CC} \) through a potential divider.

The equivalent half circuit is shown in Fig. 9.46(a), which can be simplified to Fig. 9.46(b). Q4 is replaced by its equivalent circuit.

\[
\frac{g_m}{V_T} = \frac{10 \, \mu A}{26 \, mV} = 387.6 \, \mu A/V
\]

\[
r_o = r_o = \frac{100 \, V}{10 \, \mu A} = 10 \, M\Omega
\]

\[
r_o = \frac{\beta F V_T}{I_C} = \frac{100 \times 26 \, mV}{10 \, \mu A} = 260 \, k\Omega
\]

From Eq. (9.155),

\[
A_d = -\frac{V_A}{2V_T} = -\frac{100 \, V}{2 \times 26 \, mV} = -1923 \, V/V
\]

From Eq. (9.156),

\[
R_{ad} = 2r_o = 2 \times 260 \, k\Omega = 520 \, k\Omega
\]

and \( R_o = r_o \parallel r_{o4} = \frac{r_o}{2} = \frac{10 \, M\Omega}{2} = 5 \, M\Omega \)

Since Q6 and Q7 form a compound transistor, its effective B-E voltage is that of two B-E junctions in series. Thus,

\[
R_L = 2r_{\pi6} = \frac{2V_T}{I_{B6}} = \frac{2 \times 26 \, mV}{2 \, nA} = 26 \, M\Omega
\]

The effective transconductance is \( G_m = g_m = 387.6 \, \mu A/V \). Thus, the overall voltage gain with load \( A_{d(\text{load})} \) is

\[
A_{d(\text{load})} = -G_m(R_o \parallel R_L) = -387.6 \, \mu A/V \times (5 \, M\Omega \parallel 26 \, M\Omega) = -1625 \, V/V
\]

### 9.7.4 Cascode Differential Amplifier

Notice from Eq. (9.154) that the differential gain increases with the output resistance \( R_o \) of the differential amplifier. Transistors are often connected in cascode configurations to increase the output resistance and also to improve the frequency response. A common modification to Fig. 9.44 is shown in Fig. 9.45. Transistors Q5 and Q6 are connected in a common-base configuration and form a common-base differential stage. Note that Q5 and Q6 must be biased by connecting the base terminals of Q5 and Q6 to the DC supply \( V_{CC} \) through a potential divider.

The equivalent half circuit is shown in Fig. 9.46(a), which can be simplified to Fig. 9.46(b). Q4 is replaced by its equivalent circuit. \( R'_o \), which is the equivalent output resistance of the Q2 and Q6 combination, can be determined from the test circuit shown in Fig. 9.46(c). The emitter resistance of Q6 is the parallel combination of \( r_{\pi6} \) and \( r_{o2} \); that is,

\[
R'_E = r_{\pi6} \parallel r_{o2} \approx r_{\pi6}
\]

(since \( r_{o2} >> r_{\pi6} \)). Using Eq. (9.114), we get \( R'_o \):

\[
R'_o = r_{o6}(1 + g_m6R'_E) = r_{o6}(1 + g_m6r_{\pi6})
\]

\[
= r_{o6}(1 + \beta F6) \approx \beta F6r_{o6}
\]

\[\text{(9.161)}\]
Since all devices are biased at the same current \( r_{o6} = r_{o3} = r_{o2} = r_o \), all \( \beta_F \) are the same. Thus,

\[
R'_o = \beta_F r_o \tag{9.162}
\]

The output resistance of the amplifier becomes

\[
R_o = r_o \parallel R'_o = r_o \parallel \beta_F r_o \approx r_o \tag{9.163}
\]

The differential gain becomes

\[
A_d = -g_m R_o = -g_m r_o \tag{9.164}
\]
Substituting \( g_m = I_C/V_T \) and \( r_o = V_A/I_C \) into Eq. (9.164), we get
\[
A_d = -\frac{V_A}{V_T} \quad (9.165)
\]
Thus, the differential gain is twice that of the modified current mirror amplifier in Fig. 9.44. Transistor Q6 offers a very high resistance seen from the output side.

Transistor Q6 offers a load resistance \( R_{L1} \) seen from the collector of Q2. \( R_{L1} \) can be determined from the test circuit shown in Fig. 9.47; it is the parallel combination of \( r_{o2}, r_{o6} \), and \( r_{\pi6} \) (referred to as the emitter of transistor Q6). That is,
\[
R_{L1} = r_{o2} \| \frac{r_{\pi6}}{1 + \beta_F} = \frac{r_{o6}}{1 + \beta_F} \quad (9.166)
\]
which will have a low value. Thus, transistor Q5 (or Q6) acts as the current buffer, accepting the signal current \( (g_mvi_d/2) \) from the collector of Q1 at a low resistance \( R_{L1} \) and delivering an almost equal current \( (g_mvi_d/2) \) to the load at a very high resistance \( R' \). The low resistance at the collector of Q2 improves the frequency response. Additionally, the internal capacitances of Q6 (from collector to base and emitter to base) are connected to the ground, and there is no Miller multiplication effect. As a result, the high cutoff frequency is increased.

### KEY POINTS OF SECTION 9.7

- The DC transfer characteristic of a BJT differential pair is nonlinear. However, a BJT differential pair is normally operated in the linear region where \( v_{id} < V_T \).
- The values of the load resistance \( R_C \) and the current source resistance \( R_{EE} \) should be large for large values of differential gain and CMRR, respectively. A discrete resistor limits the maximum differential input voltage range.
- The voltage gain is \( g_mR_C \) for the difference output and \( g_mR_C/2 \) for the single-sided output. A small biasing current \( I_Q \) increases the transconductance and the voltage gain; however, the differential input resistance is reduced.
- An active load increases the differential gain considerably. The gain is directly proportional to the ratio \( V_A/V_T \). Also, a cascadeplike connection increases the gain.
9.8 BiCMOS Differential Amplifiers

There are two established silicon technologies for the design of integrated circuits: BJT technology and CMOS technology using NMOS and PMOS. Each type has distinct advantages and disadvantages. An emerging technology called BiCMOS technology uses the bipolar-CMOS (BiCMOS) process and combines n- and p-channel MOSFETs together with either npn or pnp BJTs (or sometimes both) on the same semiconductor chip. A BiCMOS circuit utilizes the advantages of each type to provide the desired circuit functions.

9.8.1 BJT versus CMOS Amplifiers

We will begin by considering the basic BJT and CMOS amplifiers. Figure 9.48(a) shows a BJT amplifier with active load that is similar to the half circuit of a BJT differential pair. The output resistance is given by

$$R_o = r_o = \frac{V_A}{I_C}$$  \hspace{1cm} (9.167)

where $V_A$ is the Early voltage. Typically, $V_A = 50$ V and $I_C = 5$ μA, so $r_o = 50$ V/(5 μA) = 10 MΩ. Assuming that the current source load has infinite resistance, the voltage gain is given by

$$A_d = -g_mR_o = -g_mr_o$$

$$= -\frac{I_C}{V_T} \times \frac{V_A}{I_C} = -\frac{V_A}{V_T}$$  \hspace{1cm} (9.168)

which is independent of the biasing current $I_C$. Since typically $V_A = 50$ V and $V_T = 25.8$ mV at room temperature, the intrinsic gain of a BJT amplifier is $A_d = -1938$ V/V. The input resistance is

$$R_i = r_\pi = \beta_F \frac{V_T}{I_C}$$  \hspace{1cm} (9.169)

which is generally low. Typically, $\beta_F = 60$ and $I_C = 5$ μA, so $R_i = 60 \times 25$ mV/5 μA = 300 kΩ. Although lowering the value of $I_C$ will increase $R_i$, it will lower the $g_m$ of the transistor and hence the upper frequency limit of the amplifier—that is, $f_T$ in Eq. (8.168).
Now consider the MOSFET amplifier shown in Fig. 9.48(b). The output resistance is given by

\[ R_o = r_o = \frac{V_M}{I_D} \tag{9.170} \]

where \( V_M \) is the channel modulation voltage. Typically, \( V_A = 20 \text{ V} \) and \( I_D = 5 \mu\text{A} \), so \( r_o = 20 \text{ V}/(5 \mu\text{A}) = 4 \text{ M}\Omega \). Assuming that the current source load has infinite resistance, the voltage gain is given by

\[ A_d = -\frac{g_m R_o}{g_m} = -g_m R_o \]

\[ = -2 \sqrt{I_D K_n \left( \frac{V_M}{I_D} \right)} = -2 \left( \frac{K_n}{I_D} \right)^{1/2} V_M \tag{9.171} \]

Thus, the gain is inversely proportional to \( \sqrt{I_D} \) and will increase as the biasing current is lowered. Decreasing the DC biasing current, however, reduces the amplifier bandwidth. For example, if \( I_D = 5 \mu\text{A}, \ V_M = 20 \text{ V}, \) and \( K_n = 25 \mu\text{A}/\text{V}^2, \ A_d = -89 \text{ V/V}. \)

In summary, for the same value of biasing current \( I_D \), the values of \( g_m \) and \( r_o \) are much larger for a BJT amplifier than for a MOSFET amplifier—typically 2.5 times as large. The \( V_A \) of a BJT amplifier (typically 50 V) is greater than the \( V_M \) of a MOSFET (typically 20 V). The voltage gain of a BJT amplifier is greater than that of a MOSFET amplifier by a factor of about 10. However, a MOSFET amplifier has a practically infinite input resistance.

### 9.8.2 BiCMOS Amplifiers

A BiCMOS amplifier combines the best features of BJT and MOSFET amplifiers. It consists of cascode-like connections of BJTs and MOSFETs. The basic half circuit for a BiCMOS configuration is shown in Fig. 9.49(a). The MOSFET M1 acts as the driving device, and the BJT Q1 acts as the load. The cascode configuration is shown in Fig. 9.49(b). The BJT Q1 acts as the driving device, and the MOSFET M1 and

---

(a) Basic BiCMOS configuration  
(b) Cascode BiCMOS configuration  
(c) BiCMOS amplifier

**FIGURE 9.49** BiCMOS amplifier
the BJT Q2 act as the load. A BiCMOS amplifier is shown in Fig. 9.49(c); this amplifier is identical to the CMOS amplifier in Fig. 9.14. As its abbreviation indicates, the current mirror in a BiCMOS is bipolar rather than unipolar, as in MOS devices. Transistors M1 and M2 are the amplifying devices. Transistor Q4 acts as the load of transistor M2. The output resistance \( R_o \) is the parallel combination of the output resistance \( r_{o2} = 2V_M/I_Q \) for transistor M2 and the output resistance \( r_{o4} = 2V_A/I_Q \) for transistor Q4; that is,

\[
R_o = R'_o \parallel R'_D = r_{o2} \parallel r_{o4}
\]

\[
= \frac{2V_AV_M}{I_Q(V_A + V_M)}
\]

(9.172)

(9.173)

The differential voltage gain \( A_d \) is given by

\[
A_d = \frac{V_o}{V_{id}} = -g_m(r_{o2} \parallel r_{o4})
\]

(9.174)

where \( g_m \), which is the transconductance of the driving MOSFET M2, is given by Eq. (9.54). Substituting \((r_{o4} \parallel r_{o2}) = 2V_AV_M/I_Q(V_A + V_M) \) and \( g_m = \sqrt{2K_nI_Q} \) into Eq. (9.174), we get

\[
A_d = -\sqrt{2K_nI_Q} \frac{2V_MV_A}{I_Q(V_A + V_M)}
\]

(9.175)

Since \( r_{o4} > r_{o2} \), \( R_o \) and \( A_d \) will be greater for a BiCMOS amplifier than for a CMOS amplifier.

### 9.8.3 Cascode BiCMOS Amplifiers

Like the cascode BJT amplifier in Fig. 9.45, BiCMOS amplifiers can use cascode-like transistors to increase the voltage gain. This arrangement is shown in Fig. 9.50. Transistors Q5 and Q6 are connected in a CB configuration and form a CB differential stage. As shown in Fig. 9.46(c), the emitter resistance \( R_E \) of Q6 is the parallel combination of \( r_{\pi6} \) and \( r_{o2} \); that is,

\[
R'_E = r_{\pi6} \parallel r_{o2} \approx r_{\pi6}
\]

(9.176)

(since \( r_{o2} \gg \pi_6 \)). Using Eq. (9.114), we get \( R'_o \) as

\[
R'_o = r_{o6}(1 + g_mR'_E) = r_{o6}(1 + g_m\pi_6)
\]

\[
= r_{o6}(1 + \beta_Fr_o) \approx \beta_Fr_{o6}
\]

(9.177)

Since all devices are biased at the same current \( r_{o6} = r_{o4} = r_o \), all \( \beta_F \) are the same. Thus, \( R'_o \), which is the equivalent output resistance of the Q2 and Q6 combination, is given by

\[
R'_o = \beta_Fr_o
\]

(9.178)

Thus, the output resistance of the amplifier becomes

\[
R_o = r_o \parallel R'_o = r_o \parallel \beta_Fr_o \approx r_o
\]

(9.179)

The differential voltage gain becomes

\[
A_d = -g_mR_o = -g_m1r_o
\]

(9.180)
Substituting $g_m = \sqrt{2K_n I_Q}$ and $r_o = 2V_A/I_Q$ into the preceding equation, we get

$$A_d = -\sqrt{2K_n I_Q} \frac{2V_A}{I_Q} = -2V_A \sqrt{\frac{2K_n}{I_Q}}$$

which gives a greater voltage gain than Eq. (9.175).

### 9.8.4 Double-Cascode BiCMOS Amplifiers

A very high gain can be obtained by double cascoding, as shown in Fig. 9.51. There are two cascode connections: the first connects BJTs Q1, Q2, Q3, and Q4, and the second connects MOSFETs M5 and M6. The resistance $R_S$ looking from the collector of Q4 will be

$$R_S = r_o (1 + \beta_{F4}) \approx \beta_{F4} r_o$$

which will be the source resistance of M6. Using Eq. (9.33), we can find the output resistance $R'_o$ of M6 as

$$R'_o \approx r_{o6} (1 + g_{m6} R_S) = r_{o6} (1 + g_{m6} \beta_{F4} r_o)$$

$$\approx r_{o6} g_{m6} \beta_{F4} r_o$$

Like Q2 and Q4, transistors Q8 and Q10 are also connected in cascode, and their equations will be similar to those for Q2 and Q4. From Eq. (9.177), the resistance looking from the collector of Q8 is

$$R'_D = r_{o8} (1 + \beta_{F8}) \approx \beta_{F8} r_{o8}$$
Thus, the output resistance of the amplifier becomes

\[ R_o = \frac{1}{H_11005} \left( \frac{R_o}{H_11005} + \frac{D}{R_o} \right) \left( \frac{S}{R_o} \right) \left( \frac{1}{H_11005} \right) \left( \frac{F}{R_o} \right) \left( \frac{1}{H_9252} \right) \left( \frac{F}{R_o} \right) \left( \frac{1}{H_11005} \right) \left( \frac{F}{R_o} \right) \text{(9.186)} \]

The differential voltage gain becomes

\[ A_d = \frac{\frac{g_m}{H_11005} \frac{R_o}{H_11005} \frac{S}{R_o} \frac{1}{H_9252} \frac{F}{R_o} \frac{1}{H_11005} \frac{F}{R_o} \frac{1}{H_9252} \frac{F}{R_o}}{S} \text{(9.188)} \]

\[ A_d = -g_m R_o = -g_m \left( r_{o6} g_m 6 \beta_F r_{o4} \right) \left( \beta_F r_{o8} \right) \text{(9.189)} \]

where \( g_m = I_Q/2V_T \). Equation (9.189) shows a considerable increase in the voltage gain.

**EXAMPLE 9.15**

**Analyzing BiCMOS amplifiers** The DC biasing current of a BiCMOS amplifier is kept constant at \( I_Q = 10 \mu A \). All bipolar transistors are identical, with \( V_A = 50 \text{ V} \) and \( \beta_F = 40 \). Also, the MOS transistors are identical, with \( V_M = 20 \text{ V} \), \( K_n = 25 \mu A/V^2 \), \( W = 30 \mu m \), and \( L = 10 \mu m \). Assume \( V_T = 25.8 \text{ mV} \).
Determine the differential voltage gain $A_d$ for single-ended output (a) for the BiCMOS amplifier in Fig. 9.49(c), (b) for the cascode BiCMOS amplifier in Fig. 9.50, and (c) for the double-cascode BiCMOS amplifier in Fig. 9.51.

**SOLUTION**

$V_A = 50$ V, $\beta_F = 40$, $V_M = 20$ V, $K_n = 25$ $\mu$A/V$^2$, $W = 30$ $\mu$m, $L = 10$ $\mu$m, and $I_D = I_C = I_Q/2 = 10$ $\mu$A/2 = 5 $\mu$A.

(a) We have

\[
r_{o2} = \frac{2V_M}{I_Q} = \frac{2 \times 20 \text{ V}}{10 \text{ $\mu$A}} = 4 \text{ M$\Omega$}
\]

\[
r_{o4} = \frac{2V_A}{I_Q} = \frac{2 \times 50 \text{ V}}{10 \text{ $\mu$A}} = 10 \text{ M$\Omega$}
\]

\[
R_o = r_{o2} \parallel r_{o4} = 4 \text{ M$\Omega$} \parallel 10 \text{ M$\Omega$} = 2.86 \text{ M$\Omega$}
\]

\[
g_{m2} = \sqrt{2K_mI_Q} = \sqrt{2 \times 25 \mu \times 10 \mu} = 22.36 \mu$A/V
\]

Thus, the differential voltage becomes

\[
A_d = -g_{m2}R_o = -22.36 \mu \times 2.86 \text{ M} = -63.9
\]

(b) We have

\[
r_{o4} = r_{o6} = \frac{2V_A}{I_Q} = \frac{2 \times 50 \text{ V}}{10 \text{ $\mu$A}} = 10 \text{ M$\Omega$}
\]

\[
R'_o = \beta_Fr_{o6} = 40 \times 10 \text{ M$\Omega$} = 400 \text{ M$\Omega$}
\]

\[
R_o = r_{o4} \parallel R'_o = 10 \text{ M} \parallel 400 \text{ M} = 10 \text{ M$\Omega$}
\]

\[
g_{m2} = \sqrt{2K_mI_Q} = \sqrt{2 \times 25 \mu \times 10 \mu} = 22.36 \mu$A/V
\]

Thus, the differential voltage becomes

\[
A_d = -g_{m2}R_o = -22.36 \mu \times 10 \text{ M} = -223.6 \text{ V/V}
\]

(c) From Eq. (9.187),

\[
R_o = \beta_Fr_{o4} = 40 \times 10 \text{ M} = 400 \text{ M$\Omega$}
\]

\[
g_{m2} = \frac{I_Q}{2V_T} = \frac{10 \text{ $\mu$A}}{2 \times 25.8 \text{ mV}} = 193.8 \mu$A/V
\]

Thus, the differential voltage gain becomes

\[
A_d = -g_{m2}R_o = -193.8 \mu$A/V \times 400 \text{ M$\Omega$} = -77,520 \text{ V/V}
\]

which is considerably larger than for the other two configurations.
9.9 Frequency Response of Differential Amplifiers

The transistors (either MOSFETs or BJTs) in differential amplifiers have capacitance—hence, the gain of such amplifiers will be frequency dependent [5–7]. The techniques in Chapter 2 for analyzing frequency response can be applied to find the frequency response of differential amplifiers. In this section, we consider only the differential gain with passive and active loads. \( R_C \) represents the output resistance of the active source.

9.9.1 Frequency Response with Resistive Load

Let us first consider the differential half circuit shown in Fig. 9.52(a). Replacing the transistor by its frequency model gives the circuit in Fig. 9.52(b). The transistor can be either a BJT or a MOSFET. We can apply the same model and analysis for MOSFETs by substituting \( r_{\pi} \) and \( C_{\pi} \) parameters of the transistor model are sealed by a factor of 2. Since \( g_m(R_C || r_{o1}) \gg 1 \), \( C_{\mu} \) will dominate the high cutoff frequency \( \omega_H \). If we replace \( C_{\mu}/2 \) by its effective Miller capacitance \( C_M \), we get the equivalent circuit shown in Fig. 9.52(c) and Eq. (2.98); that is,

\[
C_M = \left( \frac{C_{\mu}}{2} \right) [1 + g_m(R_C || r_{o1})]
\]  

\[(9.190)\]

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
Thus, the high cutoff frequency $\omega_H$ is given by

$$\omega_H = \frac{1}{2r_\pi(C_{\pi}/2 + C_M)} = \frac{1}{2r_\pi(C_{\pi}/2 + (C_{\mu}/2)[1 + g_m(R_C \parallel r_{o1})])}$$  \hspace{1cm} (9.191)$$

which, for $R_C = r_{o1} = r_o$ and $g_m(R_C \parallel r_{o1}) \gg 1$, can be approximated by

$$\omega_H \approx \frac{1}{2r_\pi(C_{\mu}/2)g_m r_o/2} = \frac{2}{r_\pi(C_{\mu}g_m r_o)} = \frac{2}{r_\pi C_{\mu}g_m R_C}$$  \hspace{1cm} (9.192)$$

The signal source resistance $R_s = 0$. The effective resistance for $C_{\pi}/2$ will be $R_C = (2r_\pi \parallel R_s) = 0$. Thus, the break frequency due to $C_{\pi}/2$ will be at infinity and will not influence $\omega_H$ in Eq. (9.192). Therefore, the frequency-dependent differential gain is given by

$$A_d(j\omega) = \frac{A_{do}}{1 + j\omega/\omega_H}$$  \hspace{1cm} (9.193)$$

where $A_{do} = g_m(r_{o2} \parallel r_{o4})$ is the low-frequency differential gain with differential output voltage.

### 9.9.2 Frequency Response with Active Load

Now let us consider the output circuit for the amplifier with an active load, as shown in Fig. 9.42. The AC equivalent of that circuit is shown in Fig. 9.53(a). Replacing the transistors by their frequency model, we get Fig. 9.53(b). Since the effective transconductance is $2g_m$, the Miller capacitance $C_M$ from Eq. (2.98) becomes

$$C_M = \left(\frac{C_{\mu}}{2}\right)[1 + 2g_m(r_{o2} \parallel r_{o4})]$$  \hspace{1cm} (9.194)$$

Thus, the high cutoff frequency $\omega_H$ is given by

$$\omega_H = \frac{1}{2r_\pi(C_{\pi}/2 + (C_{\mu}/2)[1 + 2g_m(r_{o2} \parallel r_{o4})])}$$  \hspace{1cm} (9.195)$$

---

**FIGURE 9.53** Differential amplifier with active load and high-frequency equivalent

---

---
which, for $r_{o2} = r_{o4} = r_o$ and $2g_m(r_{o2} || r_{o4}) >> 1$, can be approximated by

$$\omega_H = \frac{1}{2r_\pi/(C_\mu/2)g_mr_o} = \frac{1}{r_\pi C_\mu g_mr_o}$$

Therefore, the frequency-dependent differential gain is given by

$$A_d(j\omega) = \frac{A_{do}}{1 + j\omega/\omega_H}$$

where $A_{do} = -2g_m(r_{o2} || r_{o4})$ is the low-frequency differential gain with an active load and a single-sided output.

Notice from Eq. (9.192) that the high cutoff frequency is double that for an amplifier with an active load and single-sided output. However, this high cutoff frequency for an amplifier with a passive load is obtained at the expense of the output voltage gain. For this reason, the first stage of a wide-band amplifier often uses a balanced circuit with differential voltage when the CMRR is not the prime consideration.

**KEY POINTS OF SECTION 9.9**

- The transistors in differential amplifiers have capacitance; hence, the gain of such amplifiers will be frequency dependent and exhibit a high-pass characteristic.
- The high cutoff frequency for an amplifier with a passive load is obtained at the expense of the output voltage gain. For this reason, the first stage of a wide-band amplifier often uses a balanced circuit with differential voltage.

### 9.10 Design of Differential Amplifiers

Differential amplifiers are used as the input stage and are designed for a high differential gain, a high CMRR, and high differential and common-mode input resistances. The design of a differential stage involves the following steps:

**Step 1.** Identify the specifications: the differential gain $A_d$, the CMRR, the input resistance $R_{id}$, and the DC supply voltages $V_{CC}$ and $V_{EE}$ (or $V_{DD}$ and $V_{SS}$).

**Step 2.** Select the type of differential amplifier (BJT, CMOS, or BiCMOS).

**Step 3.** Determine the biasing current $I_Q$ required for the desired differential gain and input resistance.

**Step 4.** Choose the type of current source (BJT or MOSFET) and determine its component ratings. Use the standard values of components. (See Appendix E.)

**Step 5.** For an active load with a current mirror, choose the type of current source (BJT or MOSFET) needed to obtain the desired voltage gain and determine its component ratings.

**Step 6.** Determine the voltage, current, and power ratings of active and passive components.

**Step 7.** Analyze and evaluate the complete differential amplifier to check for the desired specifications.

**Step 8.** Use PSpice/SPICE to simulate and verify your design, using the standard values of components with their tolerances.
Summary

Active current sources are commonly employed in integrated circuits to bias transistor circuits at appropriate operating points. There are various types of current sources. The output current of a good source is independent of the transistor parameters and offers a high output resistance. The design of BJT current sources can be simplified by ignoring the base currents of transistors and by assuming a constant voltage drop between the base and the emitter. Diode-connected BJTs and MOSFETs rather than diodes are normally used in current sources to give matching mirror characteristics.

A current source can be either a source or a sink, depending on how it is connected. Current mirrors are often used as an active load to increase the output resistance and the voltage gain of a differential amplifier. Also, cascode-like connections increase the voltage gain.

Emitter-coupled (or source-coupled) amplifiers offer the advantage of direct coupling of cascoded stages. They are usually used as the input stage of differential amplifiers to give a high input resistance and a high CMRR. However, the range of the differential input voltage is very small (typically two to three times $V_T$ for the BJT amplifier). The CMRR depends on the output resistance of the biasing current source. Thus, a current source with a high value of output resistance is highly desirable. Source-coupled differential pairs offer a higher input resistance than emitter-coupled pairs, but they have a low CMRR.

References


Review Questions

1. What are the advantages and disadvantages of a basic current source?
2. What are the advantages and disadvantages of a modified current source?
3. What are the advantages and disadvantages of a Widlar current source?
4. What are the advantages and disadvantages of a cascade current source?
5. What are the advantages and disadvantages of a Wilson current source?
6. What are the differences between Widlar and Wilson current sources?
7. What is a current mirror load?
8. What is the common-mode rejection ratio (CMRR)?
9. What are the advantages of an emitter-coupled pair?
10. What is the DC characteristic of an emitter-coupled pair?
11. What design criteria will yield a large value of CMRR in an emitter-coupled pair?
12. What are the advantages of a source-coupled pair?
13. What is the DC characteristic of a source-coupled pair?
14. What design criteria will yield a large value of CMRR in a source-coupled pair?
15. What is the purpose of cascodelike connections of transistors?

**Problems**

The symbol indicates that a problem is a design problem. The symbol indicates that you can check the solution to a problem using PSpice/SPICE.

Assume that the device parameters are as follows: diodes, $I_S = 10^{-13}$ A and $I_D(\text{min}) = 1$ mA to ensure conduction; transistors, $\beta_F = h_{fe} = 50$, $V_{BE} = 0.7$ V, $I_S = 10^{-14}$ A, and $V_{CE(sat)} = 0.2$ V.

### 9.3 MOSFET Current Sources

9.1 The parameters of the MOSFET current source in Fig. 9.4 are $V_t = 1$ V, $I_O = 20$ mA, $I_R = 20$ mA, $V_{DD} = 15$ V, and $V_M = 40$ V. The channel lengths are $L_1 = L_2 = 10$ $\mu$m and $L_3 = 100$ $\mu$m, and $K_x = 20$ $\mu$A/V$^2$.

Calculate the required values of (a) $K_{n1}$, $W_1$, (b) $K_{n2}$, $W_2$, (c) $K_{n3}$, $W_3$, and (d) the output resistance $R_o$ of the current source. Assume $V_{GS1} = 1.5$ V and $V_{DS1} = 5$ V.

9.2 a. Design the cascode current source in Fig. 9.6(a) to give $I_O = 10$ $\mu$A. Assume $V_{DD} = 10$ V. All MOSFETs are identical and have $L = 20$ $\mu$m, $W = 60$ $\mu$m, $V_t = 1$ V, $K_x = 20$ $\mu$A/V$^2$, and $V_M = 40$ V.

b. Calculate the output resistance $R_o$ and Thevenin’s equivalent voltage $V_{Th}$.

9.3 a. Design the modified Wilson current source in Fig. 9.7(c) to give $I_O = 10$ $\mu$A. Assume $V_{DD} = 10$ V and $V_M = 40$ V. All MOSFETs are identical and have $L = 10$ $\mu$m, $W = 40$ $\mu$m, $K_x = 20$ $\mu$A/V$^2$, and $V_M = 40$ V.

b. Calculate the output resistance $R_o$ and Thevenin’s equivalent voltage $V_{Th}$.

9.4 Design a MOSFET current source so that $R_o \geq 50$ k$\Omega$ at an output current of $I_O = 1$ mA.

9.5 Design a MOSFET current source so that $R_o \geq 500$ k$\Omega$ at an output current of $I_O = 0.1$ mA.

9.6 The Widlar current source shown in Fig. P9.6 has $I_{ref} = 50$ $\mu$A, $R = 2$ k$\Omega$, and $V_{DD} = 12$ V. The MOS parameters are $K_n = 100$ $\mu$A/V$^2$, $V_t = 1$ V, $|V_M| = 100$ V, and $(W/L)_1 = (W/L)_2 = 20$. Determine (a) the output current $I_o$, (b) the output resistance $r_{o2}$, and (c) the value of $R_{ref}$.
9.7 Design the modified Wilson current source in Fig. P9.7 to give $I_O = 10 \mu A$ by determining the resistance $R_{ref}$. Assume $V_{DD} = 12 \, V$. All MOSFETs are identical and have $L = 10 \, \mu m$, $W = 40 \, \mu m$, $K_s = 50 \, \mu A/V^2$, and $V_M = 100 \, V$. Calculate the output resistance $R_o$ and Thevenin’s equivalent voltage $V_{Th}$.

**FIGURE P9.7**

![Diagram of the current source](image)

9.8 The current source shown in Fig. P9.8 has $I_{ref} = 150 \, \mu A$ and $V_{DD} = 12 \, V$. The NMOS parameters are $K_n = 200 \, \mu A/V^2$, $V_t = 1 \, V$, $V_M = -100 \, V$, and $(W/L)_1 = (W/L)_2 = (W/L)_3 = 20$. Determine (a) the gate–source voltages ($V_{GS1}$, $V_{GS2}$, $V_{GS3}$), (b) the drain–source voltages ($V_{DS1}$, $V_{DS2}$, $V_{DS3}$) if the devices are to operate in the active region, and (c) the value of $R_{ref}$.

**FIGURE P9.8**

![Diagram of the current source](image)

9.9 Repeat Prob. 9.8 if the current source in Fig. P9.8 has only two MOSFETs.
9.4 MOS Differential Amplifiers

9.10 The parameters of the MOS differential pair in Fig. 9.8 are $R_{SS} = 150 \, \text{k}\Omega$, $I_Q = 1 \, \text{mA}$, $V_{DD} = 15 \, \text{V}$, and $R_D = 5 \, \text{k}\Omega$. The MOSFETs are identical and have $K_n = 1.5 \, \text{mA/V}^2$, $W/L = 1$, and $V_t = 1.2 \, \text{V}$.

a. Calculate the DC drain currents through the MOSFETs if $v_{gd} = 10 \, \text{mV}$.

b. Assuming $I_{D1} = I_{D2}$, calculate $A_d$, $A_c$, and CMRR; $R_{id}$ and $R_{ic}$; the small-signal output voltage if $v_{g1} = 5 \, \text{mV}$ and $v_{g2} = 10 \, \text{mV}$; and the drain voltage $V_D$.

9.11 Repeat Prob. 9.10 if the active biasing current source is replaced by resistance $R_{SS} = 150 \, \text{k}\Omega$; that is, $I_Q = 0$.

9.12 Design a MOS differential pair as shown in Fig. 9.13 in which one input terminal is grounded. The output is taken from the drain of transistor $M_1$. The DC biasing current is $I_Q = 1 \, \text{mA}$, and $V_{DD} = V_{SS} = 15 \, \text{V}$. The MOSFETs are identical and have $K_n = 1.5 \, \text{mA/V}^2$ and $V_t = 1.2 \, \text{V}$.

a. For a small-signal voltage gain of $A_1 = 20 \, \text{V/V}$.

b. Calculate the design values of $A_d$, $A_c$, and CMRR.

9.13 Design the CMOS amplifier shown in Fig. 9.16 by determining the $W/L$ ratios of the MOSFETs and the threshold voltage $V_t$. The differential voltage gain should be $A_d = 50 \, \text{V/V}$ at biasing current $I_Q = 1 \, \text{mA}$. Assume identical transistors whose channel modulation voltage is $V_M = 40 \, \text{V}$, channel constant is $K_x = 10 \, \mu\text{A/V}^2$, and channel length is $L = 10 \, \mu\text{m}$. The $W/L$ ratio of the current source is unity, and $V_{DD} = V_{SS} = 10 \, \text{V}$.

9.14 A CMOS amplifier is shown in Fig. P9.14. The parameters for the NMOS are $V_t = +2 \, \text{V}$, $V_M = -40 \, \text{V}$, and $V_{GS} = +4 \, \text{V}$ at $I_D = 1 \, \text{mA}$; the parameters for the PMOS are $V_t = -3 \, \text{V}$, $V_M = 40 \, \text{V}$, and $V_{GS} = -6 \, \text{V}$ at $I_D = 1 \, \text{mA}$. Calculate (a) $A_d$, $A_c$, and CMRR and (b) $R_{id}$ and $R_{ic}$.

FIGURE P9.14

9.15 The DC biasing current of the cascoded MOS amplifier shown in Fig. 9.19(a) is kept constant at $I_Q = 50 \, \mu\text{A}$ and $V_{DD} = V_{SS} = 5 \, \text{V}$. All MOS transistors are identical and have $V_M = 50 \, \text{V}$, $K_n = 50 \, \mu\text{A/V}^2$, $W = 50 \, \mu\text{m}$, and $L = 10 \, \mu\text{m}$.

a. Determine the differential voltage gain $A_d$ for single-ended output at the drain terminal of $M_4$.

b. Use PSpice to verify the result.
9.16 The MOS parameters of the CMOS amplifier shown in Figure P9.16 are $K_n = 100 \mu A/V^2$, $V_{GN} = 1 V$, $V_{IP} = 1.5 V$, $|V_M| = 100 V$, $L = 10 \mu m$, $(W/L)_1 = 60$, $(W/L)_7 = (W/L)_8 = 120$, and $(W/L)_2 = (W/L)_3 = (W/L)_4 = (W/L)_5 = (W/L)_6 = 30$. The DC supply voltages are $V_{DD} = V_{SS} = 5 V$, the reference current is to be set at $I_{ref} = 100 \mu A$, and $C_L \approx \infty$.

a. Determine the value of $R_{ref}$ and $V_{GS6}$.
b. Calculate $A_d$, $A_c$, and CMRR.
c. Determine the value of capacitor $C_x$ to limit the upper frequency at $f_H = 500 kHz$.

FIGURE P9.16

9.17 Design a CMOS amplifier as shown in Fig. P9.16 by determining the $W/L$ ratios of the MOSFETs and the threshold voltage $V_t$. The small-signal voltage gain should be $A_d = 250 V/V$ at biasing current $I_{ref} = 100 \mu A$. Assume identical transistors whose channel modulation voltage is $V_M = 70 V$, channel constant is $K_x = 10 \mu A/V^2$, and channel length $L = 10 \mu m$. Also, the width $(W/L)_6 = 1$ and $V_{DD} = V_{SS} = 5 V$.

9.5 Depletion MOS Differential Amplifiers

9.18 The parameters of the depletion MOS differential pair in Fig. 9.22 are $R_{SS} = 50 k \Omega$, $I_Q = 1 mA$, $V_{DD} = V_{SS} = 30 V$, and $R_D = 2 k \Omega$. The MOSFETs are identical and have $V_p = -4 V$ and $I_{DSS} = 20 mA$.

a. Calculate the DC drain currents through the MOSFETs if $v_{id} = 30 mV$.
b. Assuming $I_{D1} = I_{D2}$, calculate $A_d$, $A_c$, and CMRR; $R_{id}$ and $R_{ic}$; and the small-signal output voltage if $v_{g1} = 50 mV$ and $v_{g2} = 20 mV$.

9.19 Repeat Prob. 9.18 if the transistor current source is replaced by resistance $R_{SS} = 100 k \Omega$; that is, $I_{SS} = 0$.

9.20 a. Design a depletion MOS differential pair as shown in Fig. 9.13 in which one input terminal is grounded. The output is taken from the drain of transistor M2. The DC biasing current is $I_Q = 5 mA$, and $V_{DD} = V_{SS} = 15 V$. The MOSFETs are identical and have $V_p = -4 V$ and $I_{DSS} = 20 mA$. A small-signal voltage gain of $|A_2| = 20 V/V$ is required.
b. Calculate the design values of $A_d$, $A_c$, and CMRR.
A depletion MOS amplifier is shown in Fig. P9.21. The MOS parameters are $V_p = 4 \, \text{V}$, $I_{DSS} = -400 \, \mu\text{A}$, $V_M = 40 \, \text{V}$, $I_Q = 200 \, \mu\text{A}$, and $V_{DD} = V_{SS} = 15 \, \text{V}$.

a. Calculate the DC drain currents through the MOSFETs if $v_{id} = 10 \, \text{mV}$.

b. Assuming $I_{D1} = I_{D2}$, calculate $A_d$, $A_c$, and CMRR; $R_{id}$ and $R_{ic}$; and the small-signal output voltage if $v_{g1} = 20 \, \text{mV}$ and $v_{g2} = 10 \, \text{mV}$.

### FIGURE P9.21

![Depletion MOS Amplifier Diagram](image)

### 9.6 BJT Current Sources

#### 9.22 The parameters of the basic current source in Fig. 9.25(a) are $\beta_F = 150$, $R_1 = 20 \, \text{k}\Omega$, $V_{CC} = 15 \, \text{V}$, $V_{BE1} = V_{BE2} = 0.7 \, \text{V}$, and $V_A = 100 \, \text{V}$. Calculate (a) the output current $I_O = I_{C2}$, (b) the output resistance $R_o$, (c) Thevenin’s equivalent voltage $V_{Th}$, and (d) the collector current ratio $I_{C2}/I_{C1}$ if $V_{CE2} = 30 \, \text{V}$.

#### 9.23 a. Design the basic current source in Fig. 9.25(a) to give an output current of $I_O = 200 \, \mu\text{A}$. The transistor parameters are $\beta_F = 100$, $V_{CC} = 30 \, \text{V}$, $V_{BE1} = V_{BE2} = V_{CE1} = 0.7 \, \text{V}$, and $V_A = 150 \, \text{V}$.

b. Calculate the output resistance $R_o$, Thevenin’s equivalent voltage $V_{Th}$, and the collector current ratio if $V_{CE2} = 30 \, \text{V}$.

#### 9.24 The parameters of the modified current source in Fig. 9.27(a) are $\beta_F = 150$, $R_1 = 10 \, \text{k}\Omega$, $V_{CC} = 15 \, \text{V}$, $V_{BE1} = V_{BE2} = V_{BE3} = 0.7 \, \text{V}$, and $V_A = 100 \, \text{V}$. Calculate (a) the output current $I_O = I_{C2}$, (b) the output resistance $R_o$, (c) Thevenin’s equivalent voltage $V_{Th}$, and (d) the collector current ratio $I_{C2}/I_{C1}$ if $V_{CE2} = 30 \, \text{V}$.

#### 9.25 a. Design the modified basic current source in Fig. 9.27(a) to give an output current of $I_O = 50 \, \mu\text{A}$. The transistor parameters are $\beta_F = 150$, $V_{CC} = 30 \, \text{V}$, $V_{BE1} = V_{BE2} = V_{BE3} = 0.7 \, \text{V}$, and $V_A = 100 \, \text{V}$.

b. Calculate the output resistance $R_o$, Thevenin’s equivalent voltage $V_{Th}$, and the collector current ratio if $V_{CE2} = 20 \, \text{V}$.

#### 9.26 The multiple transistors of the current source in Fig. P9.26 have $\beta_F = 150$, $R_1 = 10 \, \text{k}\Omega$, $V_{CC} = 15 \, \text{V}$, and $V_A = 100 \, \text{V}$. The B-E voltages are equal, $V_{BE} = 0.7 \, \text{V}$. Calculate (a) the output current $I_O$, (b) the output resistance $R_o$, (c) Thevenin’s equivalent voltage $V_{Th}$, and (d) the collector current ratio if $V_{CE2} = 15 \, \text{V}$.
9.27 a. Design the Widlar current source in Fig. 9.28(a) to give $I_O = 10 \mu A$ and $I_R = 2 \text{ mA}$. The parameters are $V_{CC} = 30 \text{ V}$, $V_{BE1} = 0.7 \text{ V}$, $V_T = 26 \text{ mV}$, $V_A = 150 \text{ V}$, and $\beta_F = 100$.

b. Calculate the output resistance $R_o$ and Thevenin’s equivalent voltage $V_{Th}$.

9.28 Design a Widlar current source as shown in Fig. 9.28(a) to produce a 10-μA output current. Assume $\beta_F = 100$, $V_{CC} = 30 \text{ V}$, and $R_1 = 30 \text{ kΩ}$. Calculate the output resistance $R_o$.

9.29 Determine the output current $I_O$ and the output resistance $R_o$ of the current source circuit in Fig. P9.29. Assume $V_{CC} = 30 \text{ V}$, $R_1 = 20 \text{ kΩ}$, $R_2 = 10 \text{ kΩ}$, $V_{BE} = 0.7 \text{ V}$, $V_A = 150 \text{ V}$, and $\beta_F = 100$.

9.30 a. Design the Wilson current source in Fig. 9.30(a) to give $I_O = 10 \mu A$. The parameters are $V_{CC} = 30 \text{ V}$, $V_{BE} = 0.7 \text{ V}$, $V_T = 26 \text{ mV}$, $V_A = 100 \text{ V}$, and $\beta_F = 150$.

b. Calculate the output resistance $R_o$ and Thevenin’s equivalent voltage $V_{Th}$.

9.31 For the Wilson current source in Fig. P9.31, determine the output current $I_O$ and the output resistance $R_o$. Assume $V_{CC} = 20 \text{ V}$, $V_{BE} = 0.7 \text{ V}$, $V_T = 26 \text{ mV}$, $V_A = 150 \text{ V}$, and $\beta_F = 150$. 
9.32 Repeat Prob. 9.31 for $V_{CC} = 30$ V.

9.33 For the current source in Fig. P9.33, determine the output resistance $R_o$ and Thevenin’s equivalent voltage $V_{Th}$. Assume $V_{CC} = 30$ V, $V_{BE} = 0.7$ V, $V_T = 26$ mV, $V_A = 150$ V, and $\beta_F = 150$.

**FIGURE P9.33**

---

9.34 Determine the sensitivity $S$ of output current $I_O$ to supply voltage $V_{CC}$ for the circuit in Fig. P9.34. $S$ is defined as

\[ S = \frac{V_{CC}}{\delta I_O/\delta V_{CC}} \]

**FIGURE P9.34**

---

9.35 Design a BJT current source so that $R_o \geq 50$ kΩ at an output current of $I_O = 1$ mA.

D

9.36 Design a BJT current source so that $R_o \geq 500$ kΩ at an output current of $I_O = 1$ mA.

D

9.7 BJT Differential Amplifiers

9.37 The parameters of the emitter-coupled pair in Fig. 9.33 are $\beta_F = 150$, $R_{EE} = 20$ kΩ, $I_Q = 0.25$ mA, $V_{CC} = 12$ V, and $R_C = 10$ kΩ.

a. Calculate the DC collector currents through the transistors if $v_{id} = 10$ mV.

b. Assuming $I_{C1} = I_{C2}$, calculate $A_d$, $A_c$, and CMRR; $R_{id}$ and $R_{ic}$; and the small-signal output voltage if $v_{B1} = 30$ mV and $v_{B2} = 20$ mV. Assume $V_T = 26$ mV.

9.38 a. Design an emitter-coupled pair as shown in Fig. 9.41, in which one input terminal is grounded. The output is taken from the collector of transistor $Q_2$. The biasing current is $I_O = I_{EE} = 10$ mA, and $V_{CC} = -V_{EE} = 12$ V. The transistors are identical. Assume $V_{BE} = 0.7$ V, $V_T = 26$ mV, $\beta_F = 100$, and $V_A = 40$ V. A small-signal voltage gain of $A_1 = -150$ V/V is required.

b. Calculate the design values of $A_d$, $A_c$, and CMRR.
9.39  A differential amplifier is shown in Fig. P9.39. The transistors are identical. Assume $V_{BE} = 0.7\, V$, $V_T = 26\, mV$, $\beta_F = 50$, and $V_A = 40\, V$. Calculate the values of $A_d$, $R_{id}$, $A_c$, $R_{ic}$, and CMRR.

**FIGURE P9.39**

9.40  The parameters of the differential amplifier in Fig. 9.44 are $\beta_{F(npn)} = 100$, $\beta_{F(pnp)} = 50$, $V_A = 40\, V$, $I_Q = 10\, \mu A$, and $V_{CC} = 10\, V$. Calculate $A_d$, $R_{id}$, $R_o$, and the overall voltage gain with load $A_{d(load)}$. Assume $V_T = 26\, mV$.

9.41  The parameters of the differential amplifier in Fig. 9.45 are $\beta_{F(npn)} = 100$, $\beta_{F(pnp)} = 50$, $V_A = 40\, V$, $I_Q = 5\, \mu A$, and $V_{CC} = 10\, V$. Calculate $A_d$, $R_{id}$, $R_o$, and the overall voltage gain with load $A_{d(load)}$. Assume $V_T = 26\, mV$.

9.42  A differential amplifier is shown in Fig. P9.42. The transistors are identical. Assume $V_{BE} = 0.7\, V$, $V_T = 26\, mV$, $\beta_{F(npn)} = 100$, $\beta_{F(pnp)} = 50$, $V_A = 40\, V$, and $V_{CC} = 10\, V$. Calculate the values of $R_1$, $R_2$, $A_d$, $R_{id}$, $A_c$, $R_{ic}$, and CMRR.

**FIGURE P9.42**
9.43 The parameters of the BJT amplifier shown in Fig. P9.43 are $\beta_{FE} = 100$, $\beta_{FP} = 120$, $|V_A| = 100$ V, and $V_{BEN} = -V_{BEP} = 0.65$ V. The DC supply voltages are $V_{CC} = V_{EE} = 5$ V, the reference current is to be set at $I_{ref} = 100$ $\mu$A, and $R_L = 50$ k$\Omega$.

a. Determine the value of $R_{ref}$.
b. Calculate $A_d$, $A_c$, and CMRR.
c. Determine the value of capacitor $C_x$ to limit the upper frequency at $f_H = 500$ kHz.

FIGURE P9.43

9.44 Design a differential amplifier as shown in Fig. 9.44 to satisfy the following conditions: a differential voltage gain of $A_o \geq 1500$ V/V, CMRR of $\geq 3500$, and load of $R_L = 50$ k$\Omega$, with a coupling capacitor of $C_L = \infty$. Use all BJTs and the cascoded current source, similar to the one in Fig. 9.29. Use PSpice to verify your design with a $v_{id} = 5$ $\mu$V and $v_{ic} = 10$ mV. Supply voltages are $V_{CC} = -V_{EE} = 15$ V.

9.8 BiCMOS Differential Amplifiers

9.45 A BiCMOS amplifier is shown in Fig. P9.45. The depletion PMOS parameters are $V_p = 4$ V, $I_{DSS} = 400$ $\mu$A, and $V_M = 100$ V. The BJT parameters are $\beta_{F(npn)} = 100$, $\beta_{F(pnp)} = 50$, and $V_A = 40$ V. Assume $V_{DD} = V_{EE} = 15$ V and $I_Q = 200$ $\mu$A. Calculate $A_d$, $A_c$, and CMRR.
9.46 The DC biasing current of a BiCMOS amplifier is kept constant at $I_Q = 5 \mu A$. All bipolar transistors are identical, with $\beta_{F(\text{npn})} = 100$, $\beta_{F(\text{pnp})} = 50$, and $V_A = 40 \text{ V}$. Also, the MOS transistors are identical, with $|V_M| = 20 \text{ V}$. For the NMOS, $V_i = +2 \text{ V}$ and $V_{GS} = +4 \text{ V}$ at $I_D = 1 \text{ mA}$; for the PMOS, $V_i = -3 \text{ V}$ and $V_{GS} = -6 \text{ V}$ at $I_D = 1 \text{ mA}$. Determine the differential voltage gain $A_d$ for single-ended output (a) for the basic BiCMOS amplifier in Fig. 9.49(c), (b) for the cascode BiCMOS amplifier in Fig. 9.50, and (c) for the double-cascode BiCMOS amplifier in Fig. 9.51.

9.47 A BiCMOS amplifier is shown in Fig. P9.47. The PMOS parameters are $V_t = 3 \text{ V}$ and $V_{GS} = 6 \text{ V}$ at $I_D = 1 \text{ mA}$. The BJT parameters are $\beta_{F(\text{npn})} = 100$, $\beta_{F(\text{pnp})} = 50$, and $V_A = 40 \text{ V}$. Assume $V_{DD} = V_{EE} = 15 \text{ V}$ and $I_Q = 200 \mu A$. Calculate $A_d$, $A_c$, and CMRR.

9.48 Design a differential amplifier as shown in Fig. 9.51 to satisfy the following conditions: a differential voltage gain of $A_d \geq 1500 \text{ V/V}$, CMRR of $\geq 3500$, and load of $R_L = 50 \text{ k} \Omega$, with a coupling capacitor of $C_L = \infty$. Use all BJTs and NMOSs. Use PSpice to verify your design with $v_d = 5 \mu V$ and $v_c = 10 \text{ mV}$. Supply voltages are $V_{CC} = V_{EE} = 12 \text{ V}$.

9.9 Frequency Response of Differential Amplifiers

9.49 The emitter-coupled pair in Fig. 9.33 has $R_{EE} = 20 \text{ k} \Omega$, $I_Q = 5 \text{ mA}$, $V_{CC} = 12 \text{ V}$, and $R_C = 10 \text{ k} \Omega$. The small-signal transistor parameters are $C_\pi = 5 \text{ pF}$, $C_\mu = 2 \text{ pF}$, $\beta_{F(\text{npn})} = 100$, and $\beta_{F(\text{pnp})} = 50$. Find the frequency-dependent gains $A_d(j\omega)$ and $A_c(j\omega)$.

9.50 The emitter-coupled pair in Fig. P9.50 has $I_Q = 5 \text{ mA}$, $V_{CC} = -V_{EE} = 15 \text{ V}$, and $R_C = 10 \text{ k} \Omega$. The small-signal transistor parameters are $C_\pi = 5 \text{ pF}$, $C_\mu = 2 \text{ pF}$, $\beta_{F(\text{npn})} = 100$, and $\beta_{F(\text{pnp})} = 50$. Find the frequency-dependent gains $A_d(j\omega)$ and $A_c(j\omega)$.
9.51 The differential amplifier in Fig. 9.42 has $V_A = 40 \text{ V}$, $I_Q = 10 \mu\text{A}$ at $R_{EE} = 50 \text{k}\Omega$, and $V_{CC} = -V_{EE} = 15 \text{ V}$. The small-signal transistor parameters are $C_m = 5 \text{ pF}$, $C_{\mu} = 2 \text{ pF}$, $\beta_{F(npn)} = 100$, and $\beta_{F(pnp)} = 50$. Find the frequency-dependent gains $A_d(j\omega)$ and $A_c(j\omega)$.

9.52 The source-coupled pair in Fig. 9.8 has $R_{SS} = 50 \text{k}\Omega$, $I_Q = 10 \text{ mA}$, $V_{DD} = 30 \text{ V}$, and $R_D = 5 \text{k}\Omega$. The MOSFETs are identical and have $K_n = 1.5 \text{ mA/\mu m}^2$, $W/L = 1$, $|V_M| = 100 \text{ V}$, and $V_t = 1.2 \text{ V}$. The small-signal MOS parameters are $C_{gs} = 5 \text{ pF}$ and $C_{gd} = 2 \text{ pF}$. Find the frequency-dependent gains $A_d(j\omega)$ and $A_c(j\omega)$.

9.53 The MOS amplifier shown in Fig. 9.14 has $R_{SS} = 50 \text{k}\Omega$, $I_Q = 1 \text{ mA}$, and $V_{DD} = 15 \text{ V}$. The MOSFETs are identical and have $K_n = 1.5 \text{ mA/\mu m}^2$, $W/L = 1$, $|V_M| = 100 \text{ V}$, and $V_t = 1.2 \text{ V}$. The small-signal MOS parameters are $C_{gs} = 5 \text{ pF}$ and $C_{gd} = 2 \text{ pF}$. Find the frequency-dependent gains $A_d(j\omega)$ and $A_c(j\omega)$.

9.54 Design a differential amplifier configuration as shown in Fig. P9.54 to obtain a small-signal differential gain, with $A_d = 250 \text{ V/V}$, CMRR $\approx 10^3$, and $f_b \approx 250 \text{ kHz}$. Use any suitable transistors, BJTs, or MOSFETs. Assume $V_{CC} = V_{EE} = 12 \text{ V}$. The configuration should be the one that is least expensive. Use PSpice to verify your design and give the component estimates.
CHAPTER 10

FEEDBACK AMPLIFIERS

Learning Outcomes

After completing this chapter, students should be able to do the following:

• Describe the types and the properties of feedback amplifiers.
• Describe the different feedback configurations, their properties, and circuit implementations.
• Select the feedback configuration and analyze feedback amplifiers to meet specific requirements.
• Design a feedback network to meet desired closed-loop gain, the input impedance, and the output impedance.
• Determine the stability conditions of feedback amplifiers.
• Apply the compensation techniques to stabilize an unstable amplifier.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A, A_f$</td>
<td>Open-loop and closed-loop gain of an amplifier</td>
</tr>
<tr>
<td>$A_0, A_{of}$</td>
<td>Low-frequency open-loop and closed-loop gain of an amplifier</td>
</tr>
<tr>
<td>$i_i, i_e, i_f$</td>
<td>Input, error, and feedback current signals</td>
</tr>
<tr>
<td>$R_i, R_{if}$</td>
<td>Input resistances of an amplifier without feedback and with feedback</td>
</tr>
</tbody>
</table>
10.1 Introduction

Feedback is commonly used in amplifier circuits. A signal that is proportional to the output is compared with an input or a reference signal so that a desired output is obtained from the amplifier. The difference between the input and the feedback signals, called the error signal, is amplified by the amplifier. There are two types of feedback:

- In negative feedback, the output signal (or a fraction of it) is continuously fed back to the input side and is subtracted from the input signal to create an error signal, which is then corrected by the amplifier to produce the desired output signal.
- In positive feedback, the output signal (or a fraction of it) is continuously fed back to the input side and added to the input signal to create a larger error signal, which is then amplified to produce a larger output until the output reaches the saturation voltage limit of the amplifier.

In negative feedback, the signal that is fed back to the input side is known as the feedback signal, and its polarity is opposite that of the input signal (i.e., it is out of phase by 180° with respect to the input signal). Negative feedback in an amplifier has four major benefits: (1) It stabilizes the overall gain of the amplifier with respect to parameter variations due to temperature, supply voltage, and so on; (2) it increases or decreases the input and output impedances; (3) it reduces the distortion and the effect of nonlinearity; and (4) it increases the bandwidth. There are two disadvantages of negative feedback: (1) The overall gain is reduced almost in direct proportion to the benefits, and it is often necessary to compensate for the decrease in gain by adding an extra amplifier stage; (2) the circuit may tend to oscillate, in which case careful design is required to overcome this problem. Negative feedback is also known as degenerative feedback because it degenerates (or reduces) the output signal.

The op-amp circuits in Chapter 2 use negative feedback. The amplifier gain $A_f$ is almost independent of the op-amp gain $A$; it depends on the external circuit elements only. For example, the gain of the inverting amplifier in Fig. 3.11 is $-R_F/R_1$, which is independent of the op-amp gain $A$, and the input impedance is approximately $R_1$. The gain of the noninverting op-amp amplifier in Fig. 3.9 is $(1 + R_F/R_1)$, and its input impedance is very large. The output impedance of both amplifiers is very small.

In positive feedback, the feedback signal is in phase with the input signal. Thus, the error signal is the algebraic sum of the input and feedback signals, and it is amplified by the amplifier. Thus, the output may continue to increase, resulting in an unstable situation, and the circuit may oscillate between the limits of the power supplies at the resonant frequency of the amplifier. Positive feedback is often referred to as regenerative feedback because it increases the output signal. Positive feedback is generally applied in oscillator circuits, which we will study in Chapter 13. Note that positive feedback does not necessarily imply oscillations. In fact, positive feedback is quite useful in some applications, such as active filters.
10.2 Feedback

Consider the noninverting op-amp amplifier shown in Fig. 10.1(a). The voltages \( v_s, v_f, \) and \( v_e \) are related as follows:

\[
\begin{align*}
    v_e &= v_s - v_f \\
    v_o &= A v_e \\
    v_f &= \frac{R_1}{R_1 + R_F} v_o
\end{align*}
\]

The input and output relationships described by these equations can be represented by a block diagram as shown in Fig. 10.1(b). The voltage \( v_e \), which is the difference between \( v_s \) and \( v_f \), is amplified by the voltage gain \( A \). The feedback signal \( v_f \) is proportional to the output voltage and is fed back to the input side. Thus, the amplifier feeds the output voltage back to the input side and compares the voltages. There are two circuits: the amplifier circuit and the feedback circuit (or network) consisting of \( R_1 \) and \( R_F \). The voltages \( v_s, v_f, \) and \( v_e \) form a series circuit at the input side as shown in Fig. 10.1(a), whereas \( v_o \) is applied directly to the feedback network. That is, the noninverting amplifier uses series-shunt, or voltage-sensing/voltage-comparing, feedback.

Now consider the inverting op-amp amplifier shown in Fig. 10.2(a). There is a common node at the input side where the input current \( i_i \) and the feedback current \( i_f \) meet. The voltages and currents are related as follows:

\[
\begin{align*}
    i_e &= i_i - i_f \\
    v_e &= -R_i i_e \\
    v_o &= A v_e \\
    i_f &= \frac{-v_e - v_o}{R_F} \approx \frac{-v_o}{R_F}
\end{align*}
\]

since \( v_e \approx 0 \) and \( v_o >> v_e \). The input and output relationships described by these equations are shown by the block diagram in Fig. 10.2(b). The current \( i_e \), which is the difference between \( i_i \) and \( i_f \), is amplified by the transimpedance gain \(-R_i A\). The feedback current signal \( i_f \) is proportional to the output voltage.

![Feedback representation of noninverting op-amp amplifier](image)

**FIGURE 10.1** Feedback representation of noninverting op-amp amplifier
Thus, the amplifier feeds the output voltage back to the input side and compares the currents. Here, the feedback network consists of $R_F$. The currents $i_i$, $i_f$, and $i_e$ form a shunt connection at the input side, whereas $v_o$ is applied directly to the feedback network. Thus, the inverting amplifier uses shunt-shunt, or voltage-sensing/current-comparing, feedback.

## 10.3 Characteristics of Feedback

In the inverting and noninverting amplifiers in Figs. 10.1 and 10.2, the output voltage is fed back directly to the input side and is compared to either the voltage or the current at the input side. Feedback can be represented by the general configuration shown in Fig. 10.3, where $\beta$ is called the feedback ratio (or factor) and $A$ is the amplifier gain. The units of $A$ could be $\text{V/V}$, $\text{A/V}$, $\text{A/A}$, or $\text{V/A}$, and the units of $\beta$ will be the reciprocal of those of $A$. For the noninverting amplifier, $A$ is in $\text{V/V}$ and $\beta$ is in $\text{V/V}$; for the inverting amplifier, $A$ is in $\text{V/A}$ and $\beta$ is in $\text{A/V}$.

**NOTE** $\beta$ is the feedback ratio, whereas $\beta_F (=\beta_T)$ is the forward current gain of a bipolar transistor. To distinguish the feedback factor $\beta$ from the BJT transistor current gain $\beta_T$, in this chapter we use the hybrid parameter $h_{fe}$ for the BJT current gain.

### 10.3.1 Closed-Loop Gain

From Fig. 10.3, we can determine the closed-loop gain and the conditions for obtaining negligible error. The various signals (either voltages or currents) in Fig. 10.3 are related by the following equations:

\[
S_o = AS_e \\
S_e = S_i - S_f \\
S_f = \beta S_o
\]

### FIGURE 10.3 General feedback configuration

Thus, the amplifier feeds the output voltage back to the input side and compares the currents. Here, the feedback network consists of $R_F$. The currents $i_i$, $i_f$, and $i_e$ form a shunt connection at the input side, whereas $v_o$ is applied directly to the feedback network. Thus, the inverting amplifier uses shunt-shunt, or voltage-sensing/current-comparing, feedback.
where \( A \) = open-loop gain of the amplifier
\( S_o \) = output signal
\( S_e \) = error signal
\( S_f \) = feedback signal
\( S_i \) = input signal
\( \beta \) = feedback factor

Substituting \( S_e \) from Eq. (10.2) into Eq. (10.1) gives

\[
S_o = AS_e = AS_i - AS_f \tag{10.4}
\]

Substituting \( S_f \) from Eq. (10.3) into Eq. (10.4) yields

\[
S_o = AS_i - \beta AS_o
\]

which gives the overall gain \( A_f \) with negative feedback as

\[
A_f = \frac{S_o}{S_i} = \frac{A}{1 + \beta A} \tag{10.5}
\]

\( A_f \) is often known as the *closed-loop gain*. Equation (10.5) is derived for negative feedback. \( \beta A \) is the gain around the feedback loop, known as the *loop gain* or the *loop transmission*. Let us define \( T_L = \beta A \).

If \( T_L \gg 1 \), Eq. (10.5) becomes

\[
A_f \approx \frac{1}{\beta} \tag{10.6}
\]

That is, for large values of loop gain \( T_L \), the closed-loop gain \( A_f \) is independent of the open-loop gain \( A \) and depends on the feedback factor \( \beta \) only.

Substituting \( S_f \) from Eq. (10.3) and \( S_o \) from Eq. (10.5) into Eq. (10.2) gives the error signal \( S_e \):

\[
S_e = S_i - S_f = S_i - \beta S_o = S_i - \frac{\beta AS_i}{1 + \beta A} = \frac{S_i}{1 + \beta A} = \frac{S_i}{1 + T_L} \tag{10.7}
\]

As \( T_L \) becomes much greater than 1, \( S_e \) becomes much smaller than \( S_i \), and \( S_i \approx S_f \). Substituting \( S_o \) from Eq. (10.5) into Eq. (10.3) gives

\[
S_f = \frac{\beta AS_i}{1 + \beta A} S_i = \frac{T_L}{1 + T_L} S_i \tag{10.8}
\]

If \( T_L \gg 1 \), \( S_f \approx S_i = \beta S_o \). That is, the output signal \( S_o \) is the amplified version of the input signal \( S_i \), provided \( \beta < 1 \).

With positive feedback, the sign of \( \beta A \) changes and the closed-loop gain becomes

\[
A_f = \frac{S_o}{S_i} = \frac{A}{1 - \beta A} \tag{10.9}
\]

This confirms that the output will continue to increase until the saturation limit. Therefore, the amplifier can be made to oscillate by making \( \beta A = 1 \).
10.3.2 Gain Sensitivity

In most practical amplifiers, the open-loop gain $A$ is dependent on temperature and the operating conditions of active devices. The effect of variations in the open-loop gain $A$ can be determined from the sensitivity of the closed-loop gain $A_f$. Differentiating $A_f$ in Eq. (10.5) with respect to $A$ gives

$$\frac{dA_f}{dA} = \frac{(1 + \beta A) - \beta A}{(1 + \beta A)^2} = \frac{1}{(1 + \beta A)^2} \tag{10.10}$$

If $A$ changes by $\delta A$, then $A_f$ will change by $\delta A_f$. Thus, Eq. (10.10) yields

$$\frac{\delta A_f}{(1 + \beta A)^2} = \frac{\delta A}{1 + \beta A} \quad \text{(assuming $\delta A_f \approx dA_f$)} \tag{10.11}$$

which gives the approximate value of $\delta A_f$ for finite increments in $\delta A$. The fractional change in $A_f$ is given by

$$\frac{\delta A_f}{A_f} = \frac{1 + \beta A}{A} \left( \frac{\delta A}{A} \right) \tag{10.12}$$

which shows that a fractional change in $A$ of $(\delta A/A)$ causes a fractional change in $A_f$ of $(\delta A_f/A_f)$ such that

$$\frac{\delta A_f}{A_f} = \frac{1}{1 + \beta A} \left( \frac{\delta A}{A} \right) \tag{10.13}$$

If $\delta A/A$ is, say, 10%, the change in $A_f$ is

$$\frac{\delta A_f}{A_f} = \frac{10}{1 + \beta A} \%$$

only. Thus, the sensitivity of the closed-loop gain $A_f$ to the open-loop gain $A$ is defined as

$$S_{A_f}^{A_f} = \frac{\delta A_f}{A_f} = \frac{1}{1 + \beta A} \tag{10.13}$$

For $\beta A \gg 1$, which is generally the case, the sensitivity of $A_f$ to $A$ becomes very small. Thus, a significant change in $A$ will cause only a small change in $A_f$.

10.3.3 Feedback Factor Sensitivity

We can see from Eq. (10.6) that the closed-loop gain $A_f$ depends on the feedback factor $\beta$ only. The effect of variations of the feedback factor $\beta$ on the gain $A_f$ can be determined. Differentiating $A_f$ in Eq. (10.5) with respect to $\beta$ gives

$$\frac{dA_f}{d\beta} = -\frac{A^2}{(1 + \beta A)^2} = -A_f^2 \tag{10.14}$$

If $\beta$ changes by $\delta \beta$, then $A_f$ will change by $\delta A_f$. From Eq. (10.14), we get

$$\delta A_f = -A_f^2 \delta \beta \quad \text{(assuming $\delta A_f \approx dA_f$)} \tag{10.15}$$

Thus, the fractional change in $A_f$ is given by

$$\frac{\delta A_f}{A_f} = -A_f \delta \beta \tag{10.16}$$
The sensitivity of closed-loop gain $A_f$ to the feedback factor $\beta$ is defined as

$$S_{A_f}^\beta = \frac{\delta A_f/A_f}{\delta \beta/\beta} = -A_f\beta = -\frac{\beta A}{1 + \beta A} \quad (10.17)$$

For $\beta A \gg 1$, Eq. (10.17) can be reduced to

$$S_{A_f}^\beta = -1 \quad (10.18)$$

Therefore, the gain $A_f$ is directly sensitive to any change in the feedback factor $\beta$. The negative sign in Eq. (10.18) signifies that an increase in $\beta$ will cause a decrease in $A_f$.

**NOTE** Although the value of $A$ can be positive or negative depending on the circuit configuration, we will use only the absolute value of $A$ in equations for negative feedback such as Eq. (10.5).

### EXAMPLE 10.1

**Finding the effect of changes in the open-loop gain on the closed-loop gain** The open-loop gain of an amplifier is $A = 250$, and the feedback factor is $\beta = 0.8$.

(a) Determine the closed-loop gain $A_f = S_o/S_i$.

(b) If the open-loop gain $A$ changes by $+20\%$, determine the percentage change in the closed-loop gain $A_f$ and its value.

(c) If the feedback factor $\beta$ changes by $-20\%$, determine the percentage change in the closed-loop gain $A_f$ and its value.

**SOLUTION**

$A = 250, \beta = 0.8$, and $T_L = \beta A = 250 \times 0.8 = 200$.

(a) From Eq. (10.5),

$$A_f = \frac{250}{1 + 200} = 1.2438$$

(b) $\delta A/A = 20\%$. From Eq. (10.13),

$$\frac{\delta A_f}{A_f} = \frac{20\%}{1 + 200} = 0.1\%$$

$$\delta A_f = 0.1\% \times 1.2438 = 0.00124$$

$$A_f = 1.2438 + 0.00124 = 1.245$$

(c) $\delta \beta/\beta = 20\%$. From Eq. (10.18),

$$\frac{\delta A_f}{A_f} = -20\%$$

$$\delta A_f = -20\% \times 1.2438 = -0.249$$

$$A_f = 1.2438 - 0.249 = 0.995$$

**NOTE**: The overall gain $A_f$ does not change much with a wide variation in the open-loop gain $A$. But the gain $A_f$ changes directly with the feedback factor $\beta$. In designing a feedback amplifier, special care should be taken to ensure that the variation in the feedback factor is kept to a minimum.
10.3.4 Frequency Response

Negative feedback increases the bandwidth of an amplifier. To prove this, let us consider a simple amplifier whose open-loop gain $A$ is dependent on the frequency, which can be expressed in Laplace’s domain as

$$A(s) = \frac{A_o}{1 + s/(2\pi f_H)} \quad (10.19)$$

where $A_o$ is open-loop low-frequency gain and $f_H$ is open-loop 3-dB break frequency, in Hertz. From Eq. (10.5), the overall gain is given by

$$A_f(s) = \frac{A(s)}{1 + \beta A(s)} \quad (10.20)$$

where the feedback factor $\beta$ is independent of frequency. Substituting $A(s)$ from Eq. (10.19) into Eq. (10.20) yields

$$A_f(s) = \frac{A_o/[1 + s/(2\pi f_H)]}{1 + \beta A_o/[1 + s/(2\pi f_H)]} = \frac{A_o}{1 + \beta A_o} \frac{1}{1 + s/[2\pi f_H(1 + \beta A_o)]} \quad (10.21)$$

which gives the low-frequency closed-loop gain $A_{of}$ as

$$A_{of} = \frac{A_o}{1 + \beta A_o} = \frac{A_o}{1 + T_{Lo}} \quad (10.22)$$

where $T_{Lo} = \beta A_o$ is called the low-frequency loop gain. From Eq. (10.21), the 3-dB break frequency $f_{Hf}$ with feedback becomes

$$f_{Hf} = f_H(1 + \beta A_o) \quad (10.23)$$

Thus, without feedback, the following equations apply:

- Low-frequency gain $= A_o$
- Bandwidth $BW = f_H$
- Gain–bandwidth product $GBW = A_o f_H$ \( (10.24) \)

With feedback, these equations apply:

- Low-frequency gain $A_{of} = \frac{A_o}{1 + \beta A_o}$
- Bandwidth $BW_f = f_H(1 + \beta A_o)$
- Gain–bandwidth product $GBW_f = A_{of} BW_f = A_o f_H \quad (10.25)$

We can conclude from Eqs. (10.22) and (10.23) that feedback reduces the low-frequency gain by a factor of $(1 + \beta A_o)$ but increases the 3-dB frequency by the same amount $(1 + \beta A_o)$. However, the gain–bandwidth product remains constant at $A_o f_H$. Negative feedback allows the designer to trade gain for bandwidth, and it is widely used as a method for designing broadband amplifiers. The gain reduction is generally compensated for by adding more amplifying stages, which may also be feedback amplifiers. The plots of magnitude against frequency for $A_f$ and $A$ are shown in Fig. 10.4.
EXAMPLE 10.2

Finding the effect of feedback on the frequency of an amplifier  The feedback factor of a closed-loop amplifier is $\beta = 0.8$. The open-loop gain is expressed as

$$A(s) = \frac{250}{1 + s/(2\pi \times 100)}$$

Determine (a) the closed-loop low-frequency gain $A_{of}$, (b) the closed-loop bandwidth $B_{wf}$, and (c) the gain–bandwidth product GBW.

SOLUTION

$A_o = 250$, $\beta = 0.8$, and $f_H = 100$.

(a) From Eq. (10.22),

$$A_{of} = \frac{250}{1 + 250 \times 0.8} = 1.24378$$

(b) From Eq. (10.23),

$$B_{wf} = 100 \times (1 + 250 \times 0.8) = 20.1 \text{ kHz}$$

(c) From Eq. (10.24),

Gain–bandwidth product GBW $= A_{of}f_H = 250 \times 100 = 25 \times 10^3$

10.3.5 Distortion

An amplifier contains nonlinear devices such as transistors. As a result, the plot of the output signal $S_o$ against the input signal $S_i$ will not be linear. Thus, if the input signal is a sinusoidal waveform, the output signal will not be sinusoidal; that is, the output signal will be distorted. The effect of distortion in an amplifier is to reduce the gain of the open-loop transfer function. The closed-loop gain $A_f$, however, remains almost independent of the open-loop gain, as shown by Eq. (10.6). Therefore, negative feedback can reduce the effect of slope changes on the open-loop transfer function.
Let us consider an amplifier whose transfer characteristic ($v_o$ versus $v_i$) is nonlinear, as shown in Fig. 10.5. There are four regions of constant gain: $A_1$, $A_2$, $A_3$, and $A_4$. If negative feedback is applied with a feedback factor of $\beta$, Eq. (10.6) can be used as follows to calculate the closed-loop gains corresponding to the four regions:

\[
A_{f1} = \frac{A_1}{1 + \beta A_1} \approx \frac{1}{\beta} \quad \text{for} \quad \beta A_1 >> 1
\]

\[
A_{f2} = \frac{A_2}{1 + \beta A_2} \approx \frac{1}{\beta} \quad \text{for} \quad \beta A_2 >> 1
\]

\[
A_{f3} = \frac{A_3}{1 + \beta A_3} \approx \frac{1}{\beta} \quad \text{for} \quad \beta A_3 >> 1
\]

\[
A_{f4} = \frac{A_4}{1 + \beta A_4} \approx \frac{1}{\beta} \quad \text{for} \quad \beta A_4 >> 1
\]

Therefore, the slopes of the transfer characteristic in the four regions will be almost equal, and the transfer characteristic will be as shown in Fig. 10.6. The transfer characteristic with negative feedback is much less nonlinear than that of the original amplifier without negative feedback.
EXAMPLE 10.3

Finding the effect of amplifier nonlinearity on the closed-loop gain  The transfer characteristic of an amplifier without feedback is approximated by the following values of open-loop gain for given ranges of input voltage $v_i$:

$$A = \begin{cases} 
1000 & \text{for } 0 < v_i \leq 0.5 \text{ mV} \\
500 & \text{for } 0.5 \text{ mV} < v_i \leq 1 \text{ mV} \\
250 & \text{for } 1 \text{ mV} < v_i \leq 2 \text{ mV} \\
0 & \text{for } v_i > 2 \text{ mV}
\end{cases}$$

If the feedback factor is $\beta = 0.5$, determine the closed-loop gains of the transfer characteristic.

SOLUTION

From Eq. (10.5), the closed-loop gains become

$$A_f = \begin{cases} 
\frac{1000}{1 + 0.5 \times 1000} = 1.996 & \text{for } 0 < v_i \leq 0.5 \text{ mV} \\
\frac{500}{1 + 0.5 \times 500} = 1.992 & \text{for } 0.5 \text{ mV} < v_i \leq 1 \text{ mV} \\
\frac{250}{1 + 0.5 \times 250} = 1.984 & \text{for } 1 \text{ mV} < v_i \leq 2 \text{ mV} \\
0 & \text{for } v_i > 2 \text{ mV}
\end{cases}$$

NOTE: As the input voltage $v_i$ varies from 0 to 2 mV, the slope of output voltage versus input voltage (i.e., gain) varies from 1000 to 250 in a nonlinear fashion. But the closed-loop gain $A_f$ remains almost constant.

KEY POINTS OF SECTION 10.3

- Feedback reduces the gain of a feedback amplifier. However, the bandwidth is widened proportionally. Also, feedback reduces the effect of amplifier distortion, nonlinearity, and variations in amplifier parameters.
- For a large value of loop-gain $T_L$, the closed-loop gain $A_f$ is inversely proportional to the feedback factor $\beta$. That is, $A_f$ is sensitive to changes in the feedback network parameters.
10.4 Feedback Topologies

The feedback configuration in Fig. 10.3 represents a general form; it does not indicate whether the input and output signals are voltages or currents. In practical amplifiers, the input and output signals can be either voltages or currents. If the output voltage is the feedback signal, it can be either compared with the input voltage to generate the error voltage signal (as shown in Fig. 10.1) or compared with the input current to generate the error current signal (as shown in Fig. 10.2). Similarly, the output current can be fed back and either compared with the input voltage to generate the error voltage signal or compared with the input current to generate the error current signal. Therefore, there are four feedback configurations, depending on whether the input and output signals are voltages or currents [1].

10.4.1 Feedback Configurations

There are four feedback configurations: series-shunt, series-series, shunt-shunt, and shunt-series. A potential divider, that is, a resistor (or capacitor) in series with another resistor (or capacitor) can sense a voltage, and placing a small resistor in the wire and sensing the voltage across it can sense a current. For a voltage subtraction, apply the input and the feedback signals to two distinct nodes, whereas for current subtraction, apply them to a single node. If the feedback signal is taken from the output terminal—for example, as in Fig. 10.7(c)—then the output side is a shunt feedback; if the feedback signal is taken from the voltage across a resistor—for example, as in Fig. 10.8(c)—then it is a series connection on the output sides.

![Figure 10.7 Series-shunt feedback](image)
In series-shunt (voltage-sensing/voltage-comparing) feedback, shown in Fig. 10.7(a), the output voltage $v_o$ is the input to the feedback network, and the feedback voltage $v_f$ is proportional to the output voltage $v_o$. The feedback network forms a series circuit with the input voltage $v_i$ but a parallel circuit with the output voltage $v_o$. The input current $i_i$ flows through the loop formed by the input voltage, the amplifier, and the feedback network; that is, $v_i - v_f = v_c$. A series-shunt implementation using an op-amp is shown in Fig. 10.7(a). The discrete transistor implementation is shown in Fig. 10.7(c) and the feedback block as shown in Fig. 10.7(d).

In series-series (current-sensing/voltage-comparing) feedback, shown in Fig. 10.8(a), the output current $i_o$ is the input to the feedback network, and the feedback voltage $v_f$ is proportional to the output current $i_o$. The feedback network forms a series circuit with the input voltage and the output current. The input current flows through the loop formed by the input voltage, the amplifier, and the feedback network; that is, $v_i - v_f = v_c$. A series-series implementation using an op-amp is shown in Fig. 10.8(b). The discrete transistor implementation is shown in Fig. 10.8(c) and the feedback block as shown in Fig. 10.8(d).

In shunt-shunt (voltage-sensing/current-comparing) feedback, shown in Fig. 10.9(a), the output voltage $v_o$ is also the input to the feedback network, and the feedback current $i_f$ is proportional to the output voltage $v_o$. The feedback network is in parallel with both the input and the output voltages. The input current $i_i$ is shared by the amplifier and the feedback network; that is, $i_i - i_f = i_c$. A shunt-shunt implementation using an op-amp is shown in Fig. 10.9(b). The discrete transistor implementation is shown in Fig. 10.9(c) and the feedback block in Fig. 10.9(d).

In shunt-series (current-sensing/current-comparing) feedback, shown in Fig. 10.10(a), the output current $i_o$ is the input to the feedback network, and the feedback current $i_f$ is proportional to the output...
FIGURE 10.9  Shunt-shunt feedback

FIGURE 10.10  Shunt-series feedback
current $i_o$. The feedback network is in parallel with the input voltage but in series with the output current. The input current is shared by the amplifier and the feedback network; that is, $i_i - i_f = i_c$. A shunt-series implementation using an op-amp is shown in Fig. 10.10(b). The discrete transistor implementation is shown in Fig. 10.10(c) and the feedback block in Fig. 10.10(d).

### 10.4.2 Feedback Relationships

There are two circuits in a feedback amplifier: the amplifier circuit (or $A$ circuit) and the feedback circuit (or $\beta$ circuit). The effective gain is always decreased by a factor of $(1 + \beta A)$. In series-type arrangements, both $A$ and $\beta$ circuits are connected in series, and the effective resistance is increased by a factor of $(1 + \beta A)$. In shunt-type arrangements, $A$ and $\beta$ circuits are connected in parallel, and the effective resistance is decreased by a factor of $(1 + \beta A)$. The effects of different types of feedback are summarized in Table 10.1. Depending on the type of feedback, an amplifier is normally represented by one of four amplifier topologies: voltage, current, transconductance, or transresistance. $A$ in Eq. (10.5) simply represents gain, which could be a voltage gain, a current gain, transconductance, or transresistance of the amplifier under the open-loop condition. Thus, $A$ could be in units of V/V, A/A, A/V, or V/A.

### Key Points of Section 10.4

- A feedback amplifier can be connected in one of four possible configurations: series-shunt, series-series, shunt-shunt, or shunt-series.
- With series feedback, the effective resistance is increased by a factor of $(1 + \beta A)$, whereas with shunt feedback the effective resistance is reduced by a factor of $(1 + \beta A)$.
10.5 Analysis of Feedback Amplifiers

In op-amp circuits, the open-loop gain $A$ is independent of the feedback factor $\beta$; that is, the feedback network does not influence $A$. The first step in analyzing a feedback amplifier is to identify the main amplifier and its feedback network. However, in BJT and MOSFET amplifiers, the feedback is realized internally, and the feedback network cannot be separated from the main amplifier without affecting the open-loop gain $A$. Thus, the feedback network does influence $A$. The analysis of a feedback amplifier can be simplified by following these steps:

**Step 1.** Identify the feedback network.

**Step 2.** Identify the type of feedback on the input and output sides.

**Step 3.** Take into account the effects of the feedback network on the open-loop gain $A$ by modifying the amplifier as follows:

a. Short-circuit the shunt feedback side to the ground so that there is no voltage signal to the feedback network. For example, terminal $y$ of $R_F$ in Fig. 10.7(b) would be connected to the ground so that $R_1$ became parallel to $R_F$.

b. Sever the series feedback side so that there is no current signal to the feedback network. For example, the inverting terminal $x$ of the op-amp in Fig. 10.8(b) would be disconnected so that there was no current flowing into the feedback circuit and $R_1$ became in series with $R_F$.

**Step 4.** Represent the modified amplifier (from step 3) using one of the following equivalent amplifier topologies:

a. Voltage amplifier for series-shunt feedback

b. Transconductance amplifier for series-series feedback

c. Transresistance amplifier for shunt-shunt feedback

d. Current amplifier for shunt-series feedback

Calculate the values of the input resistance $R_i$, the output resistance $R_o$, and the open-loop gain $A$ (representing transconductance, voltage, transresistance, or current) of the amplifier.

**Step 5.** The output of the amplifier is the input to the feedback network. Find the feedback factor $\beta$ from one of the following two-port representations of the feedback network:

a. Voltage gain ($V/V$) representation for series-shunt feedback

b. Transresistance ($V/A$) representation for series-series feedback

c. Transconductance ($A/V$) representation for shunt-shunt feedback

d. Current gain ($A/A$) representation for shunt-series feedback

**Step 6.** Calculate the input resistance with feedback from one of the following equations:

$$R_{if} = R_i(1 + \beta A) \quad \text{for series-series and series-shunt feedback}$$  \hspace{1cm} (10.26)

$$R_{if} = \frac{R_i}{1 + \beta A} \quad \text{for shunt-series and shunt-shunt feedback}$$  \hspace{1cm} (10.27)

**Step 7.** Calculate the output resistance with feedback from one of the following equations:

$$R_{of} = R_o(1 + \beta A) \quad \text{for shunt-series and series-series feedback}$$  \hspace{1cm} (10.28)

$$R_{of} = \frac{R_o}{1 + \beta A} \quad \text{for series-shunt and shunt-shunt feedback}$$  \hspace{1cm} (10.29)
TABLE 10.2  V-I representations of amplifiers and feedback networks

<table>
<thead>
<tr>
<th>Feedback Type</th>
<th>Amplifier</th>
<th>Units of $A$</th>
<th>Feedback Network</th>
<th>Units of $\beta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Series-shunt</td>
<td>V-V</td>
<td>V/V</td>
<td>V-V</td>
<td>V/V</td>
</tr>
<tr>
<td>Shunt-series</td>
<td>I-I</td>
<td>A/A</td>
<td>I-I</td>
<td>A/A</td>
</tr>
<tr>
<td>Series-series</td>
<td>V-I</td>
<td>$\bar{\Omega}$</td>
<td>I-V</td>
<td>$\Omega$</td>
</tr>
<tr>
<td>Shunt-shunt</td>
<td>I-V</td>
<td>$\Omega$</td>
<td>V-I</td>
<td>$\bar{\Omega}$</td>
</tr>
</tbody>
</table>

Step 8. Use the following feedback equation to find the closed-loop gain $A_f$:

$$A_f = \frac{A}{1 + \beta A} \quad (10.30)$$

where $\beta$ is the feedback factor representing the voltage gain, current gain, transconductance, or transresistance of the feedback network.

The V-I representations of the amplifier and its feedback network for different types of feedback are shown in Table 10.2. It is important to note that the units of $A$ and $\beta$ are different in each type of representation. To use the generalized equations for $R_{if}$, $R_{of}$, and $A_f$, the specific representation is essential.

**NOTE** Since the units of $A$ can be different depending on the type of feedback configuration, we will use the symbol $\mu_s$ for the open-loop voltage gain of the amplifier.

10.6 Series-Shunt Feedback

Series-shunt feedback is normally applied to a voltage amplifier. The amplifier in Fig. 10.7(a) is replaced by a voltage amplifier with an input resistance of $R_i$, an output resistance of $R_o$, and an open-loop voltage gain of $A$ (in V/V). This arrangement is shown in Fig. 10.11. The feedback voltage $v_f$ is in series with the input voltage $v_i$ and is proportional to the output voltage $v_o$. The feedback network can be considered as a two-port network, and it can be modeled as a voltage gain circuit, as in Fig. 10.11, with an input resistance of $R_s$, an output resistance of $R_x$, and an open-loop voltage gain of $\beta$. 

![Series-shunt configuration](image-url)
Determination of the model parameters requires separating the feedback network and representing it as a two-port network of four terminals. The test conditions for determining the parameters of the feedback network are shown in Fig. 10.12. The model parameters are obtained from three equations. The first equation is

$$\beta = \frac{v_f}{v_0} \bigg|_{i_f=0}$$

which is obtained by applying a test voltage of $v_f$ at the $v_f$ side and shorting the $v_o$ side. Note that the voltage across a short circuit is zero, and no current flows through an open circuit. The second equation is

$$R_x = \frac{v_f}{i_f} \bigg|_{v_o=0}$$

which is obtained by applying a test voltage of $v_o$ at the $v_o$ side and open-circuiting the $v_f$ side. The third equation is

$$R_y = \frac{v_o}{i_y} \bigg|_{i_f=0}$$

which is obtained by applying a test voltage of $v_o$ at the $v_o$ side and open-circuiting the $v_f$ side.

**NOTE** In performing tests to find $R_x$ and $R_y$ of a feedback network, it is helpful to remember the following general rule: Short-circuit the terminals with shunt feedback, and open-circuit the terminals with series feedback.

### 10.6.1 Analysis of an Ideal Series-Shunt Feedback Network

The analysis of series-shunt feedback can be simplified by assuming an ideal feedback network and neglecting the effects of $R_s$ and $R_L$; that is, $R_s = 0$, $R_L = \infty$, $R_x = 0$, and $R_y = \infty$. An ideal feedback network is shown in Fig. 10.13(a); it can be represented by the equivalent circuit shown in Fig. 10.13(b). The feedback factor $\beta$ is in V/V, and

$$v_o = A v_e$$
The feedback signal \( v_f \), which is proportional to the output voltage \( v_o \), is
\[
(10.35)
\]
and
\[
(10.36)
\]
From Eqs. (10.34), (10.35), and (10.36), the equation for the closed-loop voltage gain \( A_f \) becomes
\[
(10.37)
\]
which is similar to Eq. (10.5). From Eq. (10.36),
\[
(10.38)
\]
The input current \( i_i \) is
\[
(10.39)
\]
Substituting \( v_s \) from Eq. (10.38) gives the input resistance \( R_{if} \) with feedback:
\[
(10.40)
\]
The input resistance \( R_{if} \) is always increased by a factor of \((1 + \beta A)\) with series feedback at the input side.

The output resistance with feedback, which is Thevenin’s equivalent resistance, can be obtained by applying a test voltage \( v_x \) to the output side and shorting the input source. The equivalent circuit for determining Thevenin’s equivalent resistance is shown in Fig. 10.14. We have
\[
(10.41)
\]
and
\[
(10.42)
\]
Substituting $v_e$ from Eq. (10.41) into Eq. (10.42) yields

$$i_x = \frac{v_x - A(-\beta v_x)}{R_o} = \frac{(1 + \beta A)v_x}{R_o}$$

which gives the output resistance $R_{of}$ with feedback as

$$R_{of} = \frac{R_o}{1 + \beta A}$$

Thus, the output resistance $R_{of}$ at the output side is reduced by a factor of $(1 + \beta A)$. Shunt feedback at the output side always lowers the output resistance by a factor of $(1 + \beta A)$.

Series-shunt feedback increases the input resistance by $(1 + \beta A)$ and reduces the output resistance by $(1 + \beta A)$. This type of feedback is normally applied to a voltage amplifier. The input impedance, output impedance, and overall voltage gain can be written in generalized form in Laplace’s domain of $s$ as follows:

$$Z_{df}(s) = [1 + \beta A(s)]Z_i(s)$$

(10.45)

$$Z_{of}(s) = \frac{Z_o(s)}{1 + \beta A(s)}$$

(10.46)

$$A_f(s) = \frac{A(s)}{1 + \beta A(s)}$$

(10.47)

**NOTES**

1. $A$ is the open-loop voltage gain of the amplifier, in V/V.
2. $\beta$ is the voltage gain of the feedback network and is less than or equal to 1 V/V.
3. $T_l = \beta A$ is the loop gain, which is dimensionless.
4. If the source has an impedance, then the overall voltage gain is reduced. The effective input voltage $v_i$ to the amplifier can be found from

$$v_i = \frac{Z_i}{Z_s + Z_i} v_x$$

(10.48)

where $Z_s$ is source impedance, $Z_i$ is input impedance of the amplifier, and $v_x$ is signal source voltage.
10.6.2 Analysis of a Practical Series-Shunt Feedback Network

The feedback network in Fig. 10.11 has a finite input resistance $R_y$ and an output resistance $R_x$, which load the original amplifier, thereby affecting the performance of the feedback amplifier. The analysis in the previous section did not take into account the loading effect of the feedback network. The loading effect can be taken into account by including $R_s$, $R_x$, $R_y$, and $R_L$ in the $A$ circuit, as shown in Fig. 10.15(a). The open-loop parameters are modified accordingly, as shown in Fig. 10.15(b). These modifications allow us to apply the equations for ideal feedback.

The equivalent input resistance $R_{ie}$ is given by

$$R_{ie} = R_i + R_x + R_s \tag{10.49}$$

where $R_s$ is the source resistance. The equivalent output resistance $R_{oe}$ is given by

$$R_{oe} = R_o \parallel R_y \parallel R_L \tag{10.50}$$

Using the voltage divider rule in Fig. 10.15(a), we see that the output voltage $v_o$ is given by

$$v_o = \frac{R_y \parallel R_L}{(R_y \parallel R_L) + R_o} A v_e \tag{10.51}$$

where $v_e$ is the voltage across $R_i$ but not across $R_{ie}$. We need to find the voltage across $R_i$. Using the voltage divider rule gives $v_e$ as

$$v_e = \frac{R_i}{R_i + R_x + R_s} v_{e1}$$

Substituting $v_e$ into Eq. (10.51) gives the modified open-loop gain $A_e$:

$$A_e = \frac{v_o}{v_{e1}} = \frac{R_y \parallel R_L}{(R_y \parallel R_L) + R_o} \times \frac{R_i}{R_i + R_x + R_s} A \tag{10.52}$$

If $R_{ie}$, $R_{oe}$, and $A_e$ are substituted for $R_i$, $R_o$, and $A$, respectively, Eqs. (10.34) through (10.48) can be applied to calculate the closed-loop parameters $R_{if}$, $R_{of}$, and $A_f$.

![FIGURE 10.15 Practical series-shunt feedback](image-url)
EXAMPLE 10.4

Finding the performance of a noninverting amplifier with series-shunt feedback  The noninverting amplifier shown in Fig. 10.7(b) has $R_L = 10 \, \text{k}\Omega$ and $R_s = 5 \, \text{k}\Omega$. The feedback resistors are $R_1 = 10 \, \text{k}\Omega$ and $R_F = 90 \, \text{k}\Omega$. The op-amp parameters are $R_i = 2 \, \text{M}\Omega$ and $R_o = 75 \, \text{Ω}$, and the open-loop voltage gain is $\mu_g = 2 \times 10^5$.

(a) Determine the input resistance seen by the source $R_{if} = v_s/i_s$, the output resistance $R_{of}$, and the closed-loop voltage gain $A_v = v_o/v_s$.

(b) Use PSpice/SPICE to verify your results.

SOLUTION

$R_L = 10 \, \text{k}\Omega$, $R_s = 5 \, \text{k}\Omega$, $R_1 = 10 \, \text{k}\Omega$, $R_F = 90 \, \text{k}\Omega$, $R_i = 2 \, \text{M}\Omega$, $R_o = 75 \, \text{Ω}$, and $\mu_g = 2 \times 10^5$. Replacing the op-amp by its equivalent circuit gives the amplifier shown in Fig. 10.16(a).

(a) The steps in analyzing the feedback network are as follows:

Step 1. $R_1$ and $R_F$, which constitute the feedback network as shown in Fig. 10.16(a), produce a feedback voltage $v_f$ proportional to the output voltage $v_o$.

![Amplifier diagram](image)

(b) Equivalent A circuit

(c) $\beta$ determination

FIGURE 10.16  Noninverting amplifier with series-shunt feedback
Step 2. The amplifier uses series-shunt feedback. Thus, $A$ must be in $V/V$; $A = \mu g = 2 \times 10^5$.

Step 3. The effect of the feedback network at the input side is taken into account by short-circuiting the shunt feedback at the output side. Similarly, the effect at the output side is taken into account by severing the series feedback at the input side. We ground $R_F$ on side 2 and sever the feedback circuit on side 1 from $R_I$. This modification is shown in Fig. 10.16(b). Then

$$R_x = R_1 \parallel R_F = 10 \, k \parallel 90 \, k = 9 \, k\Omega$$
$$R_y = R_1 + R_F = 10 \, k + 90 \, k = 100 \, k\Omega$$

Step 4. If we represent the amplifier of Fig. 10.16(b) by an equivalent voltage amplifier, the input resistance is

$$R_{ie} = R_s + R_1 + (R_1 \parallel R_F) = R_s + R_1 + R_s = 5 \, k + 2 \, M + 9 \, k = 2014 \, k\Omega$$

and the output resistance is

$$R_{of} = R_o \parallel (R_1 + R_F) \parallel R_L = 75 \parallel (10 \, k + 90 \, k) \parallel 10 \, k = 74.4 \, \Omega$$

From Eq. (10.52), the modified open-loop gain $A_e$ is

$$A_e = \frac{(R_1 + R_F) \parallel R_L}{(R_1 + R_F) \parallel R_o + R_s} \times \frac{R_i}{R_s + R_i + (R_1 \parallel R_F)A}$$

$$= \frac{[(10 \, k + 90 \, k) \parallel 2000 \, k \times 2 \times 10^5]}{[(10 \, k + 90 \, k) \parallel 10 \, k + 75] (5 \, k + 2000 \, k + 9 \, k)} = 1.9698 \times 10^5$$

Step 5. From Fig. 10.16(c), the feedback factor $\beta$ is given by

$$\beta = \left| \frac{v_f}{v_o} \right|_{v_i=0} = \frac{R_i}{R_1 + R_F} = \frac{10 \, k}{10 \, k + 90 \, k} = 0.1 \, V/V$$

Step 6. The input resistance (seen by the source) with feedback is

$$R_{if} = \frac{v_i}{i_s} = R_{of}(1 + \beta A_e) = 2014 \, k \times (1 + 0.1 \times 1.9698 \times 10^5) = 39.67 \, G\Omega$$

Step 7. The output resistance with feedback is

$$R_{of} = \frac{R_{oe}}{1 + \beta A_e} = \frac{74.4}{1 + 0.1 \times 1.9698 \times 10^5} = 3.778 \, m\Omega$$

Step 8. The closed-loop voltage gain $A_f$ is

$$A_f = \frac{v_o}{v_s} = \frac{A_e}{1 + \beta A_e} = \frac{1.9698 \times 10^5}{1 + 0.1 \times 1.9698 \times 10^5} = 9.999 \, V/V$$

which is very close to the closed-loop gain we would get if we were to use Eq. (3.18):

$$A_f = 1 + \frac{R_g}{R_1} = 1 + \frac{90 \, k}{10 \, k} = 10$$

(b) The series-shunt feedback circuit for PSpice simulation is shown in Fig. 10.17. The results of PSpice simulation are shown below, with hand calculations to the right:

- $V(5)/V_S=9.999E+00=9.999$
- $A_f = 9.99$
- INPUT RESISTANCE AT $V_S=3.967E+10=39.67 \, G\Omega$
- $R_{if} = 39.67 \, G\Omega$
- OUTPUT RESISTANCE AT $V(5)=3.776E-02=37.7 \, m\Omega$
- $R_{of} = 37.8 \, m\Omega$

The PSpice results are very close to the hand-calculated values.
Finding the performance of a BJT amplifier with series-shunt feedback

The AC equivalent circuit of a BJT amplifier is shown in Fig. 10.18(a). The transistor can be modeled as shown in Fig. 10.18(b). The DC bias currents of the transistors are $I_{C1} = 0.5 \, \text{mA}$, $I_{C2} = 1 \, \text{mA}$, and $I_{C3} = 5 \, \text{mA}$. The transistor parameters are $h_{fe} = h_{fe1} = h_{fe2} = h_{fe3} = 100$ and $r_o = r_{o3} = \infty$.

(a) Use the techniques of feedback analysis to calculate the input resistance $R_{if}$, the output resistance $R_{of}$, and the closed-loop voltage gain $A_f$.

(b) Use PSpice/SPICE to check your results.
NOTE: The following analysis can also be applied to MOSFET amplifiers by substituting a very large value of the B-E resistance $r_{n}$ tending to infinity, $1/r_{n} \approx 0$.

**SOLUTION**

For $I_{C1} = 0.5$ mA, Eqs. (8.40) and (8.42) give

$$r_{\pi1} = \frac{h_{fe} \times 25.8 \text{ mV}}{I_{C1}} = \frac{100 \times 25.8 \text{ mV}}{0.5 \text{ mA}} = 5.16 \text{ kΩ}$$

$$g_{m1} = \frac{I_{C1}}{V_{T}} = \frac{0.5 \text{ mA}}{25.8 \text{ mV}} = 19.38 \text{ mA/V}$$

For $I_{C2} = 1$ mA,

$$r_{\pi2} = \frac{h_{fe} \times 25.8 \text{ mV}}{I_{C2}} = \frac{100 \times 25.8 \text{ mV}}{1 \text{ mA}} = 2.58 \text{ kΩ}$$

$$g_{m2} = \frac{I_{C2}}{V_{T}} = \frac{1 \text{ mA}}{25.8 \text{ mV}} = 38.76 \text{ mA/V}$$

For $I_{C3} = 5$ mA,

$$r_{\pi3} = \frac{h_{fe} \times 25.8 \text{ mV}}{I_{C3}} = \frac{100 \times 25.8 \text{ mV}}{5 \text{ mA}} = 516 \text{ Ω}$$

$$g_{m3} = \frac{I_{C3}}{V_{T}} = \frac{5 \text{ mA}}{25.8 \text{ mV}} = 193.8 \text{ mA/V}$$

(a) The steps in analyzing the feedback network are as follows:

**Step 1.** $R_{E}$ and $R_{F}$, which constitute the feedback network, produce a feedback voltage proportional to the output voltage. The input voltage $v_{i}$ is compared with the feedback signal $v_{f}$. The error voltage $v_{e} = v_{i} - v_{f}$ is the B-E voltage of transistor Q1. The block diagram representing the feedback mechanism is shown in Fig. 10.19(a). The feedback network is shown in Fig. 10.19(b).

![Block diagram](image)

$$g_{m1} = \frac{I_{C1}}{V_{T}} = \frac{0.5 \text{ mA}}{25.8 \text{ mV}} = 19.38 \text{ mA/V}$$

$$r_{\pi1} = \frac{h_{fe} \times 25.8 \text{ mV}}{I_{C1}} = \frac{100 \times 25.8 \text{ mV}}{0.5 \text{ mA}} = 5.16 \text{ kΩ}$$

**FIGURE 10.19** Equivalent circuits for Example 10.5
Step 2. The amplifier uses series-shunt feedback. Thus, the voltage gain $A$ must be expressed in V/V.

Step 3. The effect of the feedback network is taken into account at the input side by shorting $R_F$ at side 2 to the ground, as shown in Fig. 10.19(b), and at the output side by removing the feedback network from the emitter of $Q_1$. These modifications are shown in the small-signal AC equivalent circuit in Fig. 10.19(c).

Step 4. If we represent the amplifier of Fig. 10.19(c) by an equivalent voltage amplifier, the input resistance at the base of $Q_1$ is

$$R_i = \frac{v_i}{i_{b1}} = r_{\pi 1} + (1 + g_{m1}r_{\pi 1})(R_E || R_F) = 5.16 \, k \Omega + 101 \times (100 || 650) = 13.91 \, k\Omega$$

The input resistance is

$$R_{ie} = \frac{v_{el}}{i_{b1}} = R_s + R_i = R_s + r_{\pi 1} + (1 + g_{m1}r_{\pi 1})(R_E || R_F) = 150 + 13.91 \, k \Omega = 14.06 \, k\Omega$$

The output resistance is

$$R_{oe} = R_L || (R_F + R_E) \left( \frac{R_2 + r_{\pi 3}}{1 + g_{m3}r_{\pi 1}} \right) = 100 || 750 \left( \frac{5 \, k + 516}{101} \right) = 33.74 \, \Omega$$

Thus,

$$v_{be1} = \left( \frac{r_{\pi 1}}{R_i} \right) v_i = \left( \frac{5.16 \, k}{13.91 \, k} \right) v_i = 0.371 v_i$$

$$v_{b2} = v_{b2} = -(R_1 || r_{\pi 2})g_{m1}v_{be1} = -(9 \, k || 2.58 \, k) \times 19.38 \, m \times 0.371 v_i = -14.412 v_i$$

$$v_{b3} = -\{R_2[\{r_{\pi 3} + (1 + g_{m3}r_{\pi 3})R_L || (R_F + R_E)\}]g_{m2}v_{be2}\}$$

$$= -\{5 \, k \left[516 + (1 + 193.8 \, m \times 516)(100 || 750)\right] \times 38.76 \, m \times (-14.412 v_i) = 1.825 \times 10^3 v_i$$

$$v_o = \frac{v_{b3} \times (1 + g_{m3}r_{\pi 3})[R_L || (R_F + R_E)]}{\{r_{\pi 3} + (1 + g_{m3}r_{\pi 3})[R_L || (R_F + R_E)]\}}$$

$$= \frac{(1.825 \times 10^3 v_i) \times (1 + 193.8 \, m \times 516)(100 || 750)}{[516 + (1 + 193.8 \, m \times 516)(100 || 750)]} = 1725.7 v_i$$

Therefore, the open-loop voltage gain $A$ is given by $A = v_o / v_i = 1725.7 \, V/V$

and $A_{fe} = \frac{A}{v_{el}} = \frac{AR_i}{R_s + R_i} = 1725.7 \times 13.91 \, k \Omega = 1707.3 \, V/V$

Step 5. From Fig. 10.19(b) and Eq. (10.33), the feedback factor $\beta$ is given by

$$\beta = \frac{v_i}{v_o} \Big|_{i=0} = \frac{R_E}{R_E + R_F} = \frac{100}{100 + 650} = 0.1333 \, V/V$$

Step 6. The input resistance (seen by the source) with feedback is

$$R_{id} = \frac{v_s}{i_s} = R_{ie}(1 + \beta A_{fe}) = 14.06 \, k\Omega \times (1 + 0.1333 \times 1707.3) = 3.21 \, M\Omega$$

Step 7. The output resistance with feedback is

$$R_{of} = \frac{R_{oe}}{1 + \beta A_{fe}} = \frac{33.74}{1 + 0.1333 \times 1707.3} = 0.147 \, \Omega$$

Step 8. The closed-loop voltage gain $A_f$ is

$$A_f = \frac{v_o}{v_s} = \frac{A_{fe}}{1 + \beta A_{fe}} = \frac{1707.3}{1 + 0.1333 \times 1707.3} = 7.47 \, V/V$$
The series-shunt feedback network for PSpice simulation is shown in Fig. 10.20. The results of PSpice simulation (.TF analysis) are shown below, with hand calculations to the right:

\[ \frac{V(8)}{V_{S}} = 7.468 \times 10^{0} = 7.468 \text{ V/V} \]

\[ A_f = 7.47 \text{ V/V} \]

INPUT RESISTANCE AT \( V_S = 3.214 \times 10^{6} = 3.214 \text{ M\Omega} \)

OUTPUT RESISTANCE AT \( V(8) = 1.460 \times 10^{-1} = 0.146 \Omega \)

Note the close agreement between the results obtained by PSpice and the hand calculations.

**FIGURE 10.20** Series-shunt feedback network for PSpice simulation

---

**KEY POINTS OF SECTION 10.6**

- Series-shunt feedback is applied to voltage amplifiers. This type of feedback increases the input resistance and reduces the output resistance by a factor of \( 1 + \beta A \).
- The amplifier is represented by a voltage amplifier and the feedback network as a voltage gain. Both \( A \) and \( \beta \) are in units of V/V.
- The loading effect of a feedback network can be taken into account by remembering the following general rule: Short-circuit the terminals with shunt feedback, and open-circuit the terminals with series feedback.

---

**10.7 Series-Series Feedback**

Series-series feedback is normally applied to a transconductance amplifier. For the series-series feedback amplifier in Fig. 10.8(a), we obtain the equivalent circuit shown in Fig. 10.21 if we represent the op-amp by its transconductance model. \( A \) is the open-loop transconductance gain of the op-amp in A/V. That is, \( A_v = \mu_k v_C / R_o \) and \( A = \mu_k / R_o \). The feedback voltage \( v_f \) is proportional to the load current \( i_o \). The feedback network can also be modeled in transimpedance form with an input resistance of \( R_x \), an output resistance of \( R_y \), and a transimpedance of \( \beta \) in V/A.
The test conditions for determining the parameters of the feedback network are shown in Fig. 10.22. The parameters of the model can be obtained from three equations. The first equation is

\[ R_x = \left. \frac{v_f}{i_f} \right|_{i_o = 0} \]  
(open-circuited output side)  
(10.53)

which is obtained by applying a test voltage of \( v_f \) at side 1 and open-circuiting side 2. The second equation is

\[ R_y = \left. \frac{v_y}{i_o} \right|_{i_i = 0} \]  
(open-circuited input side)  
(10.54)

which is obtained by applying a test voltage of \( v_y \) at side 2 and open-circuiting side 1. The third equation is

\[ \beta = \left. \frac{v_f}{i_o} \right|_{i_i = 0} \]  
(open-circuited input side)  
(10.55)

which is obtained by applying a test voltage of \( v_y \) at side 2 and open-circuiting side 1.

\[ \beta \text{ is obtained from} \]

\[ R_x \text{ is obtained from} \]

\[ R_y \text{ is obtained from} \]

\[ v_f \]

\[ v_y \]

\[ i_f \]

\[ i_o \]

\[ i_i \]

(a) \( \beta = \left. \frac{v_f}{i_o} \right|_{i_i = 0} \)  
(b) \( R_x = \left. \frac{v_f}{i_f} \right|_{i_o = 0} \)  
(c) \( R_y = \left. \frac{v_y}{i_o} \right|_{i_i = 0} \)

FIGURE 10.22 Test conditions for determining the parameters of a series-series feedback network
10.7.1 Analysis of an Ideal Series-Series Feedback Network

Let us assume an ideal series-series feedback network—that is, \( R_x = 0, \ R_y = 0, \ R_s = 0, \) and \( R_L = 0. \) The feedback amplifier in Fig. 10.21 can be simplified to the one in Fig. 10.23(a), which can be represented by the equivalent circuit shown in Fig. 10.23(b). For this circuit,

\[
\begin{align*}
    i_o &= A v_e \\
    v_f &= \beta i_o \\
    v_s &= v_e + v_f = \frac{i_o}{A} + \beta i_o = \frac{1 + \beta A}{A} i_o
\end{align*}
\]

which gives the closed-loop transconductance gain \( A_f \) as

\[
A_f = \frac{i_o}{v_s} = \frac{A}{1 + \beta A}
\]  

(10.56)

Using KVL around the input side, we get

\[
v_s = R_i i_s + v_f = R_i i_s + \beta i_o = R_i i_s + \beta A v_e = R_i i_s + \beta A R_i i_s
\]

which gives the closed-loop input resistance \( R_{if} \) as

\[
R_{if} = \frac{v_s}{i_s} = R_i (1 + \beta A)
\]  

(10.57)

To determine the output resistance with feedback, let us apply a test voltage \( v_x, \) as shown in Fig. 10.24:

\[
v_x = R_o (i_o - A v_e) = R_o (i_o + \beta A i_o) = R_o (1 + \beta A) i_o
\]

which gives the closed-loop output resistance \( R_{of} \) as

\[
R_{of} = \frac{v_x}{i_x} = R_o (1 + \beta A)
\]  

(10.58)

![Figure 10.23: Ideal series-series feedback network](image)
1. $A$ is the open-loop transconductance of the amplifier, in A/V.
2. $\beta$ is the transresistance of the feedback network, in V/A.
3. $T_L = \beta A$ is the loop gain, which is dimensionless.

### 10.7.2 Analysis of a Practical Series-Series Feedback Network

The analysis in the previous section did not take into account the loading effect of the feedback network. The open-loop parameters of the amplifier in Fig. 10.25(a) can be modified to include the loading effect due to $R_s$, $R_x$, $R_y$, and $R_L$, providing the equivalent ideal feedback network in Fig. 10.25(b). The modified parameters are given by

$$R_{ie} = R_s + R_i + R_x$$

$$R_{oe} = R_o + R_y + R_L$$

**NOTE**  It may appear from Fig. 10.25(b) that $R_o$ would have to be in parallel with $(R_y + R_L)$ for us to find $R_{oe}$ in Eq. (10.60), but if we convert the current source $A_v e$ to a voltage source, we see that the effective resistance used to find the current $i_o$ becomes $(R_o + R_y + R_L)$.

**FIGURE 10.24**  Test circuit for finding output resistance $R_{of}$

**FIGURE 10.25**  Practical series-series feedback amplifier
Substituting \( v_e \) from Eq. (10.62) into Eq. (10.61) gives the modified open-loop transconductance \( A_e \):

\[
i_o = \frac{R_o}{R_o + R_L + R_y} A v_e \tag{10.61}
\]

\[
v_e = \frac{R_i}{R_i + R_s + R_s} v_{e1} \tag{10.62}
\]

If the values of \( R_i, R_o, \) and \( A \) are replaced by \( R_{ie}, R_{oe}, \) and \( A_e \), respectively, then Eqs. (10.56) through (10.58) can be applied to calculate the closed-loop parameters \( R_{if}, R_{of}, \) and \( A_f \).

\[
A_e = \frac{i_o}{v_{e1}} = \frac{R_o R_i}{(R_o + R_L + R_y)(R_s + R_i + R_s)} A \tag{10.63}
\]

EXAMPLE 10.6

Finding the performance of a noninverting amplifier with series-series feedback  The noninverting amplifier shown in Fig. 10.8(b) has \( R_L = 4 \, \Omega \) and \( R_s = 5 \, k\Omega \). The feedback resistance is \( R_F = 5 \, \Omega \). The op-amp parameters are \( R_i = 2 \, M\Omega \) and \( R_o = 75 \, \Omega \), and the open-loop voltage gain is \( \mu g = 2 \times 10^5 \).

(a) Determine the input resistance seen by the source \( R_{if} = v_s / i_s \), the output resistance \( R_{of} \), and the closed-loop transconductance gain \( A_f = i_o / v_s \).

(b) Use PSpice/SPICE to verify your results.

**SOLUTION**

\( R_L = 4 \, \Omega, R_s = 5 \, k\Omega, R_F = 5 \, \Omega, R_i = 2 \, M\Omega, R_o = 75 \, \Omega, \) and \( \mu g = 2 \times 10^5 \). Replacing the op-amp in Fig. 10.8(b) by its equivalent circuit gives the amplifier shown in Fig. 10.26(a).

(a) The steps in analyzing the feedback network are as follows:

Step 1. \( R_F \) constitutes the feedback network, and it produces a feedback voltage \( v_f \) proportional to the output current \( i_o \).

Step 2. The amplifier uses series-series feedback. Thus, \( A \) must be in \( A/V \):

\[
A = \frac{\mu g}{R_o} = \frac{2 \times 10^5}{75} = 2.67 \, kA/V
\]

Step 3. The effect of the feedback network is taken into account by severing \( R_F \) from the op-amp at side 1 and from \( R_L \) at side 2. This modification is shown in Fig. 10.26(b). Then

\[
R_x = R_F = 5 \, \Omega
\]

\[
R_y = R_F = 5 \, \Omega
\]

Step 4. If we represent the amplifier of Fig. 10.26(b) by an equivalent voltage amplifier, the input resistance is

\[
R_{ie} = R_s + R_i + R_x = R_s + R_i + R_F = 5 \, k + 2 \, M + 5 \approx 2005 \, k\Omega
\]

and the output resistance is

\[
R_{oe} = R_o + R_F + R_L = 75 + 5 + 4 = 84 \, \Omega
\]
We find the modified open-loop transconductance gain $A_e$ from Eq. (10.63):

$$A_e = \frac{R_o R_i}{(R_o + R_L + R_F)(R_s + R_i + R_F)}$$

$$= \frac{75 \times 2000 \text{ k} \times 2.67 \text{ kA/V}}{(75 + 4 + 5)(5 \text{ k} + 2000 \text{ k} + 5)} = 2.375 \times 10^3 \text{ A/V}$$

Step 5. From Fig. 10.26(c), the feedback factor $\beta$ is given by

$$\beta = \frac{v_f}{i_o} = R_F = 5 \text{ } \Omega$$

Step 6. The input resistance (seen by the source) with feedback is

$$R_{if} = \frac{v_i}{i_s} = R_{if}(1 + \beta A_e) = 2005 \text{ k} \times (1 + 5 \times 2.375 \times 10^3) = 23.81 \text{ G}\Omega$$

Step 7. The output resistance with feedback is

$$R_{of} = R_{oe}(1 + \beta A_e) = 84 \times (1 + 5 \times 2.375 \times 10^3) = 997.6 \text{ k}\Omega$$
Step 8. The closed-loop transconductance gain \( A_f \) is

\[
A_f = \frac{i_o}{v_s} = \frac{A_e}{1 + \beta A_e} = \frac{2.375 \times 10^3}{1 + 5 \times 2.375 \times 10^3} = 200 \text{ mA/V}
\]

(b) The series-series feedback circuit for PSpice simulation is shown in Fig. 10.27. The results of PSpice/SPICE simulation (.TF analysis) are shown below, with hand calculations to the right:

\[
\text{INPUT RESISTANCE AT } V_S = 2.381 \times 10^6 = 23.81 \text{ G}\Omega
\]

\[
\text{OUTPUT RESISTANCE AT } I(VX) = 9.976 \times 10^5 = 997.6 \text{ k}\Omega
\]

The PSpice results are very close to the hand-calculated values.

**EXAMPLE 10.7**

**Finding the performance of a BJT amplifier with series-series feedback**  The AC equivalent circuit of a feedback amplifier is shown in Fig. 10.28. The DC bias currents of the transistors are \( I_{C1} = 0.5 \text{ mA}, I_{C2} = 1 \text{ mA}, \) and \( I_{C3} = 5 \text{ mA}. \) The transistor parameters are \( h_{fe} = h_{fe1} = h_{fe2} = h_{fe3} = 100 \) and \( r_\mu = r_o = \infty. \)

(a) Use the techniques of feedback analysis to calculate the input resistance \( R_{if}, \) the output resistance \( R_{of}, \) and the closed-loop transconductance gain \( A_f = i_o/v_s. \)

(b) Use PSpice/SPICE to check your results.

**NOTE:** The following analysis can also be applied to MOSFET amplifiers by substituting a very large value of the B-E resistance \( r_\pi \) tending to infinity, \( 1/r_\pi \approx 0. \)

**SOLUTION**

For \( I_{C1} = 0.5 \text{ mA}, \) Eqs. (8.40) and (8.42) give

\[
r_{\pi 1} = \frac{h_{fe} \times 25.8 \text{ mV}}{I_{C1}} = \frac{100 \times 25.8 \text{ mV}}{0.5 \text{ mA}} = 5.16 \text{ k}\Omega
\]

\[
g_{m1} = \frac{I_{C1}}{V_T} = \frac{0.5 \text{ mA}}{25.8 \text{ mV}} = 19.38 \text{ mA/V}
\]
For $I_{C2} = 1 \text{ mA}$,
\[ r_{\pi2} = \frac{h_{fe} \times 25.8 \text{ mV}}{I_{C2}} = \frac{100 \times 25.8 \text{ mV}}{1 \text{ mA}} = 2.58 \text{ k}\Omega \]
\[ g_{m2} = \frac{I_{C2}}{V_T} = \frac{1 \text{ mA}}{25.8 \text{ mV}} = 38.76 \text{ mA/V} \]

For $I_{C3} = 5 \text{ mA}$,
\[ r_{\pi3} = \frac{h_{fe} \times 25.8 \text{ mV}}{I_{C3}} = \frac{100 \times 25.8 \text{ mV}}{5 \text{ mA}} = 516 \text{ } \Omega \]
\[ g_{m3} = \frac{I_{C3}}{V_T} = \frac{5 \text{ mA}}{25.8 \text{ mV}} = 193.8 \text{ mA/V} \]

(a) The steps in analyzing the feedback network are as follows:

**Step 1.** $R_3$, $R_F$, and $R_4$ constitute the feedback network. The input voltage $v_i$ is compared with the feedback signal $v_f$. An error voltage $v_e = v_i - v_f$ is the input to the base of the transistor $Q_3$. The block diagram representing the feedback mechanism is shown in Fig. 10.29(a). The feedback network is shown in Fig. 10.29(b).

**Step 2.** The amplifier uses series-series feedback. Thus, the transconductance gain $A$ must be expressed in $A/V$.

**Step 3.** The effect of the feedback network is taken into account at the input side by removing the network from the emitter of $Q_1$ at side 2, as shown in Fig. 10.29(b), and at the output side by removing the network from the emitter of $Q_1$ at side 1. These modifications are shown in the small-signal equivalent circuit in Fig. 10.29(c).

**Step 4.** If we represent the amplifier in Fig. 10.29(c) by an equivalent transconductance amplifier, the input resistance is
\[ R_i = r_{\pi1} + (1 + g_{m1}r_{\pi1}) [R_4( R_F + R_3)] = 5.16 \text{ k}\Omega + 101 \times 100 \| 750 = 14.07 \text{ k}\Omega \]
\[ R_{ie} = R_i + R_s = 14.07 \text{ k}\Omega + 150 = 14.22 \text{ k}\Omega \]

The output resistance seen at the collector of $Q_3$ is
\[ R_o = r_o = \infty \]
Feedback Amplifiers

From Fig. 10.29(c), we get

Therefore, the open-loop transconductance $A$ is given by

$$A = \frac{v_o}{v_i} = \frac{19.14 A}{V}.
$$

**Step 5.** From Eq. (10.55), the feedback factor $\beta$ is given by

$$\beta = \left. \frac{v_i}{i_o} \right|_{i_i=0} = \frac{R_3R_4}{R_3 + R_4 + R_F} = \frac{100 \times 100}{100 + 100 + 650} = 11.76 \text{ V/A}$$
Step 6. The input resistance with feedback is

\[ R_{if} = \frac{V_s}{I_s} = R_{in}(1 + \beta A_\text{e}) = 14.22 \text{ k}\Omega \times (1 + 11.76 \times 18.94) = 3.18 \text{ M}\Omega \]

Step 7. The output resistance with feedback is

\[ R_{of} = R_0(1 + \beta A) = \infty \]

Since the output is taken across \( R_C \), the output resistance of the amplifier is

\[ R_{out} = R_{of} \parallel R_C = R_C = 500 \ \Omega \]

Step 8. The closed-loop transconductance gain \( A_f \) is

\[ A_f = \frac{I_o}{V_s} = \frac{18.94}{1 + 11.76 \times 18.94} = 84.66 \text{ mA/V} \]

(b) The series-series feedback amplifier for PSpice simulation is shown in Fig. 10.30. The results of PSpice simulation (.TF analysis) are shown below, with hand calculations to the right.

- \( I(VU) / V_S = 8.379 \times 10^{-2} = 83.79 \text{ mA/V} \)
- \( A_f = 84.66 \text{ mA/V} \)
- INPUT RESISTANCE AT \( V_S = 3.215 \times 10^6 \) \( = 3.215 \text{ M}\Omega \)
- OUTPUT RESISTANCE AT \( I(VU) = 8.639 \times 10^3 \) \( = 8.639 \text{ k}\Omega \)
- \( R_{if} \approx 3.18 \text{ M}\Omega \)
- \( R_{of} = \infty \)

The PSpice results are very close to the hand-calculated values.

**KEY POINTS OF SECTION 10.7**

- Series-series feedback is applied to transconductance amplifiers. This type of feedback increases both the input resistance and the output resistance by a factor of \((1 + \beta A)\).
- The amplifier is represented by a transconductance amplifier and the feedback network as a transresistance gain. \( A \) is in units of \( \text{A/V} \), and \( \beta \) is in units of \( \text{V/A} \).
10.8 Shunt-Shunt Feedback

In shunt-shunt feedback, as shown in Fig. 10.9(a), the feedback circuit is in parallel with the amplifier. Although any amplifier can be represented by any of the four types, the analysis of shunt-shunt feedback can be simplified by representing the amplifier in transresistance form. This is shown in Fig. 10.31, in which the amplifier has an input resistance of \( R_i \), an output resistance of \( R_o \), and an open-loop transresistance gain of \( A \) (in V/A). That is, \( A = \mu g_R \). The feedback current \( i_f \) is proportional to the output voltage \( v_o \). The feedback network is modeled in transconductance form with an input resistance of \( R_y \), an output resistance of \( R_x \), and an open-loop transconductance gain of \( \beta \).

The test conditions for determining the parameters of the feedback network are shown in Fig. 10.32. Given that the voltage across a short circuit is zero and no current flows through an open circuit, the model parameters can be defined by three equations. The first equation is

\[
R_x = \frac{v_f}{i_f} \bigg|_{v_o=0} \quad \text{(short-circuited output side)} \tag{10.64}
\]

which is obtained by applying a test voltage of \( v_f \) at side 1 and short-circuiting side 2. The second equation is

\[
R_y = \frac{v_o}{i_y} \bigg|_{v_f=0} \quad \text{(short-circuited input side)} \tag{10.65}
\]

\( \beta \) is obtained from

\[
\beta = \frac{v_f}{v_o} \bigg|_{v_f=0}
\]

\( R_x \) is obtained from

\[
R_x = \frac{v_f}{i_f} \bigg|_{v_o=0}
\]

\( R_y \) is obtained from

\[
R_y = \frac{v_o}{i_y} \bigg|_{v_f=0}
\]
which is obtained by applying a test voltage of $v_o$ at side 2 and short-circuiting side 1. The third equation is

$$\beta = \frac{i_f}{v_o} \bigg|_{v=0} \quad \text{(short-circuited input side)}$$  \hspace{1cm} (10.66)

which is obtained by applying a test voltage of $v_o$ at side 2 and short-circuiting side 1.

### 10.8.1 Analysis of an Ideal Shunt-Shunt Feedback Network

Let us assume an ideal shunt-shunt feedback network—that is, $R_s = \infty$, $R_x = \infty$, $R_y = \infty$, and $R_L = 0$. The feedback amplifier in Fig. 10.31 is simplified to that shown in Fig. 10.33(a), which can be represented by the equivalent circuit shown in Fig. 10.33(b). The output voltage $v_o$ becomes

$$v_o = A i_e$$  \hspace{1cm} (10.67)

The feedback current $i_f$ is proportional to the output voltage $v_o$. That is,

$$i_f = \beta v_o$$  \hspace{1cm} (10.68)

and

$$i_e = i_s - i_f$$  \hspace{1cm} (10.69)

Substituting $i_e$ from Eq. (10.69) and $i_f$ from Eq. (10.68) into Eq. (10.67), we get the closed-loop transresistance $A_f$ as

$$A_f = \frac{v_o}{i_s} = \frac{A}{1 + \beta A}$$  \hspace{1cm} (10.70)

From Eq. (10.69),

$$i_s = i_e + i_f$$

Substituting $i_f$ from Eq. (10.68) and $v_o$ from Eq. (10.67) into the above equation gives

$$i_s = i_e + \beta v_o = i_e + \beta A i_e = i_e (1 + \beta A)$$  \hspace{1cm} (10.71)

![Ideal feedback circuit](image)

![Equivalent circuit](image)

**FIGURE 10.33** Ideal shunt-shunt feedback circuit
The error current $i_e$ is related to $v_i$ by

$$v_i = R_i i_e$$  \hspace{1cm} (10.72)

Using $i_s$ from Eq. (10.71) and $v_i$ from Eq. (10.72), we can find the input resistance with feedback $R_{if}$:

$$R_{if} = \frac{v_i}{i_s} = \frac{i_e R_i}{i_c(1 + \beta A)} = \frac{R_i}{1 + \beta A}$$  \hspace{1cm} (10.73)

The input resistance of an amplifier with shunt feedback at the input side is always decreased by a factor of $(1 + \beta A)$.

The output resistance with feedback $R_{of}$, which is Thevenin’s equivalent resistance, can be obtained by applying a test voltage $v_x$ to the output side and open-circuiting the input current source. The equivalent circuit for determining Thevenin’s equivalent output resistance is shown in Fig. 10.34. We have

$$i_e = -i_f = -\beta v_x$$  \hspace{1cm} (10.74)

and

$$i_x = \frac{v_x - 4i_e}{R_o}$$  \hspace{1cm} (10.75)

Substituting $i_e$ from Eq. (10.74) into Eq. (10.75) yields

$$i_x = \frac{v_x + \beta Av_x}{R_o} = \frac{(1 + \beta A)v_x}{R_o}$$  \hspace{1cm} (10.76)

which gives the output resistance with feedback $R_{of}$ as

$$R_{of} = \frac{R_o}{1 + \beta A}$$  \hspace{1cm} (10.77)

Thus, shunt feedback at the output always lowers the output resistance by a factor of $(1 + \beta A)$.

The input impedance, the output impedance, and the overall gain can be written in generalized form in Laplace’s domain as follows:

$$Z_{if}(s) = \frac{Z_i(s)}{1 + \beta A(s)}$$  \hspace{1cm} (10.78)

$$Z_{of}(s) = \frac{Z_o(s)}{1 + \beta A(s)}$$  \hspace{1cm} (10.79)

$$A_f(s) = \frac{A(s)}{1 + \beta A(s)}$$  \hspace{1cm} (10.80)
1. \( A \) is the open-loop transresistance of the amplifier, in V/A.
2. \( \beta \) is the transconductance of the feedback network, in A/V.
3. \( T_L = \beta A \) is the loop gain, which is dimensionless.
4. If the source has an input impedance of \( Z_s \), then the total impedance \( Z_{in} \) seen by the source will be

\[
Z_{in}(s) = \frac{v_s}{i_s} = Z_{if}(s) + Z_o(s)
\]

10.8.2 Analysis of a Practical Shunt-Shunt Feedback Network

The open-loop parameters of the amplifier in Fig. 10.35(a) can be modified to include the loading effect due to \( R_s, R_x, R_y, \) and \( R_L \), producing the equivalent ideal feedback circuit in Fig. 10.35(b). The modified parameters are given by

\[
R_{ie} = R_s \parallel R_i \parallel R_x
\]

(10.82)

\[
R_{oe} = R_o \parallel R_y \parallel R_L
\]

(10.83)

and

\[
v_o = \frac{R_y \parallel R_L}{(R_y \parallel R_L) + R_o} A_i_e
\]

(10.84)

where \( i_e \) is the current through \( R_i \) only, not through \( R_i \) and \( R_s \parallel R_x \). Thus, by the current divider rule, \( i_e \) is given by

\[
i_e = \frac{R_s \parallel R_x}{R_s \parallel R_x + R_i} i_{e1}
\]

(10.85)

Substituting \( i_e \) from Eq. (10.85) into Eq. (10.84) gives the modified open-loop transresistance gain \( A_e \):

\[
A_e = \frac{v_o}{i_{e1}} = \frac{R_y \parallel R_L}{R_y \parallel R_L + R_o} \times \frac{R_s \parallel R_x}{R_s \parallel R_x + R_i} A
\]

(10.86)

With these values of \( R_{ie}, R_{oe}, \) and \( A_e \), we can use the equations in the preceding section to calculate the closed-loop parameters \( R_{if}, R_{of}, \) and \( A_f \).
**EXAMPLE 10.8**

**Finding the performance of an inverting amplifier with shunt-shunt feedback**  The inverting amplifier shown in Fig. 10.9(b) has $R_s = 2 \, \text{k} \Omega$, $R_L = 5 \, \text{k} \Omega$, and $R_F = 8 \, \text{k} \Omega$. The op-amp parameters are $R_i = 2 \, \text{M} \Omega$ and $R_o = 75 \, \Omega$, and the open-loop voltage gain is $\mu_g = 2 \times 10^5$.

(a) Determine the input resistance seen by the source $v_i/s$, the output resistance $R_o$, the closed-loop transresistance $A_f = v_o/i_s$, and the voltage gain $A_{vf} = v_o/v_s$.

(b) Use PSpice/SPICE to verify your results.

**SOLUTION**

$R_L = 5 \, \text{k} \Omega$, $R_s = 2 \, \text{k} \Omega$, $R_F = 8 \, \text{k} \Omega$, $R_i = 2 \, \text{M} \Omega$, $R_o = 75 \, \Omega$, and $\mu_g = 2 \times 10^5$. Replacing the op-amp in Fig. 10.9(b) by its equivalent circuit gives the amplifier shown in Fig. 10.36(a).

(a) The steps in analyzing the feedback network are as follows:

**Step 1.** $R_F$ constitutes the feedback network, and it produces a feedback current $i_f$ proportional to the output voltage $v_o$.

**Step 2.** The amplifier uses shunt-shunt feedback. Thus, $A$ must be in V/A. Converting the voltage-controlled voltage source to a current-controlled voltage source, we get

$$v_o = -\mu_g v_c = -\mu_g R_i i_c = A i_c$$

which gives the open-loop transresistance $A$:

$$A = -\mu_g R_i = -2 \times 10^5 \times 2 \times 10^6 = -4 \times 10^{11} \, \text{V/A}$$

**FIGURE 10.36** Inverting op-amp amplifier
Step 3. The effect of the feedback network is taken into account by shorting $R_F$ to the ground at sides 1 and 2. This arrangement is shown in Fig. 10.36(b). Then

$$R_e = R_F = 8 \, \text{k}\Omega$$
and

$$R_y = R_F = 8 \, \text{k}\Omega$$

Step 4. If we represent the amplifier of Fig. 10.36(a) by an equivalent transresistance amplifier, as shown in Fig. 10.36(b), we have

$$R_{ic} = R_i \parallel R_F \parallel R_L = 2 \, \text{k} \parallel 2 \, \text{M} \parallel 8 \, \text{k} = 1.6 \, \text{k}\Omega$$
and

$$R_{oe} = R_o \parallel R_F \parallel R_L = 75 \parallel 8 \, \text{k} \parallel 5 \, \text{k} = 73.2 \, \text{\Omega}$$

From Eq. (10.86), we find that the modified open-loop transresistance gain $A_e$ is

$$A_e = \frac{v_o}{i_{e1}} = \frac{-R_F \parallel R_L}{(R_F \parallel R_L) + R_o} \times \frac{R_s \parallel R_F - A}{R_s \parallel R_F + R_i}$$

$$= \frac{-(8 \, \text{k} \parallel 5 \, \text{k}) \times (2 \, \text{k} \parallel 8 \, \text{k}) \times 4 \times 10^6}{(8 \, \text{k} \parallel 5 \, \text{k} + 75)(2 \, \text{k} \parallel 8 \, \text{k} + 2 \, \text{M})} = -312.1 \, \text{\Omega}$$

Step 5. From Fig. 10.36(c), the feedback factor $\beta$ is given by

$$\beta = \left. \frac{i_f}{v_o} \right|_{v_y = 0} = -\frac{1}{R_F} = -125 \, \mu\Omega$$

The loop gain is

$$T_L = \beta A_e = 125 \, \mu\Omega \times 312.1 \times 10^6 = 39.02 \, \text{\Omega}$$

Step 6. The input resistance at the input side of the op-amp is

$$R_{id} = \frac{R_{ic}}{1 + \beta A_e} = \frac{R_{ic}}{1 + T_L}$$

$$= \frac{1.6 \, \text{k}\Omega}{1 + 39.02 \times 10^3} = 41.0 \, \text{m}\Omega$$

Step 7. The output resistance with feedback is

$$R_{of} = \frac{R_{oe}}{1 + \beta A_e} = \frac{R_{oe}}{1 + T_L} = \frac{73.2 \, \text{\Omega}}{1 + 39.02 \times 10^3} = 1.876 \, \text{\text{\mu}\Omega}$$

Step 8. The closed-loop transresistance gain $A_f$ is

$$A_f = \frac{v_o}{i_s} = \frac{A_e}{1 + \beta A_e} = \frac{A_o}{1 + T_L}$$

$$= \frac{-312.1 \times 10^6}{1 + 39.02 \times 10^3} = -8 \, \text{k}\Omega$$

$$v_o = \left( \frac{v_o}{v_s} \right) v_s = \left( \frac{v_o}{i_s} \right) \frac{i_s}{v_s} v_s = \left( \frac{-8 \, \text{k}}{2 \, \text{k}} \right) v_s = -4 v_s$$

Therefore, the overall voltage gain is $v_o / v_s = -4$.

**NOTE:** Substituting $R_1 = R_s$ in Eq. (3.40) gives the voltage gain of the inverting amplifier as $R_F / R_s = -8 \, \text{k} / 2 \, \text{k} = -4$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
The shunt-shunt feedback circuit for PSpice simulation is shown in Fig. 10.37. The results of the simulation (.TF analysis) are shown below, with hand calculations to the right:

\[
\frac{V(5)}{I_s} = -8.000E+03 = -8 \text{ k}\Omega
\]
\[
A_f = -8 \text{ k}\Omega
\]
\[
\text{INPUT RESISTANCE AT } I_s = 4.097E-02 = 40.97 \text{ m}\Omega
\]
\[
R_{if} = 41.0 \text{ m}\Omega
\]
\[
\text{OUTPUT RESISTANCE AT } V(5) = 1.876E-03 = 1.876 \text{ m}\Omega
\]
\[
R_{of} = 1.876 \text{ m}\Omega
\]

The PSpice results are very close to the hand-calculated values.

**EXAMPLE 10.9**

Finding the performance of a BJT amplifier with shunt-shunt feedback  The parameters of the amplifier in Fig. 10.38 are \(R_{C1} = 5 \text{ k}\Omega\), \(R_E = 2.5 \text{ k}\Omega\), \(R_{C2} = 5 \text{ k}\Omega\), \(R_F = 4 \text{ k}\Omega\), and \(R_s = 200 \text{ }\Omega\). The DC bias currents of the transistors are \(I_{C1} = 0.5 \text{ mA}\) and \(I_{C2} = 1 \text{ mA}\). The transistor parameters are \(h_{fe} = h_{fe1} = 150\) and \(r_o = r_o = \infty\).
(a) Use the techniques of feedback analysis to calculate the input resistance \( R_{if} \), the output resistance \( R_{of} \), and the closed-loop transresistance gain \( A_f \).

(b) Use PSpice/SPICE to check your results.

**NOTE:** The following analysis can also be applied to MOSFET amplifiers by substituting a very large value of the B-E resistance \( r_{\pi} \) tending to infinity, \( 1/r_{\pi} \approx 0 \).

**SOLUTION**

For \( I_{C1} = 0.5 \) mA, Eqs. (8.40) and (8.42) give

\[
r_{\pi1} = \frac{h_{fe} \times 25.8 \text{ mV}}{I_{C1}} = \frac{150 \times 25.8 \text{ mV}}{0.5 \text{ mA}} = 7.74 \text{ k}\Omega
\]

\[
g_{m1} = \frac{I_{C1}}{V_T} = \frac{0.5 \text{ mA}}{25.8 \text{ mV}} = 19.38 \text{ mA/V}
\]

For \( I_{C2} = 1 \) mA,

\[
r_{\pi2} = \frac{h_{fe} \times 25.8 \text{ mV}}{I_{C2}} = \frac{150 \times 25.8 \text{ mV}}{1 \text{ mA}} = 3.87 \text{ k}\Omega
\]

\[
g_{m2} = \frac{I_{C2}}{V_T} = \frac{1 \text{ mA}}{25.8 \text{ mV}} = 38.76 \text{ mA/V}
\]

The feedback current \( i_f \) is proportional to the output voltage \( v_o \). The effective input current to the amplifier is \( i_e = i_i - i_f \).

(a) The steps in analyzing the feedback network are as follows:

**Step 1.** \( R_F \) acts as the feedback network. The feedback current \( i_f \) is proportional to the output voltage \( v_o \). The effective input current to the amplifier is \( i_e = i_i - i_f \). The functional block diagram is shown in Fig. 10.39(a).

**Step 2.** The amplifier uses shunt-shunt feedback, as shown in Fig. 10.39(b). Thus, the units of the gain \( A \) must be V/A.

**Step 3.** The effect of the feedback network can be taken into account at the input side by short-circuiting \( R_F \) to the ground at side 2 in Fig. 10.39(b), and at the output side by short-circuiting \( R_F \) to the ground at side 1. These modifications are shown in the small-signal AC equivalent circuit in Fig. 10.39(c).

**Step 4.** If we represent the amplifier in Fig. 10.39(c) by an equivalent transresistance amplifier, the resistance at the base of the amplifier is

\[
R_b = \frac{v_i}{i_{b1}} = r_{\pi1} + (1 + g_{m1}r_{\pi1})(R_E \| r_{\pi2}) = 7.74 \text{ k} + 151(2.5 \text{ k} \| 3.87 \text{ k}) = 237 \text{ k}\Omega
\]

The input resistance of the amplifier is

\[
R_{ie} = \frac{v_i}{i_s} = R_b \| [R_E \| (r_{\pi1} + (1 + g_{m1}r_{\pi1})(R_E \| r_{\pi2})] = 200 \| 4 \text{ k} \| [7.74 \text{ k} + 151(2.5 \text{ k} \| 3.87 \text{ k})] = 190.3 \text{ \Omega}
\]

The output resistance of the amplifier is

\[
R_{oe} = R_E \| R_{C2} = 4 \text{ k} \| 5 \text{ k} = 2.22 \text{ k}\Omega
\]

Thus,

\[
v_{b1} = R_{ie}i_{s} = 190.3i_{s}
\]
Therefore, the open-loop transresistance $A$ is given by

$$A = \frac{v_o}{i_s} = \frac{15.865 \text{ kA}}{15.865 \text{ kV}} = 447 \Omega$$

**Step 5.** Using Eq. (10.66), we find that the feedback factor $\beta$ is

$$\beta = \frac{i_f}{v_o} \bigg|_{v_f=0} = -\frac{1}{R_F} = -\frac{1}{4 \text{ k}} = -0.25 \text{ mA/V}$$

and

$$\beta A_e = -15.865 \text{ kV/A} \times \left( -\frac{1}{4 \text{ kV/A}} \right) = 3.966$$

**Step 6.** The input resistance seen by the current source is

$$R_i = \frac{v_i}{i_s} = \frac{R_{se}}{1 + \beta A_e} = \frac{190.3 \text{ } \Omega}{1 + 0.25 \text{ mA/V} \times 15.865 \text{ kV/A}} = 38.3 \Omega$$

**Step 7.** The output resistance with feedback is

$$R_{of} = \frac{R_{oe}}{1 + \beta A_e} = \frac{2.22 \text{ k}\Omega}{1 + 0.25 \text{ mA/V} \times 15.865 \text{ kV/A}} = 447 \Omega$$
Step 8. The closed-loop transresistance gain $A_f$ is

$$A_f = \frac{v_o}{i_s} = \frac{-15.865 \text{ kV/A}}{1 + 0.25 \text{ mA/V} \times 15.865 \text{ kV/A}} = -3.195 \text{ kV/A}$$

(b) The shunt-shunt feedback amplifier for PSpice simulation is shown in Fig. 10.40. The results of PSpice/SPICE simulation (.TF analysis) are shown below, with hand calculations to the right:

- $V(5)/I_s = -3.19E+03 = -3.19 \text{ kV/A}$
- $A_f = -3.195 \text{ kV/A}$
- INPUT RESISTANCE AT $I_s = 3.854E+01 = 38.54 \Omega$
- $R_{in} = 38.3 \Omega$
- OUTPUT RESISTANCE AT $V(5) = 4.5E+02 = 450 \Omega$
- $R_{of} = 447 \Omega$

The PSpice results are very close to the hand-calculated values.

**Fig. 10.40** Two-stage shunt-shunt feedback network for PSpice simulation

---

**KEY POINTS OF SECTION 10.8**

- Shunt-shunt feedback is applied to transresistance amplifiers. This type of feedback reduces both the input and the output resistances by a factor of $(1 + \beta \mu)$. 
- The amplifier is represented by a transresistance amplifier and the feedback network as a transconductance gain. $A$ is in units of $V/A$, and $\beta$ is in units of $A/V$.

**10.9 Shunt-Series Feedback**

In shunt-series feedback, as shown in Fig. 10.10(a), the feedback network is in parallel with the amplifier at the input side and in series with the amplifier at the output side. The amplifier is represented as a current amplifier. This is shown in Fig. 10.41, in which the amplifier has an input resistance of $R_i$, an output resistance of $R_o$, and an open-loop current gain of $A$ (in $A/A$); that is, $A = \mu g R_i / R_o$. 
The feedback current $i_f$ is proportional to the output current $i_o$. The feedback network is modeled in gain form, with an input resistance of $R_y$, an output resistance of $R_x$, and a current gain of $\beta$. The test conditions for determining the parameters of the feedback network are shown in Fig. 10.42. The parameters of the model are defined by three equations. The first equation is

$$R_x = \frac{v_f}{i_f}_{|_{i_y=0}} \quad \text{(open-circuited output side)} \quad (10.87)$$

which is obtained by applying a test voltage of $v_f$ at side 1 and open-circuiting side 2. The second equation is

$$R_y = \frac{v_o}{i_y}_{|_{v_f=0}} \quad \text{(short-circuited input side)} \quad (10.88)$$

which is obtained by applying a test voltage of $v_y$ at side 2 and short-circuiting side 1. The third equation is

$$\beta = \frac{i_f}{i_y}_{|_{v_f=0}} \quad \text{(short-circuited input side)} \quad (10.89)$$

which is obtained by applying a test voltage of $v_y$ at side 2 and short-circuiting side 1.

---

**FIGURE 10.41** Shunt-series feedback amplifier

The feedback network is modeled in gain form, with an input resistance of $R_y$, an output resistance of $R_x$, and a current gain of $\beta$. The test conditions for determining the parameters of the feedback network are shown in Fig. 10.42. The parameters of the model are defined by three equations. The first equation is

$$R_x = \frac{v_f}{i_f}_{|_{i_y=0}} \quad \text{(open-circuited output side)} \quad (10.87)$$

which is obtained by applying a test voltage of $v_f$ at side 1 and open-circuiting side 2. The second equation is

$$R_y = \frac{v_o}{i_y}_{|_{v_f=0}} \quad \text{(short-circuited input side)} \quad (10.88)$$

which is obtained by applying a test voltage of $v_y$ at side 2 and short-circuiting side 1. The third equation is

$$\beta = \frac{i_f}{i_y}_{|_{v_f=0}} \quad \text{(short-circuited input side)} \quad (10.89)$$

which is obtained by applying a test voltage of $v_y$ at side 2 and short-circuiting side 1.

---

**FIGURE 10.42** Test conditions for determining the parameters of a shunt-series feedback circuit
10.9.1 Analysis of an Ideal Shunt-Series Feedback Network

Let us assume an ideal feedback network—that is, \( R_x = \infty, R_y = 0, \) and \( R_L = 0 \). The feedback amplifier in Fig. 10.41 can be simplified to the one in Fig. 10.43. The feedback factor \( \beta \) is in A/A.

If \( R_L \ll R_o \), it can be shown that

\[
R_{if} = \frac{v_i}{i_i} = \frac{R_i}{1 + \beta A} \tag{10.90}
\]

\[
R_{of} = R_o (1 + \beta A) \tag{10.91}
\]

\[
A_f = \frac{i_o}{i_i} = \frac{A}{1 + \beta A} \tag{10.92}
\]

NOTES
1. \( A \) is the open-loop current gain of the amplifier, in A/A.
2. \( \beta \) is the current gain of the feedback network, in A/A.
3. \( T_L = \beta A \) is the loop gain, which is dimensionless.

10.9.2 Analysis of a Practical Shunt-Series Feedback Network

The open-loop parameters of the amplifier in Fig. 10.44(a) can be modified to include the loading effect of \( R_v, R_x, R_y, \) and \( R_L \), producing the equivalent ideal feedback network shown in Fig. 10.44(b). The modified parameters are given by

\[
R_{ie} = R_i || R_x \tag{10.93}
\]

and

\[
R_{oe} = R_o + R_y + R_L \tag{10.94}
\]
It may appear from Fig. 10.44(b) that \( R_o \) would have to be in parallel with \( (R_y + R_L) \) for us to find \( R_{oe} \) in Eq. (10.94), but if we convert the current source \( A_i \) to a voltage source, we see that the effective resistance used to find the current \( i_o \) becomes \( (R_o / H11001 R_y / H11001 R_L) \): 

\[
(10.95)
\]

\[
(10.96)
\]

From Eqs. (10.95) and (10.96), we can derive the modified open-loop current gain \( A_e \): 

\[
A_e = \frac{i_o}{i_{e1}} = \frac{R_o R_x}{(R_y + R_L + R_o)(R_1 + R_x)} A
\]

If the values of \( R_i \), \( R_o \), and \( A \) are replaced by \( R_{ie}, R_{oe}, \) and \( A_e \), respectively, Eqs. (10.90) through (10.92) can be applied to calculate the closed-loop parameters \( R_{if}, R_{of}, \) and \( A_f \).

**EXAMPLE 10.10**

**Finding the performance of an inverting amplifier with shunt-series feedback**  The parameters of the shunt-series feedback amplifier shown in Fig. 10.10(b) are \( R_L = 5 \, \Omega, \, R_s = 2.5 \, k\Omega, \, R_F = 200 \, \Omega, \) and \( R_1 = 5 \, \Omega \). The op-amp parameters are \( R_i = 2 \, M\Omega, \, R_o = 75 \, \Omega, \) and \( \mu_g = 2 \times 10^5 \).

(a) Determine the input resistance at the op-amp input \( R_{if} = v_i / i_i \), the output resistance \( R_{of} \), and the closed-loop current gain \( A_f = i_o / i_i \).

(b) Use PSpice/SPICE to check your results.

**SOLUTION**

\( R_s = 2.5 \, k\Omega, \, R_F = 200 \, \Omega, \, R_1 = 5 \, \Omega, \, R_L = 5 \, \Omega, \, R_i = 2 \, M\Omega, \, R_o = 75 \, \Omega, \) and \( \mu_g = 2 \times 10^5 \). Replacing the op-amp by its equivalent circuit gives the amplifier shown in Fig. 10.45(a).
Converting the voltage-controlled voltage source to a current-controlled current source, we get

\[ i_o = \frac{\mu F v_i}{R_o} = \frac{\mu F R_i e}{R_o} = A i_e \]

which gives

\[ A = \frac{\mu F R_i}{R_o} = \frac{2 \times 10^5 \times 2 \times 10^6}{75} = 53.33 \times 10^8 \ A/A \]

The effect of the feedback network is taken into account by severing \( R_1 \) from the load at side 2 and shorting \( R_F \) to the ground at side 1, as shown in Fig. 10.45(b). Then

\[
egin{align*}
R_x &= R_F + R_1 = 200 + 5 = 205 \ \Omega \\
R_y &= R_F \parallel R_1 = 200 \parallel 5 = 4.88 \ \Omega \\
R_{ie} &= R_i \parallel (R_F + R_1) = R_i \parallel R_x = 2 \ M \parallel 205 = 204.98 \ \Omega \\
R_{oe} &= R_o + R_y + R_L = R_o + (R_F \parallel R_1) + R_L = 75 + 4.88 + 5 = 84.88 \ \Omega 
\end{align*}
\]

Equation (10.89) gives the feedback factor \( \beta \):

\[ \beta = \left. \frac{i_f}{i_y} \right|_{v_f=0} = \frac{R_i}{R_F + R_1} = \frac{5}{200 + 5} = 24.39 \times 10^{-3} \ A/A \]

Equation (10.97) gives the modified open-loop current gain \( A_e \):

\[ A_e = \frac{75 \times 205 \times 53.33 \times 10^8}{(4.88 + 5 + 75)(2 \ M + 205)} = 483 \times 10^3 \ A/A \]
The loop gain is

\[ T_L = \beta A_e = 24.39 \times 10^{-3} \times 483 \times 10^3 = 11.76 \times 10^3 \]

The resistances are

\[ R_{if} = \frac{R_{ie}}{1 + T_L} = \frac{204.98}{1 + 11.76 \times 10^3} = 17.4 \text{ m}\Omega \]

\[ R_{in} = \frac{V_s}{i_s} = R_{if} + R_s = 17.4 \text{ m} + 2.5 \text{ k} \approx 2.5 \text{ k}\Omega \]

\[ R_{of} = R_{oe} \times (1 + T_L) = 84.88 \times (1 + 11.76 \times 10^3) = 999.6 \text{ k}\Omega \]

Then

\[ A_f = \frac{i_o}{i_i} = \frac{483 \times 10^3}{1 + 11.763 \times 10^3} \approx 41 \]

(b) The shunt-series feedback circuit for PSpice simulation is shown in Fig. 10.46. The results of PSpice simulation (.TF analysis) are shown below, with hand calculations to the right:

\[ \frac{I(VX)}{I_s} = 4.100E+01 \approx 41 \quad A_f = 41 \]

\[ \text{INPUT RESISTANCE AT } I_s = 1.740E-02 = 17.4 \text{ m}\Omega \quad R_{if} = 17.4 \text{ m}\Omega \]

\[ \text{OUTPUT RESISTANCE AT } I(VX) = 1.000E+06 = 1 \text{ M}\Omega \quad R_{of} = 999.6 \text{ k}\Omega \]

The PSpice results are very close to the hand-calculated values.

**KEY POINTS OF SECTION 10.9**

- Shunt-series feedback is applied to current amplifiers. This type of feedback reduces the input resistance and increases the output resistance by a factor of \((1 + \beta A)\).
- The amplifier is represented by a current amplifier and the feedback network as a current gain. Both \(A\) and \(\beta\) are in units of \(A/A\).
10.10 Feedback Circuit Design

The feedback factor $\beta$ is the key parameter of a feedback amplifier and modifies its closed-loop gain $A_f$. The type of feedback used depends on the requirements for input resistance $R_{if}$ and output resistance $R_{of}$. If $A$ is independent of $\beta$, designing a feedback amplifier requires finding the value of $\beta$ that will yield the desired value of $A_f$, $R_{if}$, $R_{of}$, or bandwidth BW. The design becomes cumbersome if $A$ depends on the feedback network—that is, on $\beta$. The following iterative steps, however, will simplify the design process:

**Step 1.** Decide on the type of feedback needed to meet the specifications. Use Table 10.1 as a guide.

**Step 2.** Disconnect the feedback link; that is, make sure no feedback is present.

**Step 3.** Find the approximate open-loop parameters $A$, $R_i$, and $R_o$ of the amplifier.

**Step 4.** Find the values of feedback factor $\beta$ and feedback resistance(s) that will satisfy the closed-loop requirement. Use the relations in Table 10.1.

**Step 5.** Using the feedback resistance(s), recalculate the open-loop parameters $A$, $R_i$, and $R_o$.

**Step 6.** Find the closed-loop parameters $A_f$, $R_{if}$, and $R_{of}$.

**Step 7.** Repeat steps 3 through 5 until the desired closed-loop condition is satisfied. Normally, a number of iterations will be required.

**EXAMPLE 10.11**

**Designing a series-shunt feedback circuit** Feedback is applied to a voltage amplifier whose open-loop parameters are $R_i = 4.5 \, \text{k}\Omega$, $R_o = 500 \, \Omega$, low-frequency voltage gain $A = -450 \, \text{V/V}$, and bandwidth $BW = f_H = 10 \, \text{kHz}$. The load resistance is $R_L = 10 \, \text{k}\Omega$. Determine the values of the feedback network so that the following specifications are satisfied:

(a) Bandwidth with feedback $f_{HF} = 1 \, \text{MHz}$, $R_{if} > R_i$, and $R_{of} < R_o$

(b) $R_{if} = 50R_i$ and $R_{of} < R_o$

(c) $R_{if} > R_i$ and $R_{of} = R_o/250$

**SOLUTION**

$R_i = 4.5 \, \text{k}\Omega$, $R_o = 500 \, \Omega$, $A = -450 \, \text{V/V}$, and $f_H = 10 \, \text{kHz}$.

(a) Since the input resistance $R_{if}$ should increase and the output resistance $R_{of}$ should decrease, we can see from Table 10.1 that the feedback must be of the series-shunt type. Thus, we can use the equations derived in Sec. 10.6. The feedback network consists of $R_1$ and $R_F$, as shown in Fig. 10.47. Since $A$ is negative, the value of $v_f$ will be negative. Thus, $v_f$ is added to $v_s$ rather than subtracted from it (i.e., $v_s + v_f = v_e$), and this is accomplished by connecting the feedback signal to the negative terminal of $v_s$. To minimize the loading effects, $(R_1 + R_F)$ must be much larger than $R_L$. The condition

$$ (R_1 + R_F) >> R_L \quad (10.98) $$

is generally satisfied by choosing $R_1 + R_F = 10R_L$. Since the gain–bandwidth product remains constant, Eq. (10.24) gives $Af_{HF} = A_f f_{HF}$. That is,

$$ 450 \times 10 \, \text{kHz} = A_f \times 1 \, \text{MHz} $$

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
which gives $A_f = 450/100 = 4.5 \text{ V/V}$. From Eq. (10.37),

$$4.5 = \frac{450}{1 + 450\beta}$$

which gives $\beta = 0.22$. The feedback factor $\beta$ is related to $R_1$ and $R_F$ by

$$\beta = \frac{R_F}{R_1 + R_F} = \frac{R_F}{10R_L} \quad (10.99)$$

which, for $\beta = 0.22$ and $R_L = 10 \text{ k} \Omega$, gives

$$R_F = 10\beta R_L = 10 \times 0.22 \times 10 \text{ k} \Omega = 22 \text{ k} \Omega$$

and

$$R_1 = 10R_L - R_F = 100 \text{ k} - 22 \text{ k} = 78 \text{ k} \Omega$$

(b) Since $R_{if} = 50R_i$ and $R_{if} = R_i(1 + \beta A)$ from Eq. (10.40), we get

$$50 = 1 + \beta A \quad (\text{for negative feedback})$$

which, for $|A| = 450$, gives $\beta = 0.109$. For $\beta = 0.109$ and $R_L = 10 \text{ k} \Omega$, Eq. (10.99) gives

$$R_F = 10\beta R_L = 10 \times 0.109 \times 10 \text{ k} \Omega \approx 11 \text{ k} \Omega$$

and

$$R_1 = 10R_L - R_F = 100 \text{ k} - 11 \text{ k} = 89 \text{ k} \Omega$$

(c) Since $R_{of} = R_o/250$ and $R_{of} = R_o/(1 + \beta A)$ from Eq. (10.44), we get

$$250 = 1 + \beta A$$

which, for $|A| = 450$, gives $\beta = 0.5533$. For $\beta = 0.5533$ and $R_L = 10 \text{ k} \Omega$, Eq. (10.99) gives

$$R_F = 10\beta R_L = 10 \times 0.5533 \times 10 \text{ k} \Omega \approx 55 \text{ k} \Omega$$

and

$$R_1 = 10R_L - R_F = 100 \text{ k} - 55 \text{ k} = 45 \text{ k} \Omega$$

**NOTE:** The above solution gives approximate values for the designer to start with. More accurate values of $A_f$, $R_{if}$, and $R_{of}$ could be found by considering the loading effects of the feedback network and the load resistance (as in Example 10.4). The design steps should be repeated until the desired specifications are met.
EXAMPLE 10.12

**Designing a shunt-series feedback circuit**  An amplifier with shunt-series feedback is shown in Fig. 10.48. The amplifier has $R_1 = 6.6 \, \text{k}\Omega$, $R_2 = 2 \, \text{k}\Omega$, $R_{C1} = 5 \, \text{k}\Omega$, $R_3 = 5 \, \text{k}\Omega$, $R_4 = 10 \, \text{k}\Omega$, $R_E = 500 \, \text{Î}2$, and $R_{C2} = 5 \, \text{k}\Omega$. The transistor parameters are $h_{fe} = 150$, $r_{\pi} = r_{\pi1} = r_{\pi2} = 2.58 \, \text{k}\Omega$, and $r_o = \infty$.

(a) Determine the value of feedback resistor $R_F$ so that the closed-loop current gain $A_f$ is 10% of the open-loop current gain $A$.

(b) Use PSpice/SPICE to check your results.

NOTE: The following analysis can also be applied to MOSFET amplifiers by substituting a very large value of the B-E resistance $r_{\pi}$, tending to infinity, $1/r_{\pi} \approx 0$.

**SOLUTION**

The feedback current $i_f$ is proportional to the emitter voltage $v_e = v_F$, which in turn is proportional to the output current $i_o = -i_C$. The feedback mechanism is shown in Fig. 10.49(a) and the feedback network in Fig. 10.49(b). Replacing the transistors by their small-signal model gives the small-signal AC equivalent circuit of the amplifier shown in Fig. 10.49(c),

(a) We have

\[
g_m = \frac{h_{fe}}{r_{\pi}} = \frac{150}{2.58 \, \text{k}\Omega} = 5.814 \, \text{mA/V}
\]

\[
R_{B1} = R_1 \parallel R_2 = 6.6 \, \text{k}\Omega \parallel 2 \, \text{k}\Omega = 1.53 \, \text{k}\Omega
\]

\[
R_{B2} = R_3 \parallel R_4 \parallel R_{C1} = 5 \, \text{k}\Omega \parallel 10 \, \text{k}\Omega \parallel 5 \, \text{k}\Omega = 2 \, \text{k}\Omega
\]

Step 1. Assume that there is no feedback—that is, $R_F = \infty$.

Step 2. Find the open-loop parameters $A$, $R_i$, and $R_o$. The input resistance of the amplifier is

\[
R_i = \frac{v_i}{i_s} = R_{B1} \parallel r_{\pi1} = 1.53 \, \text{k}\Omega \parallel 2.58 \, \text{k}\Omega = 962.4 \, \text{Î}2
\]

The output resistance of the amplifier is

\[
R_o = r_o = \infty \quad \text{and} \quad R_{\text{oe}} = R_o
\]

**FIGURE 10.48** Two-stage amplifier with shunt-series feedback
From Fig. 10.49(c), we get

Therefore, the open-loop current gain \( A \) is given by

\[
\frac{A}{H11005} = \frac{i_o}{i_s}/H11002 = 209.2 \text{A}.
\]

**Step 3.** Find the values of feedback factor \( \frac{A}{H9252} \) and resistance \( R_F \). Since the closed-loop current gain \( A_f \) is 10\% of the open-loop current gain \( A \), we can write

That is,

\[
1 + \beta |A| = 10 \quad \text{or} \quad \beta |A| = 9 \quad \text{and} \quad \beta = \frac{9}{|A|} = \frac{9}{209.2} = 0.043
\]

Since the feedback network in Fig. 10.49(b) must be represented by a current amplifier, the feedback factor \( \beta \) is given by

\[
\beta = \left. \frac{i_f}{i_y} \right|_{V_f=0} = \frac{R_F}{R_E + R_F} = \frac{500}{500 + R_F}
\]

which, for \( \beta = 0.043 \), gives \( R_F = 11.13 \text{k}\Omega \).
Step 4. Find the new open-loop parameters $A$, $R_i$, and $R_o$. $R_F$ is included by short-circuiting the shunt feedback side and severing the series feedback side. This arrangement is shown in Fig. 10.49(c) by the gray lines. The input resistance of the amplifier is

$$R_i = \frac{V_i}{I_s} = (R_F + R_E) \parallel R_{B1} \parallel R_{m1} = (11.13 \text{ k} + 500) \parallel 1.53 \text{ k} \parallel 2.58 \text{ k} = 887.2 \Omega$$

The output resistance of the amplifier is

$$R_o = r_o = \infty \quad \text{and} \quad R_{oe} = R_o = \infty$$

From Fig. 10.49(c), we get

$$v_{b1} = v_{be1} = R_i I_s = 887.2 I_s$$
$$v_{b2} = -\{ R_{B2} \parallel [r_{m2} + (1 + g_m2r_{m2})(R_E \parallel R_F)] \} g_m v_{be1}$$
$$= -\{ 2 \text{ k} \parallel [2.58 \text{ k} + (1 + 58.14 \text{ m} \times 2.58 \text{ k}) \times (11.13 \text{ k} \parallel 500)] \} \times 58.14 \text{ m} \times (887.2 I_s)$$
$$= -100.7 \times 10^3 I_s$$
$$v_{be2} = \frac{v_{b2} \times r_{m2}}{[r_{m2} + (1 + g_m2r_{m2})(R_E \parallel R_F)]}$$
$$= \frac{(-100.7 \times 10^3 I_s) \times 2.58 \text{ k}}{[2.58 \text{ k} + (1 + 58.14 \text{ m} \times 2.58 \text{ k}) \times (11.13 \text{ k} \parallel 500)]} = -3.471 \times 10^3 I_s$$
$$I_o = g_m v_{be2} = 58.14 \text{ m} \times (-3.471 \times 10^3 I_s) = -201.4 I_s$$

Therefore, the open-loop current gain $A$ is given by $A = I_o/I_s = -201.4 \text{ A/A}$.

The closed-loop parameters are as follows:

$$R_{cf} = \frac{R_i}{1 + \beta A} = \frac{887.2}{1 + 0.043 \text{ A/A} \times 201.4 \text{ A/A}} = 91.8 \Omega$$

The feedback will not affect the output resistance; that is, $R_{cf} = \infty$ and

$$A_f = \frac{201.4}{1 + 0.043 \text{ A/A} \times 201.4 \text{ A/A}} = 20.85 \text{ A/A}$$

Step 5. Repeating steps 3 to 5 for the second iteration with $A = 201.4 \text{ A/A}$, we get the following values:

$$\beta = \frac{9}{|A|} = \frac{9}{201.4} = 0.0447$$
$$R_F = 10.69 \text{ k}\Omega$$
$$R_i = \frac{V_s}{I_s} = (R_F + R_E) \parallel R_{B1} \parallel R_{m1} = (10.69 \text{ k} + 500) \parallel 1.53 \text{ k} \parallel 2.58 \text{ k} = 886.2 \Omega$$
$$R_o = r_o = \infty$$
$$R_{oe} = R_o = \infty$$
$$v_{b1} = v_{be1} = R_i I_s = 886.2 I_s$$
$$v_{b2} = -\{ R_{B2} \parallel [r_{m2} + (1 + g_m2r_{m2})(R_E \parallel R_F)] \} g_m v_{be1}$$
$$= -\{ 2 \text{ k} \parallel [2.58 \text{ k} + (1 + 58.14 \text{ m} \times 2.58 \text{ k}) \times (11.13 \text{ k} \parallel 500)] \} \times 58.14 \text{ m} \times (886.2 I_s)$$
$$= -100.4 \times 10^3 I_s$$
Therefore, the open-loop current gain $A$ is given by $A = \frac{i_o}{i_s} = 201.1 \text{ A/A}$.  

The output resistance with feedback is

\[ R_{df} = \frac{R_i}{1 + \beta A} = \frac{886.2}{1 + 0.0447 \text{ A/A} \times 201.1 \text{ A/A}} = 88.5 \Omega \]

The output resistance with feedback is

\[ R_{df} = R_o(1 + \beta A) = \infty \]

Since the output is taken across $R_C$, the output resistance of the amplifier is

\[ R_{out} = R_{df} || R_C = R_C = 5 \text{ k}\Omega \]

and

\[ A_f = \frac{201.1}{1 + 0.0447 \text{ A/A} \times 201.1 \text{ A/A}} = 20.13 \text{ A/A} \]

which is 10% of $A = 201.1$. Thus, there is no need for further iterations.

(b) The shunt-series feedback amplifier for PSpice simulation is shown in Fig. 10.50. The results of PSpice simulation (.TF analysis) are shown below, with hand calculations to the right:

\[
\frac{I(VZ)}{I_S} = 2.002 \times 10^3 = 20.02 \quad A_f = 20.13 \text{ A/A} \\
\text{INPUT RESISTANCE AT } I_S = 8.803 \times 10^3 = 88.03 \Omega \quad R_{df} = 88.5 \Omega \\
\text{OUTPUT RESISTANCE AT } I(VZ) = 7.991 \times 10^3 = 79.91 \text{ T}\Omega \quad R_{df} = \infty
\]

**FIGURE 10.50** Two-stage shunt-series feedback amplifier for PSpice simulation

**NOTE:** The output current is measured through a dummy voltage source $V_Z$ connected across $R_C$. If $V_Z$ were connected in series with $R_C$, PSpice would give a very large output resistance as a result of the ideal current source of transistor $Q_2$. 
10.11 Stability Analysis

Negative feedback modifies the gain, the input resistance, and the output resistance of an amplifier. It also improves the performance parameters; for example, it reduces both the sensitivity of the gain to amplifier parameter changes and the distortion due to nonlinearities. However, negative feedback may become positive, thereby causing oscillation and instability. So far in this chapter we have assumed that the feedback network is resistive and the feedback factor $\beta$ remains constant. But $\beta$ can depend on frequency. In such cases, the closed-loop transfer function of a negative feedback circuit in Laplace’s domain of $s$ is given by

$$A_f(s) = \frac{S_o(s)}{S_i(s)} = \frac{A(s)}{1 + A(s)\beta(s)} \quad (10.100)$$

10.11.1 Closed-Loop Frequency and Stability

The open-loop gain $A(s)$ and the feedback factor $\beta(s)$ are dependent on the frequency. For physical systems, $s = j\omega$, and Eq. (10.100) can be written in the frequency domain as follows:

$$A_f(j\omega) = \frac{A(j\omega)}{1 + A(j\omega)\beta(j\omega)} \quad (10.101)$$

The loop gain $T_L(j\omega) = A(j\omega)\beta(j\omega)$ is a complex number that can be represented by its magnitude and phase, as follows:

$$T_L(j\omega) = A(j\omega)\beta(j\omega) = |A(j\omega)\beta(j\omega)|e^{j\phi(\omega)} = |T_L(j\omega)| \angle \phi \quad (10.102)$$

Whether a feedback amplifier is stable or unstable depends on the magnitude and phase of the loop gain $T_L(j\omega)$. Consider the frequency $\omega_{180}$ at which the phase angle $\phi(\omega) = \pm 180^\circ$, so $T_L(j\omega) = -|T_L(j\omega)|$. Equation (10.101) becomes

$$A_f(j\omega_{180}) = \frac{A(j\omega_{180})}{1 - |T_L(j\omega_{180})|} \quad (10.103)$$

and the feedback becomes positive. However, the value of $A_f(j\omega_{180})$ will depend on the following conditions:

1. If $|T_L(j\omega_{180})| < 1$, the denominator $(1 - |T_L(j\omega_{180})|)$ will be less than unity; that is, $A_f(j\omega_{180}) > |A(j\omega_{180})|$. In this case, the feedback system will be stable.
2. If $|T_L(j\omega_{180})| = 1$, the denominator $(1 - |T_L(j\omega_{180})|)$ will be zero, and $A_f(j\omega_{180})$ will be infinite; that is, the amplifier will have an output with zero input voltage, and the loop will oscillate without

---

**KEY POINTS OF SECTION 10.10**

- Designing feedback amplifiers requires determining the type of feedback and the feedback network.
- It is also necessary to find the component values of the network. A designer normally begins by finding the value of feedback factor $\beta$ that gives the desired closed-loop gain, with the assumption of an ideal feedback network. Once initial estimates of the component values have been made, the normal analysis is performed to verify the closed-loop gain. Normally, several iterations are required to come to the final solution.
any external input signal. In this case, the feedback system will be an oscillator. Let us assume that there is no input signal in Fig. 10.3; that is, \( S_i = 0 \). We get

\[
S_f = A(j\omega_{180})\beta(j\omega_{180})S_e = |T_L(j\omega_{180})|S_e = -S_e \tag{10.104}
\]

Since \( S_f \) is multiplied by \(-1\) in the summer block at the input side, the feedback causes the signal \( S_e \) at the amplifier input to be sustained. Thus, there will be sinusoidal signals of frequency \( \omega_{180} \) at the input and output of the amplifier. Under these conditions, the amplifier is said to oscillate at the frequency \( \omega_{180} \).

3. If \( |T_L(j\omega_{180})| > 1 \), the denominator \( (1 - |T_L(j\omega_{180})|) \) will be greater than unity; that is, \( |A_f(j\omega_{180})| < |A(j\omega_{180})| \). In this case, the feedback system will be unstable. We have

\[
S_f = A(j\omega_{180})\beta(j\omega_{180})S_e = |T_L(j\omega_{180})|S_e \tag{10.105}
\]

Since \( S_f \) is multiplied by \(-1\) in the summer block at the input side, the amplifier oscillates, and the oscillations grow in amplitude until some nonlinearity reduces the magnitude of the loop gain \( |T_L(j\omega_{180})| \) to exactly unity. In practical amplifiers, nonlinearity is always present in some form, and sustained oscillations will be obtained. This type of feedback condition is employed in oscillator circuits. Oscillators use positive feedback with a loop gain greater than unity; nonlinearity reduces the loop gain to unity.

We have seen that oscillations can occur in a negative feedback amplifier, depending on the frequency. The stability of an amplifier is determined by its response to an input or a disturbance. An amplifier is said to be absolutely stable if the output eventually settles down after a small disturbance. An amplifier is absolutely unstable if a small disturbance causes the output to build up (i.e., to increase continuously) until it reaches the saturation limits of the amplifier. The stability of an amplifier depends on its poles.

### 10.11.2 Poles and Instability

In the above analysis, the denominator of Eq. (10.100) was the key parameter in characterizing the response of an amplifier. The closed-loop gain \( A_f(s) \) of an amplifier will be infinite if

\[
1 + A(s)\beta(s) = 0 \tag{10.106}
\]

Equation (10.106) is called the characteristic equation, and its roots determine the response of the amplifier. For example, if

\[
1 + A(s)\beta(s) = s^2 + 2s + 5 = 0
\]

the roots are \( s = -1 \pm j2 \). The roots of the characteristic equation are the poles of a system.

### 10.11.3 Transient Response and Stability

Consider an amplifier with a pole pair at \( s = \sigma \pm j\omega_p \). If there is any disturbance (such as the closing of the DC power-supply switch), the equation of the transient response (after conversion from Laplace’s \( s \) domain to the time domain) will contain terms of the following form:

\[
v_o(t) = e^{\sigma t}[e^{j\omega_p t} + e^{-j\omega_p t}] = 2e^{\sigma t}\cos(\omega_p t)
\]
This is a sinusoidal output with an envelope of \( \exp (\sigma_0 t) \). Thus, the relationship between stability and the roots of this characteristic equation may be stated as follows:

1. If the poles are in the left half of the \( s \)-plane, as shown in Fig. 10.51(a), such that the denominator is of the form \((s - \sigma_0 - j\omega_n)(s - \sigma_0 + j\omega_n)\), all roots of the characteristic equation will have negative real parts, and \( \sigma_0 \) will be negative. Thus, the response due to initial conditions or disturbances will decay exponentially to zero as time approaches infinity. The plot of such a response is also shown in Fig. 10.51(a). Such a system will be stable.

2. If the poles are in the right half of the \( s \)-plane, as shown in Fig. 10.51(b), such that the denominator is of the form \((s + \sigma_0 - j\omega_n)(s + \sigma_0 + j\omega_n)\), all roots will have positive real parts, and \( \sigma_0 \) will be positive. The response will increase exponentially in magnitude as time increases until some nonlinearity limits its growth. The plot of such a response is also shown in Fig. 10.51(b). Such an amplifier will be unstable.

3. If the poles are on the \( j\omega \)-axis, as shown in Fig. 10.51(c), such that the denominator is of the form \((s + j\omega_n)(s - j\omega_n)\), the characteristic equation will not have roots with real parts, and \( \sigma_0 \) will be zero. The response will be sustained oscillations, as shown in the plot in Fig. 10.51(c). The output will be sinusoidal in response to an initial condition or a disturbance. For a feedback amplifier, this will mean instability. However, for an oscillator circuit, this will mean the normal output.

### 10.11.4 Closed-Loop Poles and Stability

The location of the poles of a closed-loop system on a complex plane determines the transient response of a system. If the poles lie in the left half plane, all time-domain exponential terms decay to zero. If the poles
lie in the right half plane, the system is likely to oscillate because its time-domain response will be growing exponentially. The plot of the poles as the loop gain varies on a complex plane, is called the root locus and it indicates how close the systems are to oscillate the system. Let us take an amplifier with one pole of the open-loop gain relation as given by

\[ A(s) = \frac{A_o}{1 + s/\omega_o} \]  

(10.107)

Therefore, the closed-loop gain becomes

\[ A_f(s) = \frac{A(s)}{1 + \beta A(s)} = \frac{A_o}{1 + s/\omega_o(1 + \beta A_o)} \]  

(10.108)

This gives the closed-loop pole \( \omega_p = -\omega_o(1 + \beta A_o) \), which has a real-valued pole in the left half plane. The pole as shown in Fig. 10.52(a) moves away from the origin as the loop gain increases.

Let us consider an open-loop gain with two poles as given by

\[ A(s) = \frac{A_o}{(1 + s/\omega_{p1})(1 + s/\omega_{p2})} \]  

(10.109)

This gives the closed-loop gain as

\[ A_f(s) = \frac{A(s)}{1 + \beta A(s)} = \frac{A_o}{\omega_{p1}\omega_{p2}} \frac{1}{(1 + s/\omega_{p1})(1 + s/\omega_{p2}) + \beta A_o} \]

\[ = \frac{A_o\omega_{p1}\omega_{p2}}{s^2 + (\omega_{p1} + \omega_{p2})s + (1 + \beta A_o)\omega_{p1}\omega_{p2}} \]  

(10.110)

This can be solved for the closed-loop two poles as given by

\[ s_1, s_2 = \frac{-(\omega_{p1} + \omega_{p2}) \pm \sqrt{(\omega_{p1} + \omega_{p2})^2 - 4(1 + \beta A_o)\omega_{p1}\omega_{p2}}}{2} \]  

(10.111)

for \( \beta = 0, s_1 = -\omega_{p1}, \) and \( s_2 = -\omega_{p2}. \) As \( \beta \) increases, the poles move toward each other and become equal at \( \beta = \beta_1. \) For \( \beta > \beta_1, \) they become complex. The pole plot is shown in Fig. 10.52(b).

**FIGURE 10.52** Closed-loop pole plots
EXAMPLE 10.13

Finding the stability due to a step input  The open-loop gain of an amplifier is given by

\[ A(s) = \frac{s}{s^2 - 2} \]

Determine the stability of the closed-loop response due to a step input signal. Assume feedback factor \( \beta(s) = 1 \).

**SOLUTION**

For a step input in Laplace’s domain of \( s \), \( S_i(s) = \frac{1}{s} \). From Eq. (10.5), the closed-loop gain is

\[ A_f(s) = \frac{S_o(s)}{S_i(s)} = \frac{s/(s^2 - 2)}{s^2 + s - 2} = \frac{s}{(s - 1)(s + 2)} \]

For a step input, \( S_i(s) = \frac{1}{s} \) and the output response is given by

\[ S_o(s) = A_f(s)S_i(s) = \frac{1}{s} \left( \frac{1}{s - 1} - \frac{1}{s + 2} \right) \]

Thus, in the time domain, the output response due to a step input becomes

\[ s_o(t) = \frac{1}{3}(e^t - e^{-2t}) \]

Therefore, at \( t = \infty \), \( s_o(t) = \infty \). The amplifier will be unstable because it has a positive pole, \( s = 1 \).

EXAMPLE 10.14

Finding the step response of a feedback amplifier  The open-loop gain of an amplifier is given by

\[ A(s) = \frac{1}{s^2 + 3} \]

Determine the closed-loop step response of the amplifier. Assume feedback factor \( \beta(s) = 1 \).

**SOLUTION**

For a step input in Laplace’s domain of \( s \), \( S_i(s) = \frac{1}{s} \). From Eq. (10.5), the closed-loop gain is

\[ A_f(s) = \frac{S_o(s)}{S_i(s)} = \frac{1/(s^2 + 3)}{1 + 1/(s^2 + 3)} = \frac{1}{s^2 + 4} \]

For a step input, the closed-loop response is

\[ S_o(s) = A_f(s)S_i(s) = \frac{1}{(s^2 + 4)s} \left( \frac{1}{s} - \frac{s}{s^2 + 4} \right) \]

Therefore, in the time domain, the output response due to a step input is given by

\[ s_o(t) = \frac{1}{4}(1 - \cos 2t) \]

Therefore, the response due to a step input is oscillatory. The amplifier will be unstable because it has poles on the \( j\omega \)-axis, \( s = \pm j2 \).
10.11.5 Nyquist Stability Criterion

If the loop gain $T_L(j\omega) = A(j\omega)B(j\omega)$ in Eq. (10.102) becomes $-1$, the closed-loop gain will tend to be infinite, and the amplifier will be unstable. Therefore, a feedback amplifier will be unstable if the loop gain $T_L(j\omega)$ becomes

$$T_L(j\omega) = A(j\omega)B(j\omega) = -1 \quad (10.112)$$

To satisfy the condition of Eq. (10.112), the magnitude and phase angle of the loop gain must be unity and $\pm 180^\circ$, respectively; that is,

$$|T_L(j\omega)| = 1 \quad (10.113)$$

$$\phi = \angle T_L(j\omega) = \pm 180^\circ \quad (10.114)$$

Therefore, investigating the stability of an amplifier requires determining the frequency response and characterizing the magnitude and phase angle against the frequency. The frequency response can be represented by a series of phasors, each at a different frequency. Joining the extremities (i.e., the magnitude points) of these phasors gives the frequency locus, as shown in Fig. 10.53, which is a plot of magnitude versus phase in polar coordinates as the frequency $\omega$ is varied from 0 to $\pm \infty$. This plot is known as a Nyquist plot. The $T_L(j\omega)$ plot for negative frequencies is the mirror image through the Re-axis (real axis) of the plot for positive frequencies. The Nyquist plot intersects the negative Re-axis at the frequency $\omega_{180}$. If the intersection occurs to the left of the point $(-1, 0)$, the magnitude of the loop gain at this frequency is greater than unity and the amplifier will be unstable. On the other hand, if the intersection occurs to the right of the point $(-1, 0)$, the amplifier will be stable. The point on the locus where the phase is $-180^\circ$ is called the phase crossover frequency $\omega_p$; the point on the locus where the gain is unity is called the gain crossover frequency $\omega_g$.

The stability of a system can be determined from the Nyquist criterion for stability, which can be stated as follows: If the Nyquist plot encircles the point $(-1, 0)$, the amplifier is unstable. This criterion tests for the poles of loop gain $T_L(s)$ in the right half plane [2]. If the Nyquist plot encircles the point $(-1, 0)$, the amplifier has poles in the right half plane, and the circuit will oscillate. The number of times the plot encircles the point $(-1, 0)$ gives the number of poles in the right half plane. There are two in Fig. 10.53.

![Typical Nyquist plot, or polar plot](image)
The Nyquist plots of amplifiers with three different characteristics are shown in Fig. 10.54. For plot A, the loop gain is less than unity. Therefore, oscillations will not build up on the closed loop, and the amplifier will be absolutely stable. For plot B through \((-1, 0)\), the loop gain is unity, and the poles are on the imaginary axis. The amplifier will be critically stable on the closed loop. For plot C, the loop gain is greater than unity, and the amplifier will be absolutely unstable on the closed loop.

### 10.11.6 Relative Stability

The term “absolutely stable” or “absolutely unstable” tells us what will eventually happen to a system. However, it does not say how stable or unstable the system is. **Relative stability** is a measure of the degree of stability, and it indicates how far the intersection of the frequency locus with the Re-axis is from the point \((-1, 0)\) on the right side. For good relative stability, the magnitude at the phase crossover frequency should have a value less than unity, and the phase angle at the gain crossover frequency should not have a value near \(\pm 180^\circ\).

Gain margin and phase margin are normally used as measures of relative stability. **Gain margin** (GM) is defined as the number of decibels by which the magnitude \(x\) of the loop gain falls short of unity when the phase angle is \(180^\circ\); that is,

\[
GM = 20 \log 1 - 20 \log x = 20 \log \left( \frac{1}{x} \right)
\]  

(10.115)

**Phase margin** (PM) is defined as the amount of degrees by which the phase angle \(\phi\) of the loop gain falls short of \(\pm 180^\circ\) when the magnitude is unity; that is,

\[
PM = \phi_m = 180 - |\phi|
\]  

(10.116)

For adequate relative stability, the gain margin and phase margin should be greater than 10 dB and 45°, respectively.
EXAMPLE 10.15

Finding the phase margin and gain margin of a feedback amplifier

Determine the phase margin and gain margin of a feedback amplifier whose loop gain is given by

\[ T_L(j\omega) = \frac{2}{(1 + j\omega)^3} \]

**SOLUTION**

The locus will cross the negative Re-axis when the phase is \( \phi = -180^\circ \) at \( \omega = \omega_p = \omega_{180} \). Thus,

\[ 3 \tan^{-1}(\omega_p) = 180^\circ \quad \text{or} \quad \tan^{-1}(\omega_p) = 60^\circ \quad \text{or} \quad \omega_p = \sqrt{3} \text{ rad/s} \]

The magnitude at \( \omega = \omega_p \) becomes

\[ x = |T_L(j\omega)| = \frac{2}{(\sqrt{1 + 3})^3} = 0.25 \]

That is, \( x < 1 \). Thus, the frequency locus will not encircle the point \((-1, 0)\), and the amplifier will be absolutely stable on the closed loop. Using Eq. (10.115), we have

\[ \text{GM} = 20 \log \left( \frac{1}{0.25} \right) = 12.04 \text{ dB} \]

The gain crossover frequency \( \omega_g \) is found when the gain is unity:

\[ x = |T_L(j\omega_g)| = \frac{2}{(\sqrt{1 + \omega_g^2})^3} = 1 \]

so \( (1 + \omega_g^2)^3 = 2^3 \quad \text{or} \quad 1 + \omega_g^2 = 1.5874 \quad \text{or} \quad \omega_g = 0.7664 \text{ rad/s} \)

The phase angle \( \phi \) (at \( \omega = \omega_g \)) is

\[ \phi_g = 3 \tan^{-1}(\omega_g) = 3 \tan^{-1}(0.7664) = 3 \times 37.47^\circ = 112.4^\circ \]

Using Eq. (10.116), we have

\[ \text{PM} = \phi_m = 180^\circ - 112.4^\circ = 67.6^\circ \]

**10.11.7 Effects of Phase Margin**

At the gain crossover frequency \( \omega_g \), the magnitude of the loop gain is unity. That is,

\[ |T_L(j\omega_g)| = |A(j\omega_g)|\beta = 1 \]

or

\[ |A(j\omega_g)| = \frac{1}{\beta} \quad (10.117) \]

where \( \beta \) is assumed to remain constant and is independent of frequency. The phase margin influences the transient and frequency responses of a feedback amplifier, and its effects can be determined from Eq. (10.101). The effects of the phase margin will be illustrated through the analysis of four cases.
Case 1. The phase margin is PM = \( \phi_m = 30^\circ \), and \( |\phi| = 180^\circ - 30^\circ = 150^\circ \). Substituting Eq. (10.117) into Eq. (10.101) yields
\[
A_f(j\omega_g) = \frac{A(j\omega_g)}{1 + e^{j\phi}} = \frac{A(j\omega_g)}{1 + e^{-j150^\circ}} = \frac{A(j\omega_g)}{1 - 0.866 - j0.5} = \frac{A(j\omega_g)}{0.134 - j0.5}
\]
which gives the magnitude of the closed-loop gain as
\[
|A_f(j\omega_g)| = \frac{|A(j\omega_g)|}{0.517} = 1.93|A(j\omega_g)| = \frac{1.93}{\beta}
\]
Therefore, there will be a peak of 1.93 times the low-frequency gain of \( 1/\beta \).

Case 2. The phase margin is PM = \( \phi_m = 45^\circ \), and \( \phi = 180^\circ - 45^\circ = 135^\circ \).
\[
A_f(j\omega_g) = \frac{A(j\omega_g)}{1 + e^{-j135^\circ}} = \frac{A(j\omega_g)}{1 - 0.71 - j0.71} = \frac{A(j\omega_g)}{0.29 - j0.707}
\]
and
\[
|A_f(j\omega_g)| = \frac{|A(j\omega_g)|}{0.765} = 1.306|A(j\omega_g)| = \frac{1.306}{\beta}
\]
In this case, there will be a peak of 1.306 times the low-frequency gain of \( 1/\beta \).

Case 3. The phase margin is PM = \( \phi_m = 60^\circ \), and \( \phi = 180^\circ - 60^\circ = 120^\circ \).
\[
A_f(j\omega_g) = \frac{A(j\omega_g)}{1 + e^{-j120^\circ}} = \frac{A(j\omega_g)}{1 - 0.5 - j0.866} = \frac{A(j\omega_g)}{0.5 - j0.866}
\]
and
\[
|A_f(j\omega_g)| = |A(j\omega_g)| = \frac{1}{\beta}
\]
In this case, there will be no peak above the low-frequency gain of \( 1/\beta \).

Case 4. The phase margin is PM = \( \phi_m = 90^\circ \), and \( \phi = 180^\circ - 90^\circ = 90^\circ \).
\[
A_f(j\omega_g) = \frac{A(j\omega_g)}{1 + e^{-j90^\circ}} = \frac{A(j\omega_g)}{1 - j1.0}
\]
and
\[
|A_f(j\omega_g)| = \frac{|A(j\omega_g)|}{\sqrt{2}} = 0.707|A(j\omega_g)| = \frac{0.707}{\beta}
\]
In this case, there will be a gain reduction below the low-frequency gain of \( 1/\beta \).

The frequency responses of a transfer function with one pole and a transfer function with two poles are shown in Fig. 10.55 for various values of the phase margin. For a 90° phase margin, the transfer function has only one pole. In a two-pole system, as the phase margin is reduced, the gain peak increases until the gain approaches infinity, and oscillation occurs at a phase margin of \( \phi_m = 0 \). After the gain peak (at a normalized frequency of \( f/f_g = 1 \)), the gain decays with a slope of \(-40 \text{ dB/decade}\) because there are two poles in the transfer function.
10.11.8 Stability Using Bode Plots

A Bode plot, which is a plot of magnitude and phase against frequency, is a very convenient method of determining the stability of an amplifier. The loop gain is plotted in decibels, and the frequency is plotted on a logarithmic scale. Consider the loop gain with a single-pole frequency given by

\[ T_L(j\omega) = A(j\omega) = \frac{A_0}{1 + j\omega/\omega_p} = \frac{A_0}{1 + j\omega/f_{p1}} \]  

whose phase angle is \( \phi = \angle T_L(j\omega) = -\tan^{-1}(f/f_{p1}) \). The typical Bode plot is shown in Fig. 10.56(a). The magnitude of the loop gain is 20 log \( |T_L(j\omega)| \) until \( f = f_{p1} \), at which point the loop gain falls at a rate of 20 dB/decade and the phase angle is \( \phi = -45^\circ \). Phase margin is the difference between 180° and the phase angle when the gain is 0 dB. Gain margin, which is the value of the magnitude in dB when the phase angle is 180°, can be read from the plot. Thus, for a single-pole amplifier, \( \phi = -90^\circ \) at \( T_L(j\omega_p) = 1 \), and the phase margin is \( \phi_m = 180 - |\phi| = 90^\circ \). There is no phase crossover, and the gain margin is infinite. An amplifier with negative feedback will always be stable.

Consider the loop gain with three-pole frequencies given by

\[ T_L(j\omega) = A(j\omega) = \frac{A_o}{(1 + j\omega/f_{p1})(1 + j\omega/f_{p2})(1 + j\omega/f_{p3})} \]  

whose phase angle can be found from

\[ \phi = \angle T_L(j\omega) = -\tan^{-1}\left(\frac{f}{f_{p1}}\right) - \tan^{-1}\left(\frac{f}{f_{p2}}\right) - \tan^{-1}\left(\frac{f}{f_{p3}}\right) \]

The typical Bode plot is shown in Fig. 10.56(b). Assuming that the poles are widely separated (i.e., by a decade: \( f_{p3} = 10f_{p2} = 100f_{p1} \)), the phase angle \( \phi \) is approximately \(-45^\circ \) at the first pole frequency \( f_{p1} \), \((-90^\circ - 45^\circ = ) -135^\circ \) at the second pole frequency \( f_{p2} \), and \((-180^\circ - 45^\circ = ) -225^\circ \) at the third pole frequency \( f_{p3} \). The phase angle at \( T_L(j\omega_p) = 1 \) is \( \phi = -270^\circ \), which gives a phase margin of \( \phi_m = 180^\circ - 270^\circ = -90^\circ \); that is, there is no phase margin, and the amplifier will be absolutely unstable. However, at \( \omega = \omega_p = \omega_{180} \), the loop gain is positive; that is, there is a gain margin.
Rather than plotting the loop gain $20 \log |T_L(j\omega)|$ directly, we can choose to plot $20 \log |A(j\omega)|$ and $20 \log \left| \frac{1}{\beta(j\omega)} \right|$ separately. This approach is shown in Fig. 10.57 for two resistive values of $\beta$. The difference between the two curves gives

$$20 \log |A(j\omega)| = 20 \log \left| \frac{1}{\beta(j\omega)} \right|$$

The sum of the two phase angles gives

$$\angle A(j\omega) = \angle \frac{1}{\beta(j\omega)}$$

This approach has the advantage of allowing us to investigate the stability of an amplifier for a variety of feedback networks simply by drawing the lines for $20 \log \left| \frac{1}{\beta(j\omega)} \right|$. For zero gain margin,

$$20 \log |A(j\omega)| = 20 \log \left| \frac{1}{\beta(j\omega)} \right| = 0$$

That is, the intersection of the $20 \log \left| \frac{1}{\beta(j\omega)} \right|$ plot with the $20 \log |A(j\omega)|$ plot gives the critical value of $\beta$. We can see from Fig. 10.57 that the $-180^\circ$ phase occurs on the $-40 \text{ dB/decade}$ segment of the Bode plot. Therefore, for stability, the $20 \log \left| \frac{1}{\beta(j\omega)} \right|$ plot should intersect the $20 \log |A(j\omega)|$ plot at a point on the $-20 \text{ dB/decade}$ segment. As a general rule, the difference of slopes at the intersection—which is called the rate of closure—should not exceed $20 \text{ dB/decade}$. 

**FIGURE 10.56** Typical Bode plots
EXAMPLE 10.16

Finding the phase crossover frequency and the phase margin of an amplifier  The open-loop gain of an amplifier has break frequencies at $f_{p1} = 10$ kHz, $f_{p2} = 100$ kHz, and $f_{p3} = 1$ MHz. The low-frequency (or DC) gain is $A_0 = 2 \times 10^5$. Calculate (a) the phase crossover frequency $f_p$ and (b) the phase margin $\phi_m$ for $\beta = 0.01$, 0.001, and 0.0001.

SOLUTION

The magnitude of the open-loop gain is given by

$$A(j\omega) = \frac{2 \times 10^5}{(1 + j\omega/10^4)(1 + j\omega/10^5)(1 + j\omega/10^6)}$$

$$|A(j\omega)| = \frac{2 \times 10^5}{[1 + (f/10^4)^2]^{1/2}[1 + (f/10^5)^2]^{1/2}[1 + (f/10^6)^2]^{1/2}}$$

and the phase angle is

$$\phi = -\tan^{-1}\left(\frac{f}{10^4}\right) - \tan^{-1}\left(\frac{f}{10^5}\right) - \tan^{-1}\left(\frac{f}{10^6}\right)$$

(a) The frequency at which $\phi = -180^\circ$ is found by iteration as $f_p = f_{180} = 333$ kHz.

(b) For $\beta = 0.01$ and $|A(j\omega)| = 1/\beta = 100$, we get $f_g = 1144.8$ kHz by iteration; that is,

$$\phi = -223.4^\circ \quad \text{and} \quad \phi_m = 180 - |\phi| = -43.4^\circ$$

Thus, the amplifier will be unstable.
For $\beta = 0.001$ and $|A(j\omega)| = 1/\beta = 1000$, we get $f_g = 423.3$ kHz by iteration; that is,

$$\phi = -188.3^\circ \quad \text{and} \quad \phi_m = 180 - |\phi| = -8.3^\circ$$

Thus, the amplifier will be unstable.

For $\beta = 0.0001$ and $|A(j\omega)| = 1/\beta = 10,000$, we get $f_g = 124.1$ kHz by iteration; that is,

$$\phi = -143.6^\circ \quad \text{and} \quad \phi_m = 180 - |\phi| = 36.4^\circ$$

Thus, the amplifier will be stable.

We can conclude that a feedback amplifier can be stable or unstable, depending on the value of $\beta$.

**EXAMPLE 10.17**

**Finding the phase crossover frequency and the phase margin of an amplifier** The open-loop gain of an amplifier has break frequencies at $f_{p1} = 100$ kHz, $f_{p2} = 200$ kHz, and $f_{p3} = 1$ MHz. The low-frequency (or DC) gain is $A_o = 800$, and the feedback factor is $\beta = 0.5$. Calculate the gain crossover frequency $f_g$ and the phase margin $\phi_m$.

**SOLUTION**

The low-frequency loop gain is $A_o\beta = 800 \times 0.5 = 400$. The magnitude of the loop gain is given by

$$|T_L(j\omega)| = x = \frac{400}{[1 + (f/f_{p1})^2]^{1/2}[1 + (f/f_{p2})^2]^{1/2}[1 + (f/f_{p3})^2]^{1/2}}$$

and the phase angle is

$$\phi = -\tan^{-1}\left(\frac{f}{f_{p1}}\right) - \tan^{-1}\left(\frac{f}{f_{p2}}\right) - \tan^{-1}\left(\frac{f}{f_{p3}}\right)$$

At the gain crossover $|T_L(j\omega_g)| = 1$, the gain crossover frequency $f_g$ can be determined from the Bode plot or by iteration. By iteration, the value of frequency that gives a loop gain of unity is found to be $f_g = 1917.03$ kHz, and the corresponding value of phase angle is $\phi = -233.5^\circ$. Thus, the phase margin is

$$\phi_m = 180^\circ - |\phi| = 180^\circ - 233.5^\circ = -53.5^\circ$$

which is negative, so the amplifier will be unstable.

**KEY POINTS OF SECTION 10.11**

- An amplifier with negative feedback can be stable or unstable, depending on the frequency and the feedback factor $\beta$. If the loop gain is $|T_L(j\omega)| \approx 1$ and its phase angle is $\phi = \pm 180^\circ$, the amplifier will be unstable.
- Nyquist and Bode plots can be used to determine the stability of a feedback amplifier. The degree of stability is normally measured by the gain margin and the phase margin. A phase margin of $45^\circ$ is usually adequate to limit the peak to 30% of the low-frequency gain.
10.12 Compensation Techniques

We know that if an amplifier has more than two poles, the phase angle of the loop gain could exceed $-180^\circ$ beyond a certain frequency. An amplifier with negative feedback can be unstable, depending on the frequency $\omega$ and the amount of feedback $\beta$. The process of stabilizing an unstable feedback amplifier is called compensation. The basic amplifier of a feedback circuit should be designed with as few stages as possible because each stage of gain adds more poles to the transfer function, making the compensation problem more difficult. An amplifier can be stabilized by adding a dominant pole, by changing the dominant pole, by Miller compensation, or by modifying the feedback path [3–5].

10.12.1 Addition of a Dominant Pole

A dominant pole can be introduced into the amplifier so that the phase shift is less than $-180^\circ$ when the loop gain is unity. Consider a feedback amplifier with loop gain of the form

$$T_L(j\omega) = A(j\omega)\beta = \frac{A_o}{(1 + j\omega/\omega_p1)(1 + j\omega/\omega_p2)(1 + j\omega/\omega_p3)}$$  \hspace{1cm} (10.125)

where $\omega_{p1} = 2\pi f_{p1}$ rad/s, $\omega_{p2} = 2\pi f_{p2}$ rad/s, and $\omega_{p3} = 2\pi f_{p3}$ rad/s. The Bode plot is shown in Fig. 10.58(a). To compensate the amplifier, a new dominant pole $\omega_D = 2\pi f_D$ is introduced so that $\omega_D < \omega_{p1} < \omega_{p2} < \omega_{p3}$, and the resultant loop gain becomes

$$T_L(j\omega) = A(j\omega)\beta = \frac{A_o}{(1 + j\omega/\omega_D)(1 + j\omega/\omega_p1)(1 + j\omega/\omega_p2)(1 + j\omega/\omega_p3)}$$  \hspace{1cm} (10.126)

![FIGURE 10.58 Compensation by addition of a dominant pole](image-url)
The Bode plot of this modified loop gain is indicated in Fig. 10.58(a) by lighter lines. The introduction of the dominant pole causes the loop gain to fall at a rate of 20 dB/decade until frequency $f_{p1}$ is reached. If the frequency $f_D$ of the dominant pole is chosen so that the loop gain is unity at frequency $f_{p1}$, then the phase shift at frequency $f_{p1}$ due to the dominant pole $f_D$ is $-90^\circ$, and the phase shift due to the first pole $f_{p1}$ is $-45^\circ$. At $f = f_{p1}$, the total phase shift is $(-90^\circ - 45^\circ =) -135^\circ$, and the phase margin is $\phi_m = 180^\circ - |\phi| = 180^\circ - 135^\circ = 45^\circ$, which means that the amplifier will be stable. The original amplifier would have been unstable in the feedback connection. This compensation is achieved at the expense of bandwidth reduction. The uncompensated unity-gain bandwidth $f_{p1}$ is much higher than the compensated unity-gain bandwidth $f_D$. This method of compensation involves a sacrifice of the frequency capability of the amplifier and is often known as narrowbanding.

A dominant pole can be implemented by adding a capacitor $C_x$ in such a manner that it adds a break frequency to the basic amplifier. With this approach, illustrated in Fig. 10.58(b) showing two possible locations, $f_D$ is given by

$$f_D = \frac{1}{2\pi RC_x}$$

(10.127)

where $R$ is the resistance seen by capacitor $C_x$.

**EXAMPLE 10.18**

**Stabilizing an amplifier by adding a dominant pole** Stabilize the amplifier in Example 10.17 by adding a dominant pole so that the phase margin is $45^\circ$.

**SOLUTION**

Since the gain–bandwidth product must remain constant, the low-frequency loop gain should be reduced from $A_0\beta$ (or $800 \times 0.5 = 400$) at $f_D$ to unity at $f_{p1}$ ($=100$ kHz) with a $-20$ dB/decade slope, which indicates a direct proportionality; that is, $f_D \times A_0\beta = f_{p1} \times 1$, which gives

$$f_D = \frac{f_{p1}}{A_0\beta} = \frac{100 \times 10^3}{400} = 250 \text{ Hz}$$

Therefore, the modified loop gain is given by

$$T_L(j\omega) = A(j\omega)\beta = \frac{400}{(1 + jf_{p1})(1 + jf_{p2})(1 + jf_{p3})}$$

**NOTES**

1. It is assumed in determining $f_D$ that the break frequency $f_{p2}$ does not affect the phase shift. However, in this example, $f_{p2}$ is close to $f_{p1}$ and will contribute to the phase shift. The gain crossover frequency, which is obtained by iteration, is $f_g = 74,795$ Hz, the phase shift is $\phi = -151^\circ$, and the phase margin is $\phi_m = 180^\circ - 151^\circ = 29^\circ$, which is less than the desired phase margin of $45^\circ$.
2. This method of compensation gives an approximate value for the dominant pole frequency. Fine-tuning is necessary to obtain the desired phase margin.
3. The value of $f_D$, which can be adjusted to obtain a phase margin of $45^\circ$, is $152$ Hz at a gain crossover frequency of $f_g = 52.106$ kHz.
10.12.2 Changing the Dominant Pole

In the compensation method just discussed in Sec. 10.12.1, a dominant pole was added to the amplifier, and the original amplifier poles were assumed to be unaffected. This approach reduces the bandwidth considerably. A second method of compensation is to change the dominant pole, adding a capacitor to the amplifier in such a way that the original dominant frequency $f_{p1}$ is reduced so that it performs the compensation function. That is, the original pole $f_{p1}$ is moved to the left so that $f_D = f_{p1}$. This modification is shown in Fig. 10.59. For a 45° phase margin in a unity feedback amplifier, $f_D$ must cause the gain to fall to unity at $f_{p2}$. Thus, $f_{p2}$ will become the unity-gain bandwidth. Since $f_{p2}$ is five or ten times the frequency $f_{p1}$, this method gives a substantial improvement in bandwidth.

The dominant pole is changed by adding a capacitor internally to the amplifier. One way of doing so is shown in Fig. 10.60(a) for the differential stage of an op-amp. The differential half circuit is shown in Fig. 10.60(b), and the small-signal equivalent circuit is shown in Fig. 10.60(c). The current $i_s$ is the collector current of $Q_2$, $C_i$ is the internal capacitance, and $R_i$ is the effective resistance; that is, $C_i = C_{pi4}$, and $R_i = R_C || r_{o2} || r_{pi4}$. Thus, the dominant pole frequency can be found from

$$f_D = f_{p1} = \frac{1}{2\pi R_i(C_i + C_x)} \quad (10.128)$$

where $C_x$ is the additional capacitance that is added to give the desired dominant frequency. The disadvantage of this method is that the value of $C_x$ is usually quite large (typically $>1000 \text{ pF}$). Thus, it will be difficult—if not impossible—to realize a compensating capacitor in an IC chip. The maximum practical size of a monolithic capacitor is about 100 pF.
We have assumed that the addition of $C_x$ will reduce the original dominant pole frequency $f_{p1}$ so that it performs the compensation function. However, the higher-frequency poles of the amplifier will also be changed by the addition of $C_x$. In practice, the effect of $C_x$ on the pole positions is usually evaluated by computer simulation. Another estimate of $C_x$ is made on the basis of the new data, and this process of trial and error continues until the desired stability condition is reached, usually after several iterations.

**EXAMPLE 10.19**

Finding the compensating capacitance to modify the dominant pole

The equivalent circuit of an output stage is shown in Fig. 10.60(c), where $R_i = 24$ kΩ and $C_i = 20$ pF.

(a) Find the break frequency $f_p$.

(b) Find the additional capacitance $C_x$ that will move the break frequency to $f_D = 40$ kHz.

**SOLUTION**

(a) The break frequency $f_p$ is given by

$$f_p = \frac{1}{2\pi C_i R_i} = \frac{1}{2\pi \times 20 \times 10^{-12} \times 24 \times 10^3} = 331.6 \text{ kHz}$$

(b) An additional capacitance $C_x$ will move the break frequency to $f_D$; that is,

$$f_D = \frac{1}{2\pi (C_i + C_x) R_i}$$

which, for $f_D = 40$ kHz, gives $C_i + C_x = 165.8$ pF; that is, $C_x = 165.8 - 20 = 145.8$ pF.
10.12.3 Miller Compensation and Pole Splitting

In a third compensation method, a small capacitance (say, $C_x$) is connected between the input and output of a gain stage in a multistage amplifier. Compensation is achieved using Miller multiplication of the capacitance. Figure 10.61(a) illustrates this form of compensation for the μA741 op-amp. $C_x$ constitutes a shunt-shunt feedback, and the feedback factor $\beta$ depends on the frequency. The Darlington pair consisting of $Q_{16}$ and $Q_{17}$ can be replaced by a single equivalent transistor $Q$, as shown in Fig. 10.61(b); the simplified equivalent circuit is shown in Fig. 10.61(c). $R_i$ and $C_i$ represent the total resistance and total capacitance, respectively, between node B and the ground. $R_o$ and $C_o$ represent the total resistance and total capacitance, respectively, between node C and the ground. $C_i$ includes the Miller capacitance due to $C_x$ of the transistor and the output capacitance of the preceding stage. Similarly, $C_o$ includes the Miller capacitance due to $C_\mu$ and the input capacitance of the succeeding stage. In the absence of the compensating capacitance (i.e., when $C_x = 0$), the two poles will be

$$f_{p1} = \frac{1}{2\pi C_i R_i} \quad (10.129)$$

and

$$f_{p2} = \frac{1}{2\pi C_o R_o} \quad (10.130)$$

The analysis of Fig. 10.61(c) will be similar to that of Fig. 2.23(b). Using Eqs. (2.93) and (2.94), we get as the new poles

$$\omega'_p \approx \frac{1}{g_m C_x R_i R_o} \quad (10.131)$$

and

$$\omega'_p \approx \frac{g_m C_x}{C_i C_o + C_x (C_i + C_o)} \quad (10.132)$$
If \( C_x \gg C_o \), Eq. (10.132) can be approximated by

\[
\omega_{p2}' = \frac{g_m}{C_i(C_o/C_x) + (C_i + C_o)} \approx \frac{g_m}{C_i + C_o}
\]  

(10.133)

From the preceding equations, we can see that as \( C_x \) is increased, \( \omega_{p1}' \) is reduced, and \( \omega_{p2}' \) is increased. This separation of poles is called pole splitting. The increase in \( \omega_{p2}' \) will move the pole frequency for a 45° phase margin to the right and will hence widen the bandwidth. In Eq. (10.131), \( C_x \) is multiplied by a factor \( g_mR_o \), and the effective capacitance is much larger: \( g_mR_oC_x \). Thus, the value of \( C_x \) will be much smaller than it is when compensation is achieved by adding or changing a pole.

**EXAMPLE 10.20**

*Finding the compensating capacitance by pole splitting*  The open-loop gain of an amplifier is \( 2 \times 10^3 \) and has break frequencies at \( f_{p1} = 100 \text{ kHz}, f_{p2} = 1 \text{ MHz}, \) and \( f_{p3} = 10 \text{ MHz}. \) The gain stage of the amplifier has the equivalent circuit shown in Fig. 10.61(c), whose parameters are \( g_m = 100 \text{ mA/V}, C_i = 50 \text{ pF}, \) and \( C_o = 10 \text{ pF}. \) Determine the value of compensating capacitance \( C_x \) that will give a closed-loop phase margin of 45° with a resistive feedback of up to \( \beta = 1 \).

**SOLUTION**

\( g_m = 100 \times 10^{-3}, C_i = 50 \text{ pF}, C_o = 10 \text{ pF}, f_{p1} = 100 \text{ kHz}, f_{p2} = 1 \text{ MHz}, \) and \( f_{p3} = 10 \text{ MHz}. \) We can find the values of \( R_i \) and \( R_o \) as follows:

\[
R_i = \frac{1}{2\pi f_{p1}C_i} = \frac{1}{2\pi \times 100 \text{ kHz} \times 50 \text{ pF}} = 31.83 \text{ k}\Omega
\]

\[
R_o = \frac{1}{2\pi f_{p2}C_o} = \frac{1}{2\pi \times 1 \text{ MHz} \times 10 \text{ pF}} = 15.92 \text{ k}\Omega
\]

From Eq. (10.133), we can find the modified value \( f_{p2}' \); that is,

\[
f_{p2}' \approx \frac{g_m}{2\pi(C_i + C_o)} = \frac{100 \times 10^{-3}}{2\pi(50 \text{ pF} + 10 \text{ pF})} = 265.3 \text{ MHz}
\]

which is more than \( f_{p3} (= 10 \text{ MHz}) \). Thus, let us assume that \( f_{p3} \) will be the second pole frequency and find the compensation capacitance \( C_x \) to set the 45° phase margin at \( f_{p3} = 10 \text{ MHz} \) with unity gain. That is, \( f_{p1} \times A_o\beta = f_{p3} \times 1 \), which gives the modified dominant pole frequency as

\[
f_{p1}' \approx f_D = \frac{f_{p3}}{A_o\beta} = \frac{10 \times 10^6}{2 \times 10^5} = 50 \text{ Hz}
\]

Thus, \( f_{p1}' = f_D = 50 \text{ Hz} \).

From Eq. (10.131), we get the capacitance \( C \) for the first dominant pole \( f_{p1}' \):

\[
C_x \approx \frac{1}{2\pi f_{p1}'g_mR_o} = \frac{1}{2\pi \times 50 \text{ Hz} \times 100 \text{ mA/V} \times 31.83 \text{ k}\Omega \times 15.92 \text{ k}\Omega} = 62.8 \text{ pF}
\]

From Eq. (2.91), we get

\[
f_{p1}' = \frac{1}{2\pi[R_i(C_i + C_x) + R_o(C_x + C_o) + g_mC_xR_oR_o]} = 49.94 \text{ Hz}
\]

which is close to 50 Hz.
10.12.4 Modification of the Feedback Path

Op-amps are normally compensated by adding a capacitor internally. This type of compensation, however, wastes bandwidth because the bandwidth is reduced considerably. For example, the dominant pole frequency of the μA741 op-amp is only 10 Hz.

Compensation can also be accomplished by modifying the feedback network so that the feedback factor $\beta$ becomes frequency dependent and has a zero that cancels the pole of the original amplifier. This method is generally used in the compensation of fixed-gain amplifiers, where achieving a wide bandwidth is of prime concern.

Let us consider a feedback network with one pole and one zero. Then the loop gain of an amplifier with three poles will be of the form

$$A(j\omega)\beta(j\omega) = \frac{A_0(1 + j\omega/\omega_z)}{(1 + j\omega/\omega_p)(1 + j\omega/\omega_{p1})(1 + j\omega/\omega_{p2})(1 + j\omega/\omega_{p3})} \quad (10.134)$$

where $\omega_p$ and $\omega_z$ are the pole and the zero of the feedback network, respectively.

A typical implementation is shown in Fig. 10.62(a) for a shunt-series feedback amplifier. The feedback network includes a capacitor $C_F$. The loading effects of $C_F$ at the input and output sides of the amplifier are shown in Fig. 10.62(b). The capacitor $C_F$ will have only a minor effect on the amplifier transfer

![Shunt-series feedback network](image1)

![Loading effect](image2)

![Feedback network for determining $\beta$](image3)

**FIGURE 10.62** Compensation by feedback path modification
function $A(s)$. The feedback network is shown in Fig. 10.62(c), from which we can find the feedback transfer function as

$$
\beta(s) = \frac{i_t}{i_o} = \frac{R_E}{R_E + R_F} \times \frac{1 + sR_FC_F}{1 + sC_F R_E/(R_E + R_F)}
$$

(10.135)

$$
= \frac{\beta_o(1 + s/\omega_z)}{1 + s/\omega_p}
$$

(10.136)

where

- $\beta_o = R_E/(R_E + R_F)$ = low-frequency feedback factor
- $\omega_z = 1/(R_F C_F)$ = zero of the feedback network
- $\omega_p = (R_E + R_F)/(R_E R_F C_F)$ = pole of the feedback network

**EXAMPLE 10.21**

**Compensating by feedback path modification** The open-loop gain of the amplifier in Example 10.12 has break frequencies at $f_{p1} = 100$ kHz, $f_{p2} = 1$ MHz, and $f_{p3} = 10$ MHz. The low-frequency gain is $A_o = 200$ A/A, and the emitter resistance is $R_E = 500$ Ω. Determine the values of compensating capacitance $C_F$ and resistance $R_F$ (a) to give a low-frequency closed-loop gain of $A_f = 20$ A/A and cancel the pole $f_{p1} = 100$ kHz and (b) to add a pole of $f_p = 10$ MHz and cancel the pole $f_{p1} = 100$ kHz.

**SOLUTION**

- $f_{p1} = 100$ kHz, $f_{p2} = 1$ MHz, $f_{p3} = 10$ MHz, $A_o = 200$ A/A, and $R_E = 500$ Ω.

(a) Substituting $A_f = 20$ A/A and $A_o = 200$ A/A into $A_f = A_o/(1 + \beta_o A_o)$ gives $\beta_o = 0.045$. Thus,

$$
\beta_o = 0.045 = \frac{R_E}{R_E + R_F}
$$

which, for $R_E = 500$ Ω, gives $R_F = 10.61$ kΩ. To cancel the pole $f_{p1}$, we use

$$
f_z = f_{p1} = \frac{1}{2\pi R_F C_F}
$$

For $f_{p1} = 100$ kHz and $R_F = 10.61$ kΩ, we get $C_F = 150$ pF.

(b) Substituting $f_p = 10$ MHz and $f_z = f_{p1} = 100$ kHz into $f_p = f_z/\beta_o$ gives $\beta_o = 0.01$. Thus,

$$
\beta_o = 0.01 = \frac{R_E}{R_E + R_F}
$$

which, for $R_E = 500$ Ω, gives $R_F = 49.5$ kΩ. To cancel the pole $f_{p1}$, we use

$$
f_z = f_{p1} = \frac{1}{2\pi R_F C_F}
$$

For $f_{p1} = 100$ kHz and $R_F = 49.5$ kΩ, we get $C_F = 32.15$ pF.
EXAMPLE 10.22

Compensating by feedback path modification

(a) A common-emitter amplifier with shunt-shunt feedback is shown in Fig. 10.63. The capacitance $C_F$ and resistance $R_F$ form the feedback network. The voltage gain without feedback is $A = \frac{A_v}{v_o/v_s} \approx 100$. Design the feedback network to meet the following specifications:

- The bandwidth with feedback must be increased by a factor of 10; that is, $BW_f = 10BW$.
- The voltage gain with feedback must be $|A_f| = \frac{v_o}{v_s} = 10$.

(b) Use PSpice/SPICE to check your results.

SOLUTION

(a) Shunt-shunt feedback will reduce both the input resistance and the output resistance, but it should widen the bandwidth. The value of $C_F$ should be such that it is virtually short-circuited over the frequency range of the amplifier. The feedback is taken from the output side because the output is out of phase with the input voltage $v_s$. Since the gain of the amplifier is $A = 10^2$, the closed-loop gain will be almost independent of $A$.

We get

$$\frac{v_o}{i_s} = \frac{1}{\beta} = -R_F$$

and

$$i_s = \frac{v_s}{R_s}$$

which gives the closed-loop gain as

$$A_f = \frac{v_o}{v_s} = -\frac{R_F}{R_s} \approx -\frac{1}{\beta R_s} \quad (V/V)$$

(10.138)

FIGURE 10.63

Common-emitter amplifier with shunt-shunt feedback
For shunt-shunt feedback, \( BW_f = BW(1 + \beta A) \). The gain must decrease by a factor of 10. Thus, 
\[
\frac{1}{\beta R_s} = \frac{R_F}{R_s} = 10
\]
which, for \( R_s = 200 \Omega \), gives \( R_F = 2 \text{k}\Omega \). We want to choose a capacitor that will ensure that \( C_F \) is virtually short-circuited at low frequency, \( f_L = 1 \text{kHz} \): let’s choose \( C_F = 5 \mu\text{F} \). Thus,
\[
X_{CF} = \frac{1}{2\pi \times 1 \text{kHz} \times 5 \mu\text{F}} = 31.8 \text{\Omega}
\]
(b) The PSpice plots of the frequency response are shown in Fig. 10.64 for \( R_F = 2 \text{k}\Omega \) and \( 1 \text{M}\Omega \). For \( R_F = 1 \text{M}\Omega \) (for almost no feedback, \( \beta \approx 0 \)), we get \( |A| = 100, f_L = 1089 \text{Hz}, f_H = 1838 \text{kHz} \), and
\[
BW = f_H - f_L = 1838 \text{k} - 1.09 \text{k} = 1836.91 \text{kHz}
\]
With feedback of \( R_F = 2 \text{k}\Omega \) (for \( \beta = 0.1 \)), we get \( |A| = 8.43 \) (expected value is 10), \( f_Lf = 198 \text{Hz}, f_Hf = 21,084 \text{kHz} \), and
\[
BW_f = f_Hf - f_Lf = 21,084 \text{k} - 0.2 \text{k} = 21,083.8 \text{kHz}
\]
(expected value is 10BW = 10 \times 1836.91 kHz = 18,369 kHz). The difference between the PSpice values and the expected values is caused by the fact that we did not include the effects of resistances such as \( R_F \) and \( R_C \) in hand calculations.

**KEY POINTS OF SECTION 10.12**

- An amplifier may be unstable when feedback is applied. The process of stabilizing an unstable amplifier is called compensation.
- The compensation of feedback amplifiers is normally achieved by connecting an external capacitor to the basic amplifier in such a way as to add a dominant pole or to split the poles. This type of compensation is normally applied to op-amps and usually reduces the bandwidth of the amplifier.
- Compensation can also be achieved by connecting a capacitor to the feedback network such that it adds a pole and a zero to the loop gain. This type of compensation is normally applied to fixed-gain amplifiers so as to yield a wide bandwidth.
**Summary**

There are two types of feedback: negative feedback and positive feedback. Negative feedback is normally used in amplifier circuits, and positive feedback is applied exclusively in oscillators. Negative feedback has certain advantages, such as stabilization of overall gain with respect to parameter variations, reduction of distortion, reduction of the effects of nonlinearity, and increase in bandwidth. However, these advantages are obtained at the expense of gain reduction, and additional amplifier stages may be required to make up the gain reduction. If the loop gain $\frac{Z_{if}}{A}$, the overall (or closed-loop) gain depends inversely on the feedback factor $\frac{Z_{of}}{A}$ and is directly sensitive to changes in the feedback factor.

The gain–bandwidth product of feedback amplifiers remains constant. If the gain is reduced by negative feedback, then the bandwidth is increased by the same amount. Depending on its implementation in electronic circuits, feedback can be classified as one of four types: series-shunt, shunt-shunt, series-series, or shunt-series. A shunt connection reduces the input (or output) impedance by a factor of $(1 + \beta A)$, and a series connection increases the input (or output) impedance by a factor of $(1 + \beta A)$. The closed-loop gain is always decreased by a factor of $(1 + \beta A)$. Table 10.3 summarizes the effects of various feedback topologies.

The stability of an amplifier depends on the poles of the transfer function. For a stable amplifier, the characteristic equation should not have any roots with positive real parts. The Nyquist criterion is one of the methods for determining the stability of feedback systems. The stability of an amplifier is normally measured in terms of phase margin and gain margin. A Bode plot can also be used to determine the stability of an amplifier. An inherently unstable amplifier can be made stable by introducing a dominant pole in the transfer function; this is generally achieved in electronic circuits by connecting a feedback capacitor.

**References**

Review Questions

1. What are the two types of feedback?
2. What are the advantages of feedback?
3. What are the disadvantages of feedback?
4. What is loop transmission?
5. What is gain sensitivity?
6. What is feedback factor sensitivity?
7. What is the difference between open-loop gain and closed-loop gain?
8. What are the four types of feedback topologies?
9. What are the features of series-shunt feedback?
10. What are the features of shunt-shunt feedback?
11. What are the features of shunt-series feedback?
12. What are the features of series-series feedback?
13. What are the effects of the feedback network on amplifier performance?
14. What are the effects of the source impedance on amplifier performance?
15. What are the effects of the load impedance on amplifier performance?
16. How do you take into account the $\beta$ dependency of $A$?
17. How do you find the modified gain $A_f$ of a feedback amplifier?
18. What are the steps in designing a feedback network?
19. What is a characteristic equation?
20. What are the effects of the poles on the stability of an amplifier?
21. What is the Nyquist criterion?
22. What are the conditions for instability?
23. What is gain crossover frequency?
24. What is phase crossover frequency?
25. What is relative stability?
26. What is a phase margin?
27. What is a gain margin?
28. What is the effect of phase margin on system response?
29. What is a Bode plot?
30. What is compensation?
31. What is the common method of compensation in amplifiers?
32. How is a dominant pole introduced in electronic circuits?

Problems

The symbol $\text{D}$ indicates that a problem is a design problem. The symbol $\text{P}$ indicates that you can check the solution to a problem using PSpice/SPICE or Electronics Workbench.

10.3 Characteristics of Feedback

10.1 Three voltage amplifiers are cascaded, as shown in Fig. P10.1.
   a. Determine the value of $\beta$ to give a closed-loop gain of $A_f = -100$.
   b. If the gain of each stage increases by 10%, determine $A_f$. Use the $\beta$ of part (a).
10.2 The open-loop gain of an amplifier with negative feedback is $A = 50$, and the feedback factor is $\beta = 0.5$.

a. Determine the closed-loop gain $A_f$.

b. If the open-loop gain $A$ changes by $+15\%$, determine the percentage change in the closed-loop gain $A_f$ and its value.

c. If the feedback factor $\beta$ changes by $+15\%$, determine the percentage change in the closed-loop gain $A_f$ and its value.

10.3 The open-loop gain of an amplifier is $A = 50$, and the feedback factor is $\beta = 0.8$. If the open-loop gain $A$ changes by $-20\%$ and the feedback factor $\beta$ changes by $+15\%$, determine the closed-loop gain $A_f$.

10.4 Two feedback amplifiers are connected in series. Each amplifier has a distortion of $20\%$. Determine the overall gain and the overall distortion if (a) each amplifier has its own feedback, as shown in Fig. P10.4(a), and (b) only one feedback is applied to the cascaded amplifiers, as shown in Fig. P10.4(b).

10.5 A feedback amplifier is to have a closed-loop gain of $A_f = 60$ dB and a sensitivity of $10\%$ to the open-loop gain $A$. Determine the open-loop gain with a unity feedback $\beta = 1$.

10.6 The feedback factor of an amplifier is $\beta = 0.5$. The open-loop gain $A$, which is frequency dependent, can be expressed as

$$A(j\omega) = \frac{2 \times 10^5}{1 + j\omega/10}$$

Determine (a) the closed-loop low-frequency gain $A_{of}$, (b) the closed-loop bandwidth $BW$, and (c) the gain–bandwidth product $GBW$.

10.7 The feedback factor of an amplifier is $\beta = 0.8$. The open-loop gain $A$ can be expressed in Laplace’s domain of $s$ as

$$A(s) = \frac{250s}{(1 + 0.1s)(1 + 0.001s)}$$

Determine (a) the closed-loop low-frequency gain $A_{of}$, (b) the closed-loop bandwidth $BW$, and (c) the gain–bandwidth product $GBW$. 
10.8 The transfer characteristic of an amplifier is described by the following values of open-loop gain $A$ for given ranges of input voltage $v_i$:

$$
A = \begin{cases} 
50 & \text{for } 0 < v_i \leq 0.25 \text{ V} \\
40 & \text{for } 0.25 \text{ V} < v_i \leq 1 \text{ V} \\
10 & \text{for } 1 \text{ V} < v_i \leq 2 \text{ V} \\
0 & \text{for } v_i > 2 \text{ V}
\end{cases}
$$

If the feedback factor is $\beta = 0.8$, determine the closed-loop gains of the transfer characteristic.

10.6 Series-Shunt Feedback

10.9 The noninverting amplifier shown in Fig. 10.7(a) has $R_1 = 40 \, \text{k}\Omega$, $R_I = 10 \, \text{k}\Omega$, $R_L = 15 \, \text{k}\Omega$, and $R_S = 0$. The op-amp parameters are $R_I = 2 \, \text{M}\Omega$ and $R_o = 50 \, \Omega$, and the open-loop voltage gain is $\mu_g = 2 \times 10^5$.

Determine (a) the input resistance seen by the source $R_{if} = v_s/i_s$, (b) the output resistance $R_{of}$, and (c) the closed-loop voltage gain $A_f = v_o/v_s$.

10.10 Repeat Prob. 10.9, ignoring the loading effects of the feedback network and the load resistance.

10.11 A voltage amplifier with negative feedback is shown in Fig. 10.47. The amplifier has an open-loop voltage gain of $A = 250$, an input resistance of $R_i = 4.5 \, \text{k}\Omega$, and an output resistance of $R_o = 500 \, \Omega$. The load resistance is $R_L = 10 \, \text{k}\Omega$. The feedback circuit has $R_f = 24 \, \text{k}\Omega$ and $R_F = 8 \, \text{k}\Omega$. The source has a resistance of $R_S = 2 \, \text{k}\Omega$. Determine (a) the input resistance $R_{if} = v_s/i_s$, (b) the output resistance $R_{of}$, and (c) the overall voltage gain $A_f = v_o/v_s$.

10.12 Repeat Prob. 10.11, ignoring the loading effects of the feedback network and the load resistance.

10.13 The feedback amplifier in Fig. P10.13 has $A_1 = 50$, $A_2 = 60$, $R_S = 500 \, \Omega$, $R_I = 15 \, \text{k}\Omega$, $R_2 = 1.5 \, \text{k}\Omega$, $R_3 = 250 \, \Omega$, $R_4 = 1.5 \, \text{k}\Omega$, $R_5 = 250 \, \Omega$, $R_6 = 2 \, \text{k}\Omega$, $R_L = 4.7 \, \text{k}\Omega$, $R_F = 500 \, \Omega$, $C_1 = C_2 = C_3 = 0.1 \, \text{\mu F}$, and $v_s = 100 \, \text{mV}$. Determine (a) the input resistance $R_{if} = v_s/i_s$, (b) the output resistance $R_{of}$, and (c) the overall voltage gain $A_f = v_o/v_s$. Assume $C_1$, $C_2$, and $C_3$ are shorted at the operating frequency.

10.14 Repeat Prob. 10.13, ignoring the loading effects of the feedback network and the load resistance.
10.15 Use PSpice/SPICE to plot output impedance against frequency for the feedback amplifier in Prob. 10.13. The frequency should be varied from 10 Hz to 1 MHz in increments of 1 decade and 10 points per decade.

10.16 The AC equivalent circuit of the CE amplifier in Fig. P10.16 has $R_1 = 10 \, \text{k} \Omega$, $R_2 = 1.5 \, \text{k} \Omega$, $R_C = 1.5 \, \text{k} \Omega$, $R_E = 250 \, \Omega$, $R_s = 200 \, \Omega$, $R_3 = 24 \, \text{k} \Omega$, $R_4 = 8 \, \text{k} \Omega$, and $R_L = 1 \, \text{k} \Omega$. The transistor parameters are $h_{fe} = 150$, $r_{\pi} = 2.5 \, \text{k} \Omega$, and $r_o = 25 \, \text{k} \Omega$. Use the techniques of feedback analysis to calculate the (a) input resistance $R_{if}$, (b) the output resistance $R_{of}$, and (c) the closed-loop voltage gain $A_f$.

![FIGURE P10.16](image-url)

10.17 Use the techniques of feedback analysis to calculate the input resistance $R_{if}$, the output resistance $R_{of}$, and the closed-loop voltage gain $A_f$ of the amplifier in Fig. 10.18(a) with series-shunt feedback. The DC bias currents of the transistors are $I_{C1} = 0.1 \, \text{mA}$, $I_{C2} = 0.5 \, \text{mA}$, and $I_{C3} = 2 \, \text{mA}$. The transistor parameters are $h_{fe} = h_{fe1} = h_{fe2} = h_{fe3} = 150$, $r_{\pi} = 25 \, \text{k} \Omega$, and $r_o = \infty$.

10.18 The emitter follower in Fig. P10.18 has $R_B = 75 \, \text{k} \Omega$, $R_E = 750 \, \Omega$, $R_L = 10 \, \text{k} \Omega$, and $R_s = 250 \, \Omega$. The transistor parameters are $h_{fe} = 150$, $r_{\pi} = 250 \, \Omega$, and $r_o = \infty$. Draw a block diagram of the feedback mechanism. Use the techniques of feedback analysis to calculate (a) the input resistance $R_{if}$, (b) the output resistance $R_{of}$, and (c) the closed-loop voltage gain $A_f$.

![FIGURE P10.18](image-url)

10.19 Use the techniques of feedback analysis to calculate the input resistance $R_{if}$, the output resistance $R_{of}$, and the closed-loop voltage gain $A_f$ of the amplifier in Fig. P10.19. The transistor parameters are $h_{fe} = h_{fe1} = h_{fe2} = 10$, $r_{\pi1} = r_{\pi2} = 250 \, \Omega$, $r_o = 1.5 \, \text{k} \Omega$, and $r_{\mu} = \infty$. 
For the amplifier in Fig. 10.18(a), determine the value of the feedback resistor $R_F$ so that the closed-loop voltage gain $A_f$ is 25% of the open-loop voltage gain $A$. The DC bias currents of the transistors are $I_{C1} = 0.5$ mA, $I_{C2} = 1$ mA, and $I_{C3} = 5$ mA. The transistor parameters are $h_{fe} = 100$, $r_o = 25$ kΩ, and $r_{\mu} = \infty$. Use the techniques of feedback analysis to calculate (a) the input resistance $R_{if}$, (b) the output resistance $R_o$, and (c) the closed-loop voltage gain $A_f$.

The AC equivalent circuit of a feedback amplifier is shown in Fig. 10.18(a). The DC bias currents of the transistors are $I_{C1} = 0.5$ mA, $I_{C2} = 1$ mA, and $I_{C3} = 5$ mA. The transistor parameters are $h_{fe} = 100$, $r_o = 25$ kΩ, and $r_{\mu} = \infty$. If $R_D1 = 1.5$ kΩ, then $R_D2 = 1$ kΩ, $R_{SR} = 500$ Ω, $R_F = 5$ kΩ, and $C_F = 20$ pF. Determine (a) the voltage gain without feedback $A = v_o/v_s$, (b) the voltage gain with feedback $A_f$, and (c) the feedback capacitor $C_F$ to limit the high frequency $f_H = 50$ kHz.
10.23 The MOS amplifier shown in Fig. P10.23 is biased to have the following small-signal MOS parameters: \( g_{m1} = g_{m2} = 1 \text{ mA/V}, r_{o1} = r_{o2} = 20 \text{ k}\Omega, g_{m3} = g_{m4} = 1.2 \text{ mA/V}, \) and \( r_{o3} = r_{o4} = 22 \text{ k}\Omega. \) If \( C_F = 6 \text{ nF} \) and \( C_1 = 4 \text{ nF}, \) determine (a) the voltage gain without feedback \( A = \frac{v_o}{v_s}, \) (b) the voltage gain with feedback \( A_f, \) and (c) the high cutoff frequency \( f_H. \)

**FIGURE P10.23**

![MOS Amplifier Diagram](image)

10.24 A transconductance amplifier with negative feedback is shown in Fig. P10.24. The amplifier has an open-loop transconductance of \( A = 50 \times 10^{-3} \text{ A/V}, \) an input resistance of \( R_i = 25 \text{ k}\Omega, \) and an output resistance of \( R_o = 50 \text{ k}\Omega. \) The feedback circuit has \( R_F = 2.5 \text{ k}\Omega. \) The source resistance is \( R_s = 1 \text{ k}\Omega, \) and the load resistance is \( R_L = 500 \text{ \Omega.} \) Determine (a) the input resistance \( R_{if} = \frac{v_s}{i_s}, \) (b) the output resistance \( R_{of}, \) and (c) the overall transconductance gain \( A_f = \frac{i_o}{v_s}. \)

**FIGURE P10.24**

![Transconductance Amplifier Diagram](image)

10.25 Repeat Prob. 10.24, ignoring the loading effects of the feedback network and the load resistance; that is, assume \( R_s = 0 \) and \( R_L = 0. \)

10.26 Use the techniques of feedback analysis to determine the input and output resistance of the CE transistor amplifier in Fig. P10.26. The circuit parameters are \( R_s = 500 \text{ \Omega, } R_E = 250 \text{ \Omega, } R_2 = 15 \text{ k}\Omega, R_1 = 5 \text{ k}\Omega, R_C = 5 \text{ k}\Omega, \) and \( R_L = 10 \text{ k}\Omega. \) The \( \pi \)-model parameters are \( r_o = 25 \text{ k}\Omega, h_{fe} = 150, \) \( r_\pi = 250 \text{ \Omega, } g_m = 0.3876 \text{ A/V, and } r_{\mu} = \infty. \)
10.27 Use the techniques of feedback analysis to calculate the input resistance $R_{if}$, the output resistance $R_{of}$, and the closed-loop transconductance gain $A_f$ of the amplifier in Fig. 10.28. The DC bias currents of the transistors are $I_{C1} = 0.1 \, \text{mA}$, $I_{C2} = 0.5 \, \text{mA}$, and $I_{C3} = 2 \, \text{mA}$. The transistor parameters are $h_{fe} = h_{fe1} = h_{fe2} = h_{fe3} = 150$, $r_o = 25 \, \text{k}\Omega$, and $r_{\mu} = \infty$.

10.28 The AC equivalent circuit of a feedback amplifier is shown in Fig. P10.28. The circuit values are $R_{C1} = 2.5 \, \text{k}\Omega$, $R_{C2} = 5 \, \text{k}\Omega$, $R_{C3} = 1.5 \, \text{k}\Omega$, $R_{E1} = 100 \, \Omega$, $R_{E2} = 100 \, \Omega$, $R_F = 750 \, \Omega$, and $R_s = 0$. The transistor parameters are $h_{fe} = 100$, $r_i = 2.5 \, \text{k}\Omega$, $r_o = 25 \, \text{k}\Omega$, and $r_{\mu} = \infty$. Use the techniques of feedback analysis to calculate (a) the input resistance $R_{if}$, (b) the output resistance $R_{of}$, and (c) the closed-loop voltage gain $A_f$.

10.29 For the amplifier in Fig. 10.28 determine the value of the feedback resistor $R_F$ so that the closed-loop transconductance gain $A_f$ is $25\%$ of the open-loop transconductance gain $A$. The DC bias currents of the transistors are $I_{C1} = 0.5 \, \text{mA}$, $I_{C2} = 1 \, \text{mA}$, and $I_{C3} = 5 \, \text{mA}$. The transistor parameters are $h_{fe} = h_{fe1} = h_{fe2} = h_{fe3} = 100$, $r_o = 25 \, \text{k}\Omega$, and $r_{\mu} = \infty$. 

---

**FIGURE P10.26**

**FIGURE P10.28**
10.30 For the amplifier in Fig. P10.30, determine the value of the feedback resistor $R_F$ so that the closed-loop transconductance gain $A_f$ is $5 \text{ mA/V}$. The transistor parameters are $h_{fe} = 100$, $r_e = 250 \Omega$, $r_o = 50 \Omega$, and $r_{\mu} = \infty$. Assume $R_t = 2 \text{ M}\Omega$, $R_o = 50 \Omega$, and $A = 10^5 \text{ V/V}$.

**FIGURE P10.30**

10.31 The MOS amplifier shown in Fig. P10.31 is biased to have the following small-signal MOS parameters: $g_{m1} = 1.2 \text{ mA/V}$, $r_{o1} = 25 \text{ k}\Omega$, $g_{m2} = 1.6 \text{ mA/V}$, and $r_{o2} = 25 \text{ k}\Omega$. If $R_D = 1.5 \text{ k}\Omega$, then $R_{SR1} = 500 \Omega$, $R_{SR2} = 2 \text{ k}\Omega$, and $R_F = 8 \text{ k}\Omega$. Determine (a) the voltage gain without feedback $A = i_o/v_s$, (b) the voltage gain with feedback $A_f$, and (c) the feedback capacitor $C_F$ to limit the high frequency $f_H = 50 \text{ kHz}$.

**FIGURE P10.31**

10.8 *Shunt-Shunt Feedback*

10.32 The inverting op-amp amplifier shown in Fig. 10.9(c) has $R_F = 40 \text{ k}\Omega$ and $R_1 = 10 \text{ k}\Omega$. The op-amp has an input resistance of $R_i = 5 \text{ M}\Omega$, an output resistance of $R_o = 50 \Omega$, and an open-loop voltage gain of $A = 2 \times 10^5 \text{ V/V}$. The load resistance is $R_L = 15 \text{ k}\Omega$. Determine (a) the input resistance seen by the source $R_{in} = v_s/i_s$, (b) the output resistance $R_{of}$, (c) the closed-loop transresistance $A_f = v_o/i_i$, and (d) the overall voltage gain $A_{vf} = v_o/v_s$. Assume source resistance $R_s = 1 \text{ k}\Omega$.

10.33 A transresistance amplifier with negative feedback is shown in Fig. P10.33. The open-loop transresistance is $A = 750 \text{ k}\Omega$, the input resistance is $R_i = 5.5 \text{ k}\Omega$, and the output resistance is $R_o = 500 \Omega$. The feedback circuit has $R_F = 47 \text{ k}\Omega$. The source has a resistance of $R_s = 0$. Determine (a) the input resistance seen by the source $R_{in} = v_s/i_s$, (b) the output resistance $R_{of}$, and (c) the overall voltage gain $A_{vf} = v_o/v_s$. Assume load resistance $R_L = 1 \text{ k}\Omega$. 
10.34 Repeat Prob. 10.33, ignoring the loading effects of the feedback network and the load resistance and assuming that the source has a resistance of $R_s = 1.5 \, \text{k} \Omega$.

10.35 Use the techniques of feedback analysis to calculate the input resistance $R_{if}$, the output resistance $R_{of}$, and the closed-loop transresistance gain $A_f$ of the amplifier in Fig. 10.38. The circuit values are $R_{C1} = 5 \, \text{k} \Omega$, $R_E = 2.5 \, \text{k} \Omega$, $R_{C2} = 5 \, \text{k} \Omega$, $R_F = 4 \, \text{k} \Omega$, and $R_s = 200 \, \Omega$. The transistor parameters are $h_{fe} = 150$, $r_m = 2 \, \text{k} \Omega$, $r_o = 25 \, \text{k} \Omega$, and $r_{\mu} = \infty$. Use the techniques of feedback analysis to calculate (a) the input resistance $R_{if}$, (b) the output resistance $R_{of}$, and (c) the closed-loop voltage gain $A_f$.

10.36 The AC equivalent circuit of a feedback amplifier is shown in Fig. 10.38. The circuit values are $R_{C1} = 5 \, \text{k} \Omega$, $R_E = 2.5 \, \text{k} \Omega$, $R_{C2} = 5 \, \text{k} \Omega$, $R_F = 4 \, \text{k} \Omega$, and $R_s = 200 \, \Omega$. The transistor parameters are $h_{fe} = 100$, $r_m = 2 \, \text{k} \Omega$, $r_o = 25 \, \text{k} \Omega$, and $r_{\mu} = \infty$. Use the techniques of feedback analysis to calculate (a) the input resistance $R_{if}$, (b) the output resistance $R_{of}$, and (c) the closed-loop voltage gain $A_f$.

10.37 The AC equivalent circuit of the amplifier in Fig. P10.37 has $R_1 = 6.6 \, \text{k} \Omega$, $R_2 = 1 \, \text{k} \Omega$, $R_C = 1 \, \text{k} \Omega$, $R_E = 100 \, \Omega$, $R_s = 500 \, \Omega$, $R_F = 8 \, \text{k} \Omega$, and $R_L = 5 \, \text{k} \Omega$. The transistor parameters are $h_{fe} = 100$, $r_m = 581 \, \Omega$, and $r_o = 22.5 \, \text{k} \Omega$. Use the techniques of feedback analysis to calculate (a) the input resistance $R_{if}$, (b) the output resistance $R_{of}$, and (c) the closed-loop voltage gain $A_f$.

10.38 The AC equivalent circuit of a feedback amplifier is shown in Fig. P10.38. The circuit values are $R_s = 1 \, \text{k} \Omega$, $R_C = 10 \, \text{k} \Omega$, and $R_F = 24 \, \text{k} \Omega$. The transistor parameters are $h_{fe} = 150$, $r_m = 500 \, \Omega$, $r_o = 25 \, \text{k} \Omega$, and $r_{\mu} = \infty$. Use the techniques of feedback analysis to calculate (a) the input resistance $R_{if}$, (b) the output resistance $R_{of}$, and (c) the closed-loop voltage gain $A_f$. 

FIGURE P10.33

(a) Amplifier

(b) Feedback circuit

FIGURE P10.37
The AC equivalent circuit of a feedback amplifier is shown in Fig. P10.39. The circuit values are $R_D = 10 \, \text{k}\Omega$, $R_F = 50 \, \text{k}\Omega$, and $R_s = 1 \, \text{k}\Omega$. The transistor parameters are $g_m = 1 \, \text{mA/V}$, $r_d = 50 \, \text{k}\Omega$, and $r_o = 25 \, \text{k}\Omega$. Use the techniques of feedback analysis to calculate (a) the input resistance $R_{in}$, (b) the output resistance $R_{of}$, and (c) the closed-loop voltage gain $A_f$.

Determine the value of the feedback resistor $R_F$ so that the closed-loop voltage gain $A_f$ of the amplifier in Fig. 10.38 is 25% of the open-loop voltage gain $A$. The circuit values are $R_{C1} = 5 \, \text{k}\Omega$, $R_E = 2.5 \, \text{k}\Omega$, $R_{C2} = 5 \, \text{k}\Omega$, and $R_s = 200 \, \text{\mu}\Omega$. The transistor parameters are $h_{fe} = 150$, $r_m = 2 \, \text{k}\Omega$, $r_o = 25 \, \text{k}\Omega$, and $r_{\mu} = \infty$. Assume that the coupling capacitors can be considered as short-circuited at the operating frequency range.

Determine the value of the feedback resistor $R_F$ so that the closed-loop current gain $A_f$ of the amplifier in Fig. P10.41 is 10% of the open-loop voltage gain $A$. The transistor parameters are $h_{fe} = 150$, $r_m = 2 \, \text{k}\Omega$, $r_o = 25 \, \text{k}\Omega$, and $r_{\mu} = \infty$. Assume that the coupling capacitors can be considered as short-circuited at the operating frequency range.
10.42 The MOS amplifier shown in Fig. P10.42 is biased to have the following small-signal MOS parameters: $g_{m1} = 1.2 \text{ mA/\text{V}}$ and $r_{o1} = 25 \text{ k\Omega}$. If $R_F = 100 \text{ k\Omega}$, then $R_D = 2 \text{ k\Omega}$ and $C_F = 10 \text{ nF}$. Determine (a) the voltage gain without feedback $A = v_o/i_s$, (b) the voltage gain with feedback $A_F$, and (c) the high cutoff frequency $f_H$.

**FIGURE P10.42**

![Mos Amplifier Diagram](image)

10.43 The MOS amplifier shown in Fig. P10.43 is biased to have the following small-signal MOS parameters: $g_{m1} = 1.2 \text{ mA/\text{V}}$, $r_{o1} = 25 \text{ k\Omega}$, $g_{m2} = 1.6 \text{ mA/\text{V}}$, and $r_{o2} = 25 \text{ k\Omega}$. If $R_F = 100 \text{ k\Omega}$, then $R_1 = 400 \text{ k\Omega}$, $R_s = 1 \text{ k\Omega}$, and $C_F = 10 \text{ nF}$. Determine (a) the voltage gain without feedback $A = v_o/v_s$, (b) the voltage gain with feedback $A_F$, and (c) the high cutoff frequency $f_H$.

**FIGURE P10.43**

![Mos Amplifier Diagram](image)

10.44 The MOS amplifier shown in Fig. P10.44 is biased to have the following small-signal MOS parameters: $g_{m1} = g_{m2} = 1.2 \text{ mA/\text{V}}$ and $r_{o1} = r_{o2} = 20 \text{ k\Omega}$. If $R_D = 2 \text{ k\Omega}$, then $C_F = 6 \text{ nF}$, and $C_1 = 2 \text{ nF}$. Determine (a) the voltage gain without feedback $A = v_o/i_s$, (b) the voltage gain with feedback $A_F$, and (c) the high cutoff frequency $f_H$. 
The current amplifier in Fig. P10.45 has negative feedback. The open-loop current gain is $A = 60$, the input resistance is $R_i = 500 \ \Omega$, and the output resistance is $R_o = 27 \ \Omega$. The feedback circuit has $R_F = 20 \ \Omega$ and $R_1 = 2.5 \ \Omega$. The source resistance is $R_s = 500 \ \Omega$, and the load resistance is $R_L = 100 \ \Omega$. Determine (a) the input resistance $R_{if} = \frac{v_s}{i_s}$, (b) the output resistance $R_{of}$, and (c) the overall current gain $A_f = \frac{i_o}{i_s}$.

Repeat Prob. 10.45, ignoring the loading effects of the feedback network and the load resistance and assuming that the source has a resistance of $R_s = 5 \ \Omega$.

The CE amplifier in Fig. 10.48 has $R_F = 10 \ \Omega$, $R_1 = 6.6 \ \Omega$, $R_2 = 1 \ \Omega$, $R_{C1} = 5 \ \Omega$, $R_E = 500 \ \Omega$, $R_3 = 5 \ \Omega$, $R_4 = 10 \ \Omega$, $R_{C2} = 5 \ \Omega$, and $R_s = 200 \ \Omega$. The transistor parameters are $h_{fe} = 150$, $r_i = 258 \ \Omega$, and $r_o = 25 \ \Omega$. Use the techniques of feedback analysis to calculate (a) the input resistance $R_{if}$, (b) the output resistance $R_{of}$, and (c) the closed-loop voltage gain $A_f$.

Determine the value of the feedback resistor $R_F$ so that the closed-loop current gain $A_f$ of the amplifier in Fig. 10.48 with shunt-series feedback is 25% of the open-loop current gain $A$. The amplifier has $R_1 = 6.6 \ \Omega$, $R_2 = 2 \ \Omega$, $R_{C1} = 5 \ \Omega$, $R_E = 500 \ \Omega$, $R_3 = 5 \ \Omega$, $R_4 = 10 \ \Omega$, $R_{C2} = 5 \ \Omega$, and $R_s = 0$. The transistor parameters are $h_{fe} = 100$, $r_i = 2.58 \ \Omega$, and $r_o = 25 \ \Omega$. 
10.49 Determine the value of the feedback resistor $R_F$ so that the midfrequency closed-loop current gain $A_f$ of the amplifier in Fig. P10.49 is 10% of the open-loop current gain $A$. The transistor parameters are $h_{fe} = 100$, $r_{se} = 2.58 \, \text{k} \Omega$, and $r_o = 25 \, \text{k} \Omega$. Assume that the coupling capacitors can be considered as short-circuited at the operating frequency range.

**FIGURE P10.49**

10.50 The MOS amplifier shown in Fig. P10.50 is biased to have the following small-signal MOS parameters: $g_{m1} = 1.2 \, \text{mA/V}$, $r_{o1} = 25 \, \text{k} \Omega$, $g_{m2} = 1.6 \, \text{mA/V}$, and $r_{o2} = 25 \, \text{k} \Omega$. If $R_D = 1.5 \, \text{k} \Omega$, then $R_{SR1} = 500 \, \text{k} \Omega$, $R_{SR2} = 2 \, \text{k} \Omega$, and $R_F = 8 \, \text{k} \Omega$. Determine (a) the voltage gain without feedback $A = i_o/v_s$, (b) the voltage gain with feedback $A_f$, and (c) the feedback capacitor $C_F$ to limit the high frequency $f_H = 50 \, \text{kHz}$.

**FIGURE P10.50**

10.11 Stability Analysis

10.51 The open-loop gain of an amplifier is given by

$$A(s) = \frac{6}{s^2 + 2s - 30}$$

Determine the closed-loop response due to a step input. Assume feedback factor $\beta(s) = 1$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
10.52 The open-loop gain of an amplifier is given by

\[ A(s) = \frac{s}{s^2 + 100} \]

Determine the closed-loop response due to a step input. Assume feedback factor \( \beta(s) = 1 \).

10.53 The loop gain of a feedback amplifier is given by

\[ T_L(j\omega) = \frac{A_o}{j\omega(1 + j\omega T_1)(1 + j\omega T_2)} \]

where \( A_o \) is a gain constant and \( T_1 \) and \( T_2 \) are time constants. Determine the phase margin and gain margin of the amplifier.

10.54 The loop gain of a feedback amplifier is given by

\[ T_L(j\omega) = \frac{A_o}{j\omega[(j\omega)^2 K_2 + j\omega K_1 + 1]} \]

where \( A_o \) is a gain constant and \( K_1 \) and \( K_2 \) are constants. Determine the phase margin and gain margin of the amplifier.

10.55 The loop gain of a feedback amplifier is given by

\[ T_L(s) = \frac{10}{s(s + 1)(s + 2)} \]

Determine the phase margin and gain margin of the amplifier.

10.56 If the phase margin of an amplifier is \( PM = 40^\circ \) and the magnitude of the open-loop gain is \( |A(j\omega)| = 50 \), find the magnitude of the closed-loop gain \( |A_f(j\omega)| \).

10.57 The open-loop gain of an amplifier has break frequencies at \( f_{p1} = 10 \text{ kHz}, f_{p2} = 100 \text{ kHz}, \) and \( f_{p3} = 1 \text{ MHz} \). The low-frequency gain is \( A_o = 250 \), and the feedback factor is \( \beta = 0.9 \). Calculate the gain margin \( GM \) and the phase margin \( PM \).

10.58 For the amplifier in Prob. 10.57, determine the frequency at the dominant pole so that the phase margin is \( PM = 45^\circ \).

10.59 The feedback amplifier in Fig. 10.60(a) has \( g_m = 40 \times 10^{-3}, R_1 = 3.5 \text{ k}\Omega, C_1 = 10 \text{ pF}, R_o = 24 \text{ k}\Omega, \) and \( C_o = 5 \text{ pF} \).

a. Calculate the two pole frequencies for \( C_x = 0 \) and the value of feedback capacitance \( C_x \) so that the frequency of the dominant pole is \( f_D = 1.5 \text{ kHz} \).

b. Use PSpice/SPICE to plot the closed-loop transimpedance \( A_f \) and the input impedance \( Z_i \) against frequency.

10.60 The equivalent circuit of an output stage is shown in Fig. 10.60(c), where \( R_1 = 22 \text{ k}\Omega \) and \( C_1 = 18 \text{ pF} \).

a. Find the break frequency \( f_p \).

b. Find the additional capacitance \( C_x \) that will move the break frequency to \( f_D = 50 \text{ kHz} \).

10.61 The open-loop gain of the multistage CE amplifier in Fig. 8.68 has break frequencies at \( f_{p1} = 805 \text{ kHz}, f_{p2} = 9.6 \text{ MHz}, f_{p3} = 13 \text{ MHz}, \) and \( f_{p4} = 61 \text{ MHz} \). The midfrequency gain is \( A_{mid} = 9594 \). Determine the value of compensating capacitance \( C_x \) that will give a closed-loop phase margin of \( 45^\circ \) with a resistive feedback of up to \( \beta = 1 \).
10.62 The open-loop gain of the multistage MOSFET amplifier in Fig. 7.61(a) has break frequencies at $f_{p1} = 13.75$ kHz, $f_{p2} = 248$ MHz, and $f_{p3} = 400$ MHz. The midfrequency gain is $A_{mid} = 311$. Determine the value of compensating capacitance $C_x$ that will give a closed-loop phase margin of $45^\circ$ with a resistive feedback of $\beta = 1$.

10.63 The open-loop gain of the CE amplifier in Fig. 8.53 has a midfrequency gain of $A_{mid} = -300$ V/V. Determine the values of compensating capacitance $C_F$ and resistance $R_F$ in a shunt-shunt feedback network to give a midfrequency closed-loop gain of $A_f = -30$ V/V. The low break frequency should be less than 1 kHz. Assume source resistance $R_s = 250 \Omega$.

10.64 The open-loop gain of the amplifier in Example 10.12 has break frequencies at $f_{p1} = 10$ kHz, $f_{p2} = 100$ kHz, and $f_{p3} = 1$ MHz. The low-frequency gain is $A_o = 100$ A/A, and the emitter resistance is $R_E = 200 \Omega$. Determine the values of compensating capacitance $C_F$ and resistance $R_F$ (a) to give a low-frequency closed-loop gain of $A_f = 20$ A/A and cancel the pole $f_{p1} = 10$ kHz and (b) to add a pole of $f_p = 10$ MHz and cancel the pole $f_{p1} = 10$ kHz.

10.65 The transistor $Q_4$ of the amplifier output stage as shown in Fig. 10.60(a) has an input resistance $r_{be4} = 20$ k$\Omega$ and an input capacitance $C_{ij4} = 40$ pF.
   a. Find the break frequency $f_p$.
   b. Find the value of additional compensating capacitance $C_x$ which will modify the dominant pole and will move the break frequency to $f_D = 25$ kHz.

10.66 The differential amplifier shown in Fig. 10.60(a) has an open-loop gain with break frequencies at $f_{p1} = 50$ kHz, $f_{p2} = 1$ MHz, and $f_{p3} = 10$ MHz. The gain stage of the amplifier has the equivalent circuit shown in Fig. 10.60(c) whose parameters are $g_m = 10$ mA/V, $C_i = 20$ pF, and $C_o = 5$ pF. Determine the value of compensating capacitance $C_x$ by pole splitting that will give a closed-loop phase margin of $45^\circ$ with a resistive feedback of up to $\beta = 1$.

10.67 The amplifier shown in Fig. P10.67 has an open-loop gain with break frequencies at $f_{p1} = 150$ kHz and $f_{p2} = 1$ MHz. The parameters of $Q_4$ are $g_m3 = 1$ mA/V, and the output capacitance of $C_{o3} = 10$ pF. The input capacitance of $Q_2$ is $C_{i2} = 10$ pF. Determine the value of compensating capacitance $C_x$ by pole splitting that will give a closed-loop phase margin of $50^\circ$ with a resistive feedback of up to $\beta = 1$.

**FIGURE P10.67**

The amplifier shown in Fig. P10.67 has an open-loop gain with break frequencies at $f_{p1} = 150$ kHz and $f_{p2} = 1$ MHz. The parameters of $Q_4$ are $g_m3 = 1$ mA/V, and the output capacitance of $C_{o3} = 10$ pF. The input capacitance of $Q_2$ is $C_{i2} = 10$ pF. Determine the value of compensating capacitance $C_x$ by pole splitting that will give a closed-loop phase margin of $50^\circ$ with a resistive feedback of up to $\beta = 1$.
10.68 The open-loop gain of the amplifier shown in Fig. 10.61(a) has break frequencies at $f_{p1} = 50$ kHz, $f_{p2} = 1$ MHz, and $f_{p3} = 10$ MHz. The low-frequency gain is $A_o = 100$ A/A, and the emitter resistance $R_E = 400$ Ω. Determine the values of compensating capacitance $C_F$ and resistance $R_F$ by feedback path modification (a) to give a low-frequency closed-loop gain of $A_f = 20$ A/A and to cancel the pole $f_{p1} = 100$ kHz and (b) to add a pole of $f_p = 50$ MHz and to cancel the pole $f_{p1} = 50$ kHz.

10.69 A common-source amplifier with shunt-shunt feedback is shown in Fig. P10.69. The capacitor $C_F$ and resistor $R_F$ form the feedback network. The voltage gain without feedback is $A = \left| A_v \right| = v_o/v_s = 20$.

a. Design the feedback network to meet the following specifications: The bandwidth with feedback must be increased by 10 times; that is, $BW_f = 10 \cdot BW$. The voltage gain with feedback is $\left| A_f \right| = v_o/v_s = 5$.

b. Use PSpice to verify your results.

**FIGURE P10.69**
Learning Outcomes

After completing this chapter, students should be able to do the following:

- Describe the types of power amplifiers and their transfer characteristics.
- Select a power amplifier to meet certain requirements.
- Analyze and design power amplifiers.
- Design power amplifiers to meet certain specifications.
- Apply the methods for eliminating crossover distortion and reducing offsets and nonlinearities on the output voltage.
- Apply an active current source to bias the output stage.
- Identify and describe the internal structure of IC power op-amps.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$v_i, v_o$</td>
<td>Input and output voltages of an amplifier</td>
</tr>
<tr>
<td>$I_p, V_p, V_{pp}$</td>
<td>Peak current, peak voltage, and peak-to-peak voltage of a transistor</td>
</tr>
<tr>
<td>$I_C, I_D$</td>
<td>Collector and drain currents of a transistor</td>
</tr>
<tr>
<td>$I_Q, I_R$</td>
<td>Quiescent and reference currents of an amplifier</td>
</tr>
<tr>
<td>$\alpha, k$</td>
<td>Conduction angle and duty cycle of a transistor</td>
</tr>
</tbody>
</table>
11.1 Introduction

The amplifiers in Chapters 7 and 8 were operated as input and/or intermediate stages to obtain a large voltage gain or current gain. The transistors within the amplifiers were operated in the active region so that their small-signal models were valid. These stages were not required to provide appreciable amounts of power, and the distortion of the output signal was negligible because the transistors operated in the active region.

The requirements for the output stages of audio-frequency power amplifiers are significantly different from those of small-signal low-power amplifiers. An output stage must deliver an appreciable amount of power and be capable of driving low-impedance loads such as loudspeakers. The distortion of the output signal must also be low. Distortion is measured by a quality factor known as total harmonic distortion (THD), which is the rms value of the harmonic components of the output signal, excluding the fundamental, expressed as a percentage of the rms of the fundamental component. The THD of high-fidelity audio amplifiers is usually less than 0.1%.

The DC power requirement of an audio amplifier must be as small as possible so that the efficiency of the amplifier is as high as possible. Increasing the efficiency of the amplifier reduces the amount of power dissipated by the transistors and the amount of power drawn from DC supplies, thereby reducing the cost of the power supply and prolonging the life of batteries in battery-powered amplifiers. Also, a low DC power requirement helps to keep the internal junction temperature of the transistors well below the maximum allowable temperature (in the range of 150°C to 200°C for silicon devices). As a result, a low DC power requirement minimizes the size of heat sinks and can eliminate the need for cooling fans. Therefore, an output stage should deliver the required amount of power to the load efficiently.

11.2 Classification of Power Amplifiers

Power amplifiers [1, 2] are generally classified into six types: A, B, AB, and C for analog designs and classes D and E for switching designs. This classification is based on the shape of the drain current in a MOSFET or collector current waveform for a BJT in response to a sinusoidal input signal. In an analog amplifier, the input signal to the amplifying devices causes a proportional output current to flow out of the output terminal, whereas in switching amplifiers the output current of the amplifying device is pulsating type. The output current and the load power come from the DC power supply. The load of power amplifiers is generally a speaker. The waveforms of the collector (or the drain) currents for various types of amplifiers are shown in Fig. 11.1. Table 11.1 shows the conduction angle and duty cycle of the input
signal for analog circuits. The duty cycle for the class D and E amplifiers is generally variable up to 50% of the switching period.

In a class A amplifier, the DC biasing collector current $I_C$ of a transistor is higher than the peak amplitude of the AC output current $I_p$. Thus, the transistor in a class A amplifier conducts during the entire cycle of the input signal, and the conduction angle is $\theta = \omega t = 360^\circ$. That is, the collector current of a

*FIGURE 11.1* Output voltages for various classes of amplifiers
A class E amplifier, shown in Fig. 11.1(f), is similar to the class D amplifier, except that a capacitor \( C_p \) is connected across the transistor. As a result, there are \( LC \) circuits when the transistor is switched on: one parallel \( L-C_p \) circuit and one series \( LC \) circuit via the load. When the transistor is switched on, the inductor \( L \) supplies its stored energy to the series \( LC \) circuit and the load. When the transistor is switched off, the inductor \( L \) replenishes its energy from the supply voltage.

In a class E amplifier, the transistor is biased at zero DC current and conducts for only a half-cycle of the input signal, with a conduction angle of \( \theta = 180^\circ \); that is, \( I_C = I_p \sin \omega t \). The waveform of the collector current for a class E amplifier is shown in Fig. 11.1(f). The negative halves of the sinusoid are provided by another transistor that also operates in the class E mode and conducts during the alternate half-cycles.

In a class AB amplifier, the transistor is biased at a nonzero DC current that is much smaller than the peak amplitude of the AC output current. The transistor conducts for slightly more than half a cycle of the input signal. The conduction angle is greater than \( 180^\circ \) but much less than \( 360^\circ \); that is, \( 180^\circ < \theta << 360^\circ \). Thus, \( I_C = I_p \sin \omega t \) and \( I_C < I_p \). The waveform of the collector current for a class AB amplifier is shown in Fig. 11.1(c). The negative halves of the sinusoid are provided by another transistor that also operates in the class AB mode and conducts for an interval slightly greater than the negative half-cycle. The currents from the two transistors are combined to form the load current. Both transistors conduct for an interval near the zero crossings of the input signal.

In a class C amplifier, as shown in Fig. 11.1(d) the transistor conducts for an interval shorter than a half-cycle. The conduction angle of the transistor is less than \( 180^\circ \); that is, \( \theta < 180^\circ \). The negative halves of the collector current are provided by another transistor. The collector current is of pulsating type and is much more distorted than the current generated by other classes of amplifier. The nonlinear distortion can be filtered out by passing this output through a parallel \( LC \)-resonant circuit. The resonant circuit is tuned to the frequency of the input signal and acts as a band-pass filter, giving an output voltage proportional to the amplitude of the fundamental component of the current waveform. Class C amplifiers are normally used in radio frequency applications.

A class D amplifier, shown in Fig. 11.1(e), operates transistors as switches, which are either completely turned on or completely turned off. The transistor output is connected to the supply voltage via a large inductor \( L \) and also to the load via a serial \( LC \) circuit (not shown). The output voltage is a pulsed waveform in response to a pulse-width modulation (PWM) input voltage. When the transistor is off, the inductor \( L \) supplies its stored energy to the series \( LC \) circuit and the load. When the transistor is switched on, the inductor \( L \) replenishes its energy from the supply voltage.

A class E amplifier, shown in Fig. 11.1(f), is similar to the class D amplifier, except that a capacitor \( C_p \) is connected across the transistor. As a result, there are \( LC \) circuits when the transistor is switched on: one parallel \( L-C_p \) circuit and one series \( LC \) circuit via the load.

Class A and class B amplifiers are commonly used in audio-frequency applications. Although there are many other types of amplifiers, we will consider the following kinds: emitter followers, class A amplifiers, class B push-pull amplifiers, complementary class AB push-pull amplifiers, quasi-complementary class AB push-pull amplifiers, transformer-coupled class AB push-pull amplifiers, and power op-amps.
**KEY POINTS OF SECTION 11.2**

- A power amplifier can be classified into one of six groups—A, B, AB, C, D, or E—depending on the conduction interval of the transistors used in the amplifier.
- To avoid distortion due to clipping, the maximum peak collector current of a transistor is limited to a specified value.

### 11.3 Power Transistors

The amplifying devices can be either MOSFETs or BJTs. The MOSFETs offer superior performances over the BJTs [3–5]. They require virtually zero input current, and they have faster switching times, no secondary breakdown, and stable gain and response time over a wide temperature range. The BJT current gain $\beta_F$ can vary widely with temperature, and the variation of the MOSFET transconductance $g_m$ with temperature is less than the variation of the BJT current gain $\beta_F$. As a result, power MOSFETs are replacing power BJTs in most applications.

There are no significant differences in the internal construction of small-signal and power BJTs. Although the mechanism of operation of power MOSFETs is same as that of small-signal MOSFETs, the power MOSFETs differ in their internal construction from the small-signal transistors in order to have more channel width for obtaining more current-carrying capability. To achieve a large channel width with good characteristics, power MOSFETs are fabricated with a repetitive pattern of small cells operating in parallel. The voltage ratings of power MOSFETs range from 50 V to 100 V, with current ratings from 10 A to 30 A.

There are two basic power MOSFET structures. The first is called a DMOS device, which uses a double-diffusion process; its cross section is shown in Fig. 11.2(a). It has two parallel current paths from the drain to the source. The $p$-substrate region is diffused deeper than the $n^+$-source. The $n$-drift region must be moderately doped so that the drain breakdown voltage is sufficiently large and the thickness of the $n$-drift region is made as thin as possible to minimize drain resistance. The second structure, as shown in Fig. 11.2(b), uses a vertical channel known as a VMOS structure. In this case, the $p$-substrate diffusion is performed over the entire surface, and a V-shaped groove is then formed, extending through the $n$-drift region. For a high efficiency, on-resistance $R_{on}$, which should be low, can be found from

$$R_{DS(on)} = R_{SC} + R_{CH} + R_{DC}$$  \hspace{1cm} (11.1)

where $R_{SC}$ = source terminal contact resistance

$R_{CH}$ = channel resistance

$R_{DC}$ = drain terminal contact resistance

The values of the contact resistances $R_{SC}$ and $R_{DC}$ are proportional to the semiconductor resistivity. From Eq. (7.3), we can find the channel resistance $R_{CH}$ in the linear region of operation as given by

$$R_{CH} = \frac{V_{DS}}{I_D} = \frac{L}{W\mu C_{ox}(V_{GS} - V_t)}$$  \hspace{1cm} (11.2)

This explains why the channel length should be low and the width should be high to reduce on-resistance. An increase in the drain current increases the power loss and the junction temperature, which in turn increases the threshold voltage $V_t$ and $R_{CH}$, thereby limiting the drain current.

For the sake of illustration of power amplifiers, we will show and analyze circuits using BJTs. The BJT analysis can be applied to MOSFET circuits by substituting $\beta_F = 0$ for the BJT current gain. Table 11.2 lists the circuit parameters for replacing a BJT by a MOSFET in a circuit.
TABLE 11.2 Circuit parameters of MOSFETs and BJTs

<table>
<thead>
<tr>
<th>Circuit Parameters</th>
<th>MOSFETs</th>
<th>BJTs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Supply voltage</td>
<td>Power supply ±$V_{DD}$</td>
<td>±$V_{CC}$</td>
</tr>
<tr>
<td>Output current</td>
<td>Drain current $i_D$</td>
<td>Collector current $i_C$</td>
</tr>
<tr>
<td>Driving voltage</td>
<td>Gate-source voltage $v_{GS}$</td>
<td>Base-emitter voltage $v_{BE}$</td>
</tr>
<tr>
<td>Input voltage</td>
<td>Gate voltage $v_G$</td>
<td>Base voltage $v_B$</td>
</tr>
<tr>
<td>Transistor input current</td>
<td>$i_G \approx 0$</td>
<td>Base current $i_B$</td>
</tr>
<tr>
<td>Current ratio</td>
<td>$\frac{i_D}{i_G} \approx \infty$</td>
<td>$\frac{i_C}{i_B} = \beta_F$</td>
</tr>
<tr>
<td>Diode-connected transistor</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
11.4 Class A Amplifiers

The common-source (common-emitter) amplifiers in Secs. 7.8 and 7.9 and the source and emitter followers in Secs. 8.8 and 8.9 fall into the class A category. The transistors of class A amplifiers are always turned on. Therefore, the transistor can become hot, with most of the power provided by the power supply being dissipated as heat. Although efficiency is poor (around 20%), the accuracy of the signal amplifications is quite high. We have analyzed the amplifiers to derive the small-signal parameters without any consideration to the circuit efficiency. For power amplifiers, efficiency is a major performance parameter.

11.4.1 Emitter Followers

An emitter follower is a class A amplifier; its circuit diagram is shown in Fig. 11.3(a). Section 8.9 discussed the characteristics of an emitter follower: a very low output impedance, a very high input impedance, and a voltage gain of almost unity at a large value of load resistance. The voltage gain and the DC current of transistor Q1 are affected by the values of load resistance $R_L$. The peak-to-peak voltage swing is less than $V_{CC}$. If the emitter resistance $R_E$ in Fig. 11.3(a) can be replaced by a current source, as shown in Fig. 11.3(b), the peak-to-peak voltage swing can be increased to a value larger than $V_{CC}$. The voltage gain can be maintained at almost unity even with a small load resistance, on the order of 100 $\Omega$.

![Emitter follower circuit diagram](image)
Transfer Characteristic

Assuming that the diode drop is $V_{D1} = 0.7$ V, the B-E drop of a transistor is $V_{BE} = 0.7$ V, and the transistor current gain $\beta_F >> 1$, the reference current $I_R$ can be approximated by $I_1$. Using KVL for the current source, we get

$$0 = I_R R_1 + V_{BE} - V_{CC}$$

which gives the reference current $I_R$ as

$$I_R = \frac{V_{CC} - V_{BE}}{R_1} \quad (11.3)$$

The output voltage is given by

$$v_O = v_1 - V_{BE} \quad (11.4)$$

which yields the transfer characteristic shown in Fig. 11.3(c). At $v_1 = 0$ (i.e., $v_O = -V_{BE}$), the characteristic has an offset voltage. Thus, for $I_{C1} > 0$, $Q_1$ will be on, and the positive peak value of output voltage is

$$+V_{O(max)} = V_{CC} - V_{CE(sat)} \quad (11.5)$$

$Q_1$ will be turned off when $i_{C1} = 0$ and $i_O = -I_R$. The output voltage can also be written as

$$v_O = i_O R_L = -I_R R_L$$

$$= \frac{V_{CC} - V_{BE} R_L}{R_1} \quad (11.6)$$

If the value of $R_L$ is less than $R_1$, the peak negative value of the output voltage will be less than the peak negative value of $V_{CC} - V_{CE(sat)}$ and the output will not be symmetrical. This will cause distortion (or clipping). Thus, the condition that avoids distortion and obtains the maximum output voltage swing is given by

$$R_L \geq R_1 \quad (11.7)$$

and the maximum voltage swing (peak to peak) without clipping is

$$V_{pp} = 2(V_{CC} - V_{CE(sat)}) \quad (11.8)$$

Signal Waveforms

Let us assume that the input is a sinusoidal voltage. If we neglect saturation voltage $V_{CE(sat)}$, the output voltage $v_O$ can swing from $-V_{CC}$ to $V_{CC}$, with the quiescent value being zero, as shown in Fig. 11.4(a). The collector–emitter (C-E) voltage will become $v_{CE1} = V_{CC} - v_O$, which is shown in Fig. 11.4(b). Assuming that $I_R = I_Q$ is selected to give the maximum output voltage swing, the collector current $i_{C1}$ is shown in Fig. 11.4(c). The instantaneous power dissipation in $Q_1$, shown in Fig. 11.4(d), is given by

$$P_{DI1} \approx v_{CE1} i_{C1} = V_{CC}(1 - \sin \omega t)I_R(1 + \sin \omega t) \quad (11.9)$$

and has an average value of $V_{CC} I_R \quad 2$.

Output Power and Efficiency

The power efficiency of the output stage of an emitter follower is defined by

$$\eta = \frac{\text{Load power } P_L}{\text{Supply power } P_S} \quad (11.10)$$
Assuming that the output voltage is sinusoidal with a peak value of $V_p$, the average load power will be

$$P_L = \frac{V_p^2}{2R_L} \quad (11.11)$$

The average current drawn by transistor $Q_1$ will be $I_R$, and thus the average power drawn from the positive supply will be $V_{CC}I_R$. Since the current in transistor $Q_2$ remains constant at $I_R$, the power drawn from the negative supply will also be $V_{CC}I_R$, if we neglect the power drawn by the current source consisting of the diode $D_1$ and resistor $R_1$. Thus, the total average supply power will be

$$P_S = 2V_{CC}I_R \quad (11.12)$$

From Eqs. (11.11) and (11.12), we get the power efficiency as

$$\eta = \frac{P_L}{4R_LV_{CC}I_R} = \frac{1}{4}\left(\frac{V_p}{R_LI_R}\right)\left(\frac{V_p}{V_{CC}}\right)$$

which will give the maximum efficiency when

$$V_p = V_{CC} = R_LI_R$$

That is, $\eta_{max} = 25\%$, which is rather low. In practice, the peak output voltage is limited to less than $V_{CC}$ to avoid transistor saturation and associated nonlinear distortion. Thus, the efficiency actually ranges from 10\% to 20\%. Emitter followers are generally used as output stages for high-frequency ($\approx 10$ MHz), low-power ($\leq 1$ W) amplifiers.

### Example 11.1

**Designing an emitter follower**

(a) Design the emitter follower of the circuit in Fig. 11.3(b). Assume $V_{CC} = 12$ V, $V_{BE} = 0.7$ V, $V_{CE(sat)} = 0.5$ V, $I_R = 5$ mA, and $R_L = 650$ $\Omega$. Assume identical transistors of current gain $h_{fe} = \beta_F = 100$. Note that $h_{fe}$ is the hybrid parameter and both $h_{fe}$ and $\beta_F$ represent the current gain of a BJT.

(b) Determine the critical value of load resistance to avoid clipping (or distortion).

(c) Calculate the peak-to-peak output voltage swing if $R_L = 650$ $\Omega$. 

(d) Calculate the peak-to-peak output voltage swing and the power efficiency $\eta$ if $R_L = 2.5 \, k\Omega$.

(e) Use PSpice/SPICE to plot the transfer function of the emitter follower for the values in part (a). The PSpice model parameters for the diode are

\[ IS=100E-15 \quad RS=16 \quad BV=100 \quad IBV=100E-15 \]

and those for the transistors are

\[ BF=100 \quad VA=100 \]

**SOLUTION**

(a) Determine the value of $R_1$ from Eq. (11.3):

\[ R_1 = \frac{V_{CC} - V_{BE}}{I_R} = \frac{12 \, V - 0.7 \, V}{5 \, mA} = 2260 \, \Omega \]

(b) From Eq. (11.7), we find the critical value of load resistance to avoid clipping:

\[ R_{L(crit)} = R_1 = 2260 \, \Omega \]

(c) For $R_L = 650 \, \Omega$, the negative peak output voltage is

\[ -V_{O(max)} = -I_R R_L = -5 \, mA \times 650 \, \Omega = -3.25 \, V \]

The positive peak output voltage is

\[ +V_{O(max)} = V_{CC} - V_{CE(sat)} = 12 \, V - 0.5 \, V = 11.5 \, V \]

Therefore, the peak-to-peak output voltage swing will be from $11.5 \, V$ to $-3.25 \, V$.

(d) For $R_L = 2.5 \, k\Omega$ (which is greater than $R_{L(crit)} = 2260 \, \Omega$), the negative peak output voltage will be limited to

\[ -V_{O(max)} = -V_{CC} + V_{CE(sat)} = -12 \, V + 0.5 \, V = -11.5 \, V \]

Therefore, the peak-to-peak output voltage swing will be from $-11.5 \, V$ to $+11.5 \, V$. Thus, Eq. (11.13) gives

\[ \eta = \frac{(11.5 \, V)^2}{4 \times 2.5 \, k\Omega \times 12 \, V \times 5 \, mA} = 22\% \]

(e) The emitter-follower circuit for PSpice simulation is shown in Fig. 11.5.
11.4.2 Basic Common-Emitter Amplifier

The simplest kind of class A amplifier is shown in Fig. 11.7(a). This configuration lacks biasing stability (i.e., emitter resistance $R_E$) and is not suitable for power amplifiers. However, we will use this circuit to derive the power efficiency of class A amplifiers. Let us assume that the nonlinearity introduced by the transistor is negligible so that the output signals will be sinusoidal for sinusoidal input signals. These assumptions will simplify the various power calculations. The waveforms of the collector current and voltages are shown in Fig. 11.7(b).

**Transfer Characteristic**

The input voltage $v_I$ is related to the collector current $I_C$ by

$$v_I = V_{BE} = V_T \ln \left( \frac{I_C}{I_S} \right)$$  \hspace{1cm} (11.14)
Using $i_C$ from Eq. (11.14), we can find the output voltage as

$$v_O = V_{CC} - R_C i_C = V_{CC} - R_C I_S \ln \left( \frac{v_I}{V_T} \right)$$  \hspace{1cm} (11.15)

Therefore, the transfer characteristic ($v_O$ versus $v_I$), which is shown in Fig. 11.8, is nonlinear.

**Output Power and Efficiency**

The average DC power required from the power supply is given by

$$P_S = V_{CC} I_C$$  \hspace{1cm} (11.16)

The average load or output power is given by

$$P_L = \left( \frac{V_p}{\sqrt{2}} \right) \left( \frac{I_p}{\sqrt{2}} \right) = \frac{V_p I_p}{2}$$  \hspace{1cm} (11.17)

$$= \frac{I_p R_L I_p}{2} = \frac{I_p^2 R_L}{2}$$  \hspace{1cm} (11.18)

**NOTE**  $\sqrt{2}$ converts the peak value to an rms value.

**FIGURE 11.7**  Basic common-emitter class A amplifier

**FIGURE 11.8**  Transfer characteristic of a common-emitter class A amplifier
where $V_p$ and $I_p$ are the peak values of the AC output voltage and current, respectively. Using the minimum and maximum values of the output signals, we can express $V_p$ and $I_p$ as

$$V_p = \frac{V_{CE(max)} - V_{CE(min)}}{2}$$

$$I_p = \frac{I_{C(max)} - I_{C(min)}}{2}$$

where $V_{CE}$ ideally extends over its full range. Using Eqs. (11.19) and (11.20), we can express Eq. (11.17) as

$$P_L = \frac{V_p I_p}{2} = \frac{(V_{CE(max)} - V_{CE(min)})(I_{C(max)} - I_{C(min)})}{8}$$

which will give the maximum load power $P_{L(max)}$ when $V_{CE(min)} = 0$, $I_{C(min)} = 0$, $V_{CE(max)} = V_{CC}$, and $I_{C(max)} = 2I_C$. Thus, Eq. (11.21) gives $P_{L(max)}$ as

$$P_{L(max)} = \frac{V_{CC}(2I_C)}{8} = \frac{V_{CC}I_C}{4}$$

The conversion efficiency, which is defined as the ratio of load power to the DC source power, is expressed as

$$\eta = \frac{P_L}{P_S} \times 100\%$$

Substituting Eqs. (11.16) and (11.21) into Eq. (11.23) gives the maximum efficiency as

$$\eta_{max} = \frac{(V_{CE(max)} - V_{CE(min)})(I_{C(max)} - I_{C(min)})}{8V_{CC}I_C}$$

which, for $V_{CE(min)} = 0$, $I_{C(min)} = 0$, $V_{CE(max)} = V_{CC}$, and $I_{C(max)} = 2I_C$, becomes

$$\eta_{max} = \frac{V_{CE(max)}I_{C(max)}}{8V_{CC}I_C} = \frac{V_{CC}(2I_C)}{8V_{CC}I_C} = \frac{1}{4} = 25\%$$

Hence, the maximum efficiency for a class A amplifier under ideal conditions is 25%. Although in practice the actual efficiency will be less than 25%, this percentage is often used as a guideline for determining the biasing requirement $I_C$. For example, if $V_{CC} = 30$ V and $P_{L(max)} = 50$ W, then

$$P_S = \frac{P_{L(max)}}{\eta_{max}} = \frac{50}{0.25} = 200 \text{ W}$$

and

$$I_C = \frac{P_S}{V_{CC}} = \frac{200}{30} = 6.67 \text{ A}$$

The quality of an amplifier is often measured by the *figure of merit* $F_m$, which is defined by

$$F_m = \frac{\text{Maximum collector dissipation}}{\text{Maximum output power}} = \frac{P_{C(max)}}{P_{L(max)}}$$

The maximum collector dissipation is given by

$$P_{C(max)} = \frac{V_{CC}I_C}{2}$$
Substituting Eqs. (11.22) and (11.27) into Eq. (11.26) yields
\[ F_m = \frac{V_{CC}I_c/2}{V_{CC}I_c/4} = 2 \]
(11.28)
Thus, the collector power dissipation is twice the maximum output power. That is, for a maximum output of 50 W, the collector must be able to dissipate at least 100 W. This requirement is the major disadvantage of class A amplifiers because it necessitates the use of a large and expensive heat sink to cool the transistors.

### 11.4.3 Common-Emitter Amplifiers with Active Load

Because of their high voltage gain, common-emitter stages are often used as output-stage drivers in integrated circuit design. A common-emitter stage is shown in Fig. 11.9(a). A current source consisting of Q2 and Q3 establishes the reference current \( I_R \), which is given by
\[ I_R = \frac{V_{CC} - V_{EB2} = V_{EB3}}{R_1} \]
(11.29)
With no load, \( R_L = \infty \), \( i_O = 0 \), and \( i_{C1} = I_Q = I_R \). Thus, the load current \( i_O \) is given by
\[ i_O = I_R - i_{C1} \]
(11.30)
where the collector current \( i_{C1} \) is related to the input voltage \( v_I \) by
\[ i_{C1} = I_S \exp\left(\frac{v_I}{V_T}\right) \]
(11.31)

**Transfer Characteristic**

The output voltage \( v_O \) is given by
\[ v_O = R_Li_O = R_L(I_R - i_{C1}) = R_L\left[I_R - I_S \exp\left(\frac{v_I}{V_T}\right)\right] \]
(11.32)
which is the transfer characteristic ($v_O$ versus $v_I$), shown in Fig. 11.9(b). When $i_{C1} = 0$, transistor $Q_1$ is cut off and transistor $Q_2$ supplies $I_Q$ to the load. The output voltage becomes

$$v_O = R_L i_R = R_L \frac{V_{CC} - V_{BE2}}{R_1} \tag{11.33}$$

If this value of $v_O$ is less than the maximum possible positive output voltage of $V_{CC} - V_{CE2(sat)}$, distortion will occur, as shown in Fig. 11.9(b). Thus, the condition for the maximum positive output voltage swing is $R_L \geq R_1$. As the input voltage $v_I$ is increased, the current in transistor $Q_1$ increases and $v_O$ becomes negative according to Eq. (11.32), until $Q_1$ saturates and the output voltage becomes

$$v_O = -V_{CC} + V_{CE1(sat)}$$

The transfer characteristic is basically a simple exponential and shows distortion on the output voltage. The input voltage required to produce the maximum output voltage swing is typically a few tens of millivolts or less.

### Output Power and Efficiency

All the equations derived for the emitter follower and the basic common-emitter circuits apply to this common-emitter stage. Thus, the maximum efficiency is $\eta_{max} = 25\%$, and the figure of merit is $F_m = 2$. The breakdown voltage rating of $Q_1$ and $Q_2$ must be $2V_{CC}$.

#### 11.4.4 Transformer-Coupled Load Common-Emitter Amplifier

The efficiency of an amplifier can be improved with a transformer-coupled load. A class A amplifier with a transformer-coupled load is shown in Fig. 11.10(a). The elimination of the collector resistance $R_C$, used for DC biasing in Fig. 11.7(a), accounts for the increase in efficiency. The transformer at the output stage provides an impedance match in order to transfer maximum power to the load. A load such as the impedance of a loudspeaker is usually very small, typically 4 $\Omega$ to 16 $\Omega$.

The voltage and current relations of the output transformer are

$$V_{pL} = \left( \frac{n_p}{n_s} \right) V_{sL} \quad \text{and} \quad I_{pL} = \left( \frac{n_s}{n_p} \right) I_{sL}$$

![Class A amplifier with transformer-coupled load](image)

**FIGURE 11.10** Class A amplifier with transformer-coupled load
where \( n_s \) and \( n_p \) = secondary and primary windings, respectively
\( V_{sL} \) and \( V_{pL} \) = secondary and primary voltages, respectively
\( I_{sL} \) and \( I_{pL} \) = the secondary and primary currents, respectively

The effective load resistance referred to the primary side can be found from

\[
R'_L = \left( \frac{V_{pL}}{I_{pL}} \right) = \left( \frac{n_p}{n_s} \right)^2 \left( \frac{V_{sL}}{I_{sL}} \right) = \left( \frac{n_p}{n_s} \right)^2 R_L
\]

(11.34)

The AC (dynamic) load line is determined by \( R'_L \). The DC (static) load line is almost vertical because of the very small primary resistance of the transformer. Assuming \( V_{CE(min)} = 0 \) in Eq. (11.19) and \( I_{C(min)} = 0 \) in Eq. (11.20), the peak values of the output voltage and current at the primary side of the transformer are

\[
V_p = \frac{V_{CE(max)} + V_{CC}}{2}
\]

(11.35)

\[
I_p = \frac{I_{C(max)} + I_C}{2}
\]

(11.36)

Equation (11.21) gives the maximum efficiency as

\[
\eta_{max} = \frac{V_{CE(max)}I_{C(max)} + V_{CC}I_C}{8V_{CC}I_C} = \frac{2V_{CC}(I_C)}{8V_{CC}I_C} = \frac{1}{2} = 50\%
\]

(11.37)

Hence, the maximum efficiency of a class A stage is doubled by using a transformer coupled to the load. The value of \( V_p \) for a transformer-coupled stage is \( V_{CC} \); for the basic common-emitter amplifier, it is only \( V_{CC}/2 \).

Equation (11.18) gives the maximum load power as

\[
P_{L(max)} = \frac{I_p^2R'_L}{2} = \frac{V_{CC}^2}{2R'_L}
\]

(11.38)

The maximum collector dissipation is given by

\[
P_{C(max)} = V_{CC}I_C = V_{CC}I_p = \frac{V_{CC}^2}{R'_L}
\]

(11.39)

Substituting Eqs. (11.38) and (11.39) into Eq. (11.26) yields

\[
F_m = \frac{V_{CC}^2/R'_L}{V_{CC}^2/2R'_L} = 2
\]

(11.40)

Thus, the figure of merit for the transformer-coupled class A stage is the same as that for the basic common-emitter stage.

**EXAMPLE 11.2**

**Designing a transformer-coupled class A amplifier** Design a transformer-coupled class A amplifier with high efficiency to supply an output power of \( P_L = 10 \) W at a load resistance of \( R_L = 4 \) \( \Omega \). Assume a DC supply voltage of 12 V and BJTs of \( h_{fe} = 100 \) and \( V_{CE(sat)} = 0.7 \) V. Note that \( h_{fe} \) is the hybrid parameter and both \( \beta_f = \beta_p \) and \( h_{fe} \) represent the current gain of a BJT.
SOLUTION

The design steps are as follows:

**Step 1.** Determine the maximum collector-to-emitter voltage of the transistors:

\[ V_{CE(max)} \approx 2V_{CC} = 2 \times 12 = 24 \text{ V} \]

**Step 2.** From Eq. (11.40), calculate the collector power dissipation, which must be at least twice the AC power:

\[ P_C \approx F_m P_L = 2P_L = 2 \times 10 = 20 \text{ W} \]

**Step 3.** Calculate the value of the quiescent collector current \( I_C \):

\[ I_C = \frac{P_C}{V_{CC}} = \frac{20}{12} = 1.67 \text{ A} \]

**Step 4.** Calculate the slope of the load line to find the AC load resistance \( R'_L \):

\[ R'_L = \frac{V_{CC}}{I_C} = \frac{12}{1.67} = 7.19 \Omega \]

**Step 5.** From Eq. (11.34), calculate the required turns ratio of the transformer:

\[ n_p = \left( \frac{R'_L}{R_L} \right)^{1/2} = \left( \frac{7.19}{4} \right)^{1/2} = 1.34 \]

**Step 6.** Calculate the peak collector current:

\[ I_{C(max)} = 2I_C = 2 \times 1.67 = 3.34 \text{ A} \]

**Step 7.** Calculate the quiescent base current \( I_B \):

\[ I_B = \frac{I_C}{h_{fe}} = \frac{1.67}{100} = 16.7 \text{ mA} \]

**Step 8.** Calculate the base resistance \( R_B \):

\[ R_B = \frac{V_{CC} - V_{CE(sat)}}{I_B} = \frac{12 \text{ V} - 0.7 \text{ V}}{16.7 \text{ mA}} = 677 \Omega \]

**KEY POINTS OF SECTION 11.4**

- An emitter follower biased with an active current source is the most commonly used output stage.
- An emitter follower has a high input impedance and a low gain of almost unity. However, it exhibits an offset voltage of approximately \( V_{BE} \approx -0.7 \text{ V} \) at \( V_i = 0 \).
- In a class A amplifier, the transistor conducts a continuous DC biasing current. As a result, the maximum power efficiency is only 25%. The figure of merit, which is the ratio of maximum collector power dissipation to maximum output power, is 2.
- The power efficiency of a class A stage can be increased to 50% with a transformer-coupled load.
11.5 Class B Push-Pull Amplifiers

In a class B push-pull amplifier, two complementary transistors (one npn transistor and one pnp transistor) are employed to perform the push-pull operation [1, 6]. The efficiency is much higher than that of a class A amplifier, and the accuracy of the output is improved. It is the most commonly used audio-power amplifier. This section discusses two types of class B amplifiers.

11.5.1 Complementary Push-Pull Amplifiers

A complementary push-pull amplifier is shown in Fig. 11.11(a). For \( v_I > 0 \), transistor \( Q_P \) remains off and transistor \( Q_N \) operates as an emitter follower. For a sufficiently large value of \( v_I \), \( Q_N \) saturates and the maximum positive output voltage becomes

\[
V_{CE(max)} = V_C - V_{CE1(sat)}
\]

For \( v_I < 0 \), transistor \( Q_N \) remains off and transistor \( Q_P \) operates as an emitter follower. For a sufficiently large negative value of \( v_I \), \( Q_P \) saturates and the maximum negative output voltage becomes

\[
-V_{CE(max)} = -(V_C - V_{CE2(sat)}) = -V_C + V_{CE2(sat)}
\]

Assuming identical transistors of \( V_{BE1} = V_{BE2} = V_{BE} \), the output voltage is given by

\[
v_O = v_I - V_{BE} \quad \text{for} \quad -0.7 \leq v_I \leq 0.7 \quad (11.41)
\]

which gives the transfer characteristic of \( v_O \) versus \( v_I \) shown in Fig. 11.11(b). However, during the interval \(-0.7 \leq v_I \leq 0.7 \), both \( Q_P \) and \( Q_N \) remain off, and \( v_O = 0 \). This causes a dead zone and crossover distortion on the output voltage, as illustrated in Fig. 11.12.

![Complementary class B push-pull amplifier](image)
Output Power and Efficiency

Let us assume that $V_{CE1(sat)} = V_{CE2(sat)} = V_{CE(sat)} = 0$ and $I_{C(min)} = 0$. Assuming a sinusoidal variation of the collector current $i_{C1} = I_p \sin \omega t$, the average collector current of a transistor can be found from

$$I_{C1} = \frac{1}{2\pi} \int_0^\pi i_{C1} \, dt = \frac{1}{2\pi} \int_0^\pi I_p \sin (\omega t) \, d(\omega t) = \frac{I_p}{\pi}$$

(11.42)

The average current drawn from the DC supply source by transistors $Q_N$ and $Q_P$ is

$$I_{dc} = 2I_{C1} = \frac{2I_p}{\pi}$$

(11.43)

Thus, the average input power supplied from the DC source is

$$P_S = I_{dc}V_{CC} = \frac{2I_pV_{CC}}{\pi}$$

(11.44)

From Eq. (11.17), the output power is given by

$$P_L = \frac{I_p^2R_L}{2} = \frac{I_pV_p}{2}$$

Thus, the power efficiency becomes

$$\eta = \frac{P_L}{P_S} = \frac{I_pV_p/2}{2I_pV_{CC}/\pi} = \frac{\pi}{4} \left( \frac{V_p}{V_{CC}} \right)$$

(11.45)

which gives $\eta = 50\%$ at $V_p = 2V_{CC} \pi$ and $\eta = 78.5\%$ at $V_p = V_{CC}$. At $V_p = V_{CC}$, the maximum output power is given by

$$P_{L\text{(max)}} = \frac{I_p^2R_L}{2} = \frac{I_pV_p}{2} = \frac{I_pV_{CC}}{2R_L}$$

(11.46)
Thus, the maximum power efficiency is

\[
\eta_{\text{max}} = \frac{P_{L(\text{max})}}{P_S} = \frac{I_p V_{CC}^2 / 2}{2I_p V_{CC} / \pi} = \frac{\pi}{4} = 78.5\% 
\]

Therefore, the maximum efficiency of a complementary push-pull class B amplifier is much higher than that of a class A amplifier.

The average collector power dissipation for both transistors is given by

\[
2P_C = P_S - R_L = 2I_p V_{CC} / \pi - \frac{I_p^2 R_L}{2} = \frac{2V_p V_{CC}}{\pi R_L} - \frac{V_p^2}{2R_L} 
\]  

(11.47)

\[
\frac{dP_C}{dl_p} = \frac{2V_{CC}}{\pi} - \frac{2I_p R_L}{2} = 0 
\]

which gives the peak current for maximum collector power dissipation as

\[
I_{p(\text{max})} = \frac{2V_{CC}}{\pi R_L} 
\]  

(11.49)

and the corresponding maximum peak voltage as

\[
V_{p(\text{max})} = I_{p(\text{max})} R_L = \frac{2V_{CC}}{\pi} 
\]  

(11.50)

Substituting \(I_{p(\text{max})}\) from Eq. (11.49) and \(V_{p(\text{max})}\) from Eq. (11.50) into Eq. (11.47) gives the maximum collector dissipation as

\[
2P_{C(\text{max})} = \frac{4V_{CC}^2}{\pi^2 R_L} - \frac{2V_{CC}^2}{\pi^2 R_L} = \frac{2V_{CC}^2}{\pi^2 R_L} 
\]  

(11.51)

Normalizing \(P_C\) in Eq. (11.48) with respect to \(P_{C(\text{max})}\) in Eq. (11.51), we get

\[
\frac{P_C}{P_{C(\text{max})}} = \pi \left( \frac{V_p}{V_{CC}} \right) - \frac{\pi^2}{4} \left( \frac{V_p}{V_{CC}} \right)^2 
\]

(11.52)

which becomes 100% at \(V_p = 2V_{CC} / \pi\) and 67.4% at \(V_p = V_{CC}\). The normalized plot of \(P_C\) (with respect to \(P_{C(\text{max})}\)) versus peak voltage \(V_p\) is shown in Fig. 11.13. The power dissipation is highest at 50% efficiency.

We can obtain the figure of merit from Eqs. (11.46) and (11.51) as follows:

\[
F_m = \frac{P_{C(\text{max})}}{P_{L(\text{max})}} = \frac{V_{CC}^2 / \pi^2 R_L}{V_{CC}^2 / 2R_L} = \frac{2}{\pi^2} \approx \frac{1}{5} = \frac{2}{10} = 20\% 
\]

(11.53)

Thus, the figure of merit for class B amplifiers exceeds that of class A amplifiers by a factor of 10. The power dissipation rating of the individual transistors is only approximately one-fifth of the output power, and this results in much smaller heat sinks, which are normally needed to keep the junction temperature of power transistors within the maximum permissible limit.
Dead-Zone Minimization

The dead zone can be reduced to practically zero by using feedback with an op-amp, as shown in Fig. 11.14(a). The op-amp is connected in unity-gain mode with series-shunt feedback. With this arrangement, either QP or QN will be on if \( v_I \) and \( v_O \) differ by \( \pm \frac{V_{BE}}{A} \), where \( A \) is the open-loop gain of the op-amp. Thus, for \( A = 10^5 \) and \( V_{BE} = 0.7 \), the dead zone will be reduced to less than \( \pm (0.7 \times 10^5) = \pm 7 \mu V \). The transfer characteristic is shown in Fig. 11.14(b). Resistance \( R_1 \) limits the current drawn by the transistors from the op-amp output. It also provides the base current necessary for the load current \( I_L \). Since the emitter current of a transistor is related to the base current by a factor of \((1 + h_{fe})\), the maximum value of \( R_1 \) can be found approximately from

\[
I_B(1 + h_{fe}) = \frac{V_{BE}(1 + h_{fe})}{R_1} \geq \frac{V_{O(max)}}{R_L} = \frac{V_{CC} - V_{CE(sat)}}{R_L}
\]

which gives the maximum value of \( R_1 \) as

\[
R_1 \leq \frac{V_{BE}(1 + h_{fe})R_L}{V_{CC} - V_{CE(sat)}} \tag{11.54}
\]

![Figure 11.13](image) Power dissipation versus peak output voltage

![Figure 11.14](image) Minimization or elimination of the dead zone with feedback
### EXAMPLE 11.3

**Finding the efficiency and power dissipation of a complementary push-pull amplifier**

(a) Calculate the efficiency and power dissipation of each transistor in the complementary push-pull output stage in Fig. 11.14(a) if \( V_{CC} = V_{EE} = 12 \) V and \( R_L = 50 \) Ω. The parameters of the transistors are \( \beta_F = h_{fe} = 100 \), \( V_{CE(sat)} = 0.2 \) V, and \( V_{BE} = 0.72 \) V.

(b) Use PSpice/SPICE to plot the transfer characteristic. The PSpice model parameters of the transistors are \( BF=100 \) \( \text{VJE}=0.7 \text{V} \)

#### SOLUTION

(a) The peak load voltage is

\[
V_p = V_{CC} - V_{CE(sat)} = 12 - 0.2 = 11.8 \text{ V}
\]

The peak load current is

\[
I_p = \frac{V_p}{R_L} = \frac{11.8 \text{ V}}{50 \text{ Ω}} = 0.236 \text{ A}
\]

From Eq. (11.44), the DC power from the supply source is

\[
P_S = \frac{2I_pV_{CC}}{\pi} = \frac{2 \times 0.236 \text{ A} \times 12 \text{ V}}{\pi} = 1.803 \text{ W}
\]

From Eq. (11.17), the output power is

\[
P_L = \frac{I_pV_p}{2} = \frac{0.236 \text{ A} \times 11.8 \text{ V}}{2} = 1.392 \text{ W}
\]

Thus, the power efficiency is

\[
\eta = \frac{P_L}{P_S} = \frac{1.392 \text{ W}}{1.803 \text{ W}} = 77.2\%
\]

The power dissipation of each transistor can be found from

\[
P_C = \frac{P_S - P_L}{2} = \frac{1.803 \text{ W} - 1.392 \text{ W}}{2} = 206 \text{ mW}
\]

(b) From Eq. (11.54), the maximum value of \( R_1 \) is

\[
R_1 \leq \frac{0.72 \times (1 + 100) \times 50}{12 - 0.2} = 308 \text{ Ω}
\]

We will let \( R_1 = 300 \) Ω. The complementary class B push-pull amplifier circuit for PSpice simulation is shown in Fig. 11.15.

The transfer characteristic is shown in Fig. 11.16(a), which gives \( V_{O(max)} = 10.1 \) V (expected value is 11.8), \( V_{O(min)} = -10.1 \) V (expected value is -11.8), and \( v_O = 16.38 \mu\text{V} \) (expected value is 7 \( \mu\text{V} \)) at \( v_I = 0 \). The input and output voltages are shown in Fig. 11.16(b) for \( v_I = 10 \sin (2000\pi t) \).
11.5.2 Transformer-Coupled Load Push-Pull Amplifier

The power efficiency of an amplifier can be improved considerably by using a transformer-coupled class B push-pull configuration, as shown in Fig. 11.17(a). The amplifier has three stages: input transformer TX1, the gain stage of transistors Q1 and Q2, and output transformer TX2. Resistance \( R_B \) and a battery \( V_{BB} \) are used to provide DC-biasing voltage \( V_{BE} \) for the transistors. Two transistors are employed to perform the push-pull operation. For \( v_I > 0 \), the base of transistor Q1 is positive and that of Q2 becomes negative because of the transformer action. The \( npn \) transistor Q2 remains off, and the \( npn \) transistor Q1 operates as an amplifier. For \( v_I < 0 \), the base of transistor Q2 is positive and that of Q1 becomes negative because of the transformer action. Transistor Q1 remains off, and transistor Q2 operates as an amplifier. The input transformer TX1 of the input stage supplies a virtually distortion-free input signal and matches the output impedance of the driver stage to the input impedance of the output stage.

**Signal Waveforms**

Assuming the input current is sinusoidal, the collector currents of the transistors are as shown in Fig. 11.17(b). The load current shown in Fig. 11.17(c) is composed of the two collector currents \( i_{C1} \) and \( i_{C2} \). The load
current is distorted near zero crossings because the transistors are nonlinear devices and because the base current $i_B = 0$ (and hence $i_C = 0$) for $V_{BE} \leq 0.7$ V. This distortion, shown in Fig. 11.17(c) by dashed lines, is usually referred to as crossover distortion.

**Output Power and Efficiency**

The AC load line for a single transistor ($Q_N$) is shown in Fig. 11.18. The maximum peak current of a transistor is $V_{CC} \div R_L$. The average current drawn from the DC supply source by transistors $Q_N$ and $Q_P$ is

$$I_{dc} = 2I_{C1} = \frac{2I_p}{\pi}$$

(11.55)
Thus, the average input power supplied from the DC source is

\[ P_{dc} = I_{dc}V_{CC} = \frac{2I_pV_{CC}}{\pi} \]  \hspace{1cm} (11.56)

From Eq. (11.18), the maximum output power is

\[ P_{L(max)} = I_{P}^2R'_{L} = \frac{I_pV_p}{2} = \frac{I_pV_{CC}}{2} = \frac{V_{CC}^2}{2R'_{L}} \]  \hspace{1cm} (11.57)

where the effective load resistance \( R'_{L} \) (referred to as the primary side of \( TX_2 \)) is given by

\[ R'_{L} = \left( \frac{n_p}{n_s} \right)^2 R_L \]  \hspace{1cm} (11.58)

Thus, the maximum power efficiency is

\[ \eta_{max} = \frac{P_{L(max)}}{P_{dc}} = \frac{I_pV_{CC}/2}{2I_pV_{CC}/\pi} = \frac{\pi}{4} = 78.5\% \]  \hspace{1cm} (11.59)

Therefore, the maximum efficiency of a transformer-coupled push-pull class B amplifier is much higher than that of a class A amplifier.

The average collector power dissipation for both transistors is given by

\[ 2P_C = P_{dc} - P_L = \frac{2I_pV_{CC}}{\pi} - \frac{I^2pR'_{L}}{2} \]  \hspace{1cm} (11.60)

\[ = \frac{2V_pV_{CC}}{\pi R'_{L}} - \frac{V_{CC}^2}{2R'_{L}} \]  \hspace{1cm} (11.61)

From Eq. (11.49), the peak current for maximum collector power dissipation is

\[ I_{p(max)} = \frac{2V_{CC}}{\pi R'_{L}} \]  \hspace{1cm} (11.62)

From Eq. (11.50), the peak voltage for maximum collector power dissipation is

\[ V_{p(max)} = I_pR'_{L} = \frac{2V_{CC}}{\pi} \]  \hspace{1cm} (11.63)

Substituting Eqs. (11.62) and (11.63) into Eq. (11.51) gives the maximum collector dissipation as

\[ 2P_{C(max)} = \frac{4V^2_{CC}}{\pi^2 R'_{L}} - \frac{2V^2_{CC}}{\pi^2 R^2_{L}} = \frac{2V^2_{CC}}{\pi^2 R^2_{L}} \]  \hspace{1cm} (11.64)

The power efficiency becomes 50% at \( V_p = V_{p(max)} \). At \( V_p = V_{CC} \), the maximum efficiency of 78.5% occurs.
We can obtain the figure of merit from Eqs. (11.57) and (11.64) as follows:

\[ F_m = \frac{P_{C(\text{max})}}{P_{L(\text{max})}} = \frac{V_{CC}^2/\pi^2 R_L^*}{V_{CC}^2/2R_L^*} = \frac{2}{\pi^2} \approx \frac{1}{5} = 20\% \]  

which is the same as that for a complementary push-pull amplifier. The figure of merit for transformer-coupled amplifiers exceeds that of class A amplifiers by a factor of 10.

**DC Biasing**

Resistor \( R_B \) and battery \( V_{BB} \), shown in Fig. 11.19(a), provide the B-E DC voltage \( V_{BE} \), which is approximately 0.7 V for a silicon transistor. In practice, a \( V_{CC} \) supply with a suitable voltage divider is used rather than a separate supply, as shown in Fig. 11.19(b). \( R_1 \) and \( R_2 \) are chosen so that \( V_{BE} \approx 0.7 \) V (for silicon transistors). The parallel combination of \( R_1 \) and \( R_2 \) is kept as small as possible so that voltage drop \( 2R_BI_B1 \ll V_{BE} \), which is generally satisfied by choosing \( 2R_BI_B1 = 0.1V_{BE} \). Since the diode drop is similar to the B-E voltage of a transistor, a silicon diode is often used instead of resistance \( R_1 \), as shown in Fig. 11.19(c).

**EXAMPLE 11.4**

**Designing a transformer-coupled class B amplifier** Design a transformer-coupled class B push-pull amplifier, as shown in Fig. 11.17(a), to supply a maximum output power of \( P_{L(\text{max})} = 10 \) W at a load resistance of \( R_L = 4 \) \( \Omega \). Assume a DC supply voltage of 15 V and transistors of \( h_{fe} = 100 \) and \( V_{BE} = 0.7 \) V.

**SOLUTION**

The design steps are as follows:

**Step 1.** Determine the maximum collector-to-emitter voltage of the transistors:

\[ V_{CE(\text{max})} = 2V_{CC} = 2 \times 15 \text{ V} = 30 \text{ V} \]

**Step 2.** From Eq. (11.57), calculate the effective load resistance:

\[ R_L^* = \frac{V_{CC}^2}{2P_{L(\text{max})}} = \frac{(15 \text{ V})^2}{2 \times 10 \text{ W}} = 11.25 \text{ } \Omega \]
Step 3. Calculate the peak current of each transistor:

\[
I_p = \frac{V_{CC}}{R_L} = \frac{15 \text{ V}}{11.25 \text{ } \Omega} = 1.33 \text{ A}
\]

Step 4. From Eq. (11.55), calculate the average current of each transistor:

\[
I_{C1} = \frac{I_p}{\pi} = \frac{1.33 \text{ A}}{\pi} = 0.424 \text{ A}
\]

Step 5. From Eq. (11.64), calculate the maximum collector power dissipation:

\[
P_C = \left( \frac{2}{\pi^2} \right) P_{L(max)} = \left( \frac{2}{\pi^2} \right) \times 10 \text{ W} = 2 \text{ W}
\]

Step 6. Calculate the DC power from the source:

\[
P_S = 2I_{C1}V_{CC} = 2 \times 0.424 \text{ A} \times 15 \text{ V} = 12.72 \text{ W}
\]

Step 7. From Eq. (11.58), calculate the required turns ratio of the transformer:

\[
n_p = \left( \frac{R'_L}{R_L} \right)^{1/2} = \left( \frac{11.25 \text{ } \Omega}{4 \text{ } \Omega} \right)^{1/2} = 1.68
\]

Step 8. Calculate the required quiescent base current \(I_{B1}\):

\[
I_{B1} = \frac{I_{C1}}{h_{fe}} = \frac{0.424 \text{ A}}{100} = 4.24 \text{ mA}
\]

Step 9. Calculate the biasing resistances \(R_1\) and \(R_2\). We have

\[
V_{BB} - V_{BE} = R_B(I_{B1} + I_{B2}) = R_B(2I_{B1})
\]

If we let \(R_B(2I_{B1}) = 10\% \) of \(V_{BE} = 0.1V_{BE} = 0.1 \times 0.7 \text{ V} = 0.07 \text{ V} \), then

\[
R_B = \frac{0.07 \text{ V}}{2I_{B1}} = 8.25 \text{ } \Omega
\]

and \(V_{BB} = V_{BE} + R_B(2I_{B1}) = 0.7 \text{ V} + 0.07 \text{ V} = 0.77 \text{ V}\)

Since \(V_{BB} = \frac{R_1V_{CC}}{R_1 + R_2} = 0.77 \text{ V}\)

and \(R_B = \frac{R_1R_2}{R_1 + R_2} = 8.25 \text{ } \Omega\)

Solving for \(R_1\) and \(R_2\), we get

\[
R_1 = \frac{R_BV_{CC}}{V_{CC} - V_{BB}} = \frac{8.25 \text{ } \Omega \times 15 \text{ V}}{15 \text{ V} - 0.77 \text{ V}} = 8.7 \text{ } \Omega
\]

\[
R_2 = \frac{R_BV_{CC}}{V_{BB}} = \frac{8.25 \text{ } \Omega \times 15 \text{ V}}{0.77 \text{ V}} = 161 \text{ } \Omega
\]
The crossover distortion of a complementary class B push-pull amplifier is minimized or eliminated in a class AB amplifier, in which the transistors operate in the active region when the input voltage $v_I$ is small ($v_I \approx 0$ V). The quiescent DC-biasing current is zero. As a result, the maximum power efficiency is 78.5%, and the maximum figure of merit is only 20%.

Because of B-E voltage drops, a push-pull amplifier exhibits a dead zone in the transfer characteristic, which increases distortion of the output voltage. The crossover distortion and nonlinearities can be reduced practically to zero by applying feedback.

---

**KEY POINTS OF SECTION 11.5**

- In a class B push-pull amplifier, a pnp and an npn transistor form a pair, and each transistor conducts for only 180°. The quiescent DC-biasing current is zero. As a result, the maximum power efficiency is 78.5%, and the maximum figure of merit is only 20%.
- Because of B-E voltage drops, a push-pull amplifier exhibits a dead zone in the transfer characteristic, which increases distortion of the output voltage. The crossover distortion and nonlinearities can be reduced practically to zero by applying feedback.

---

**11.6 Complementary Class AB Push-Pull Amplifiers**

The crossover distortion of a complementary class B push-pull amplifier is minimized or eliminated in a class AB amplifier, in which the transistors operate in the active region when the input voltage $v_I$ is small ($v_I \approx 0$ V). The transistors are biased in such a way that each transistor conducts for a small quiescent current $I_Q$ at $v_I = 0$ V. A biasing circuit is shown in Fig. 11.20(a). A biasing voltage $V_{BB}$ is applied between the bases of $Q_N$ and $Q_P$. For $v_I = 0$, a voltage $V_{BB}/2$ appears across the B-E junction of each $Q_N$ and $Q_P$. Choosing $V_{BB}/2 = V_{BEN} = V_{EBP}$ will ensure that both transistors will be on the verge of conducting. That is, $v_O = 0$ for $v_I = 0$. A small positive input voltage $v_I$ will then cause $Q_N$ to conduct; similarly, a small negative input voltage will cause $Q_P$ to conduct.

**FIGURE 11.20** Elimination of the dead zone in a class AB amplifier
11.6.1 Transfer Characteristic

The output voltage \( v_O \) is given by

\[
v_O = v_I + \frac{V_{BB}}{2} - V_{BEN} (= V_{EBP}) \tag{11.66}
\]

which, for identical transistors of \( V_{BEN} = V_{EBP} \) and \( V_{BB} \), gives \( v_O = v_I \). Therefore, most of the crossover distortion is eliminated. The transfer characteristic is shown in Fig. 11.20(b). For positive \( v_O \), a current \( i_O \) flows through \( R_L \); that is,

\[
i_N = i_P + i_O \tag{11.67}
\]

Any increase in \( i_N \) will cause a corresponding increase in \( V_{BEN} \) above the quiescent value of \( V_{BB} \). Since \( V_{BB} \) must remain constant, the increase in \( V_{BEN} \) will cause an equal decrease in \( V_{EBP} \) and hence in \( i_P \). Thus,

\[
V_{BB} = V_{BEN} + V_{EBP} \tag{11.68}
\]

which, expressed in terms of saturation current \( I_S \), becomes

\[
2VT \ln \left( \frac{I_Q}{I_S} \right) = V_T \ln \left( \frac{i_N}{I_S} \right) + V_T \ln \left( \frac{i_P}{I_S} \right)
\]

After simplification, we get

\[
I_Q^2 = i_N i_P \tag{11.69}
\]

\[
= i_N(i_N - i_O) = i_N^2 - i_N i_O \tag{11.70}
\]

which can be solved for the current \( i_N \) for a given quiescent current \( I_Q \). Thus, as \( i_N \) increases, \( i_P \) decreases by the same ratio. However, their product remains constant. As \( v_I \) becomes positive, \( Q_N \) acts as an emitter follower delivering output power, and \( Q_P \) conducts only a very small current. When \( v_I \) becomes negative, the opposite occurs: \( Q_P \) acts as an emitter follower, and \( v_O \) follows the input signal \( v_I \). The circuit operates in class AB mode because both transistors remain on and operate in the active region.

11.6.2 Output Power and Efficiency

The power relationships in class AB amplifiers are identical to those in class B amplifiers, except that the class AB circuit dissipates a quiescent power of \( I_Q V_{CC} \) per transistor. Thus, from Eq. (11.44), we can find the average power supplied from the DC source as

\[
P_S = \frac{2i_P V_{CC}}{\pi} + I_Q V_{CC} = V_{CC} \left( I_Q + \frac{2i_P}{\pi} \right) \tag{11.71}
\]

11.6.3 Biasing with Diodes

The biasing circuit in Fig. 11.20(a) has a serious problem when the temperatures of \( Q_N \) and \( Q_P \) increase as a result of their power dissipation. Recall that the value of \( V_{BE} \) for a given current falls with temperature at approximately 2.5 mV °C. Thus, if the biasing voltage \( V_{BB} \) remains constant with temperature,
\( V_{BE} (= V_{BB} - 2) \) is also held constant, and the collector current will increase as temperature increases. The increase in the collector current increases the power dissipation, in turn increasing the collector current and causing the temperature to rise further. This phenomenon, in which a positive feedback mechanism leads to excessive temperature rise, is called \textit{thermal runaway}. Thermal runaway can ultimately lead to the destruction of the transistors unless they are protected.

To avoid thermal runaway, the biasing voltages must decrease as the temperature increases. One solution is to use diodes that have a compensating effect, as shown in Fig. 11.20(c). The diodes must be in close contact with the output transistors so that their temperature will increase by the same amount as that of \( Q_N \) and \( Q_P \). Therefore, in discrete circuits, the diodes should be mounted on the metal of \( Q_N \) or \( Q_P \). Since resistances \( R_1 \) and \( R_2 \) provide the quiescent current \( I_Q \) for the transistors and also ensure that the diodes conduct, to guarantee the base biasing current for \( Q_N \) when the load current becomes maximum we must have

\[
I_R = I_{D1} + \frac{i_N}{1 + h_{fe}} \approx I_{D1} + \frac{I_Q + i_O}{1 + h_{fe}}
\]

Thus, the values of \( R_1 \) and \( R_2 \) can be found from

\[
R_1 = R_2 = \frac{V_{CC} - V_{D1} (= V_{D2} = V_{BB}/2)}{1/I_{D1(min)} + (I_Q + i_O)/(1 + h_{fe})}
\]

where \( I_Q = I_S \exp (V_{BB} - 2V_T) \) and \( I_{D1(min)} \) is the minimum current needed to ensure diode conduction. Because \( I_Q \) is usually smaller than \( i_O(max) \), \( I_Q \) can often be neglected in finding the values of \( R_1 \) and \( R_2 \).

**EXAMPLE 11.5**

**Designing a biasing circuit for a class AB amplifier**

(a) Design a biasing circuit for the class AB amplifier of Fig. 11.20(c) to supply the maximum output voltage at a load resistance of \( R_L = 50 \Omega \). The quiescent biasing current \( I_Q \) is 2 mA. Assume a DC supply voltage of 12 V. The diode parameters are \( I_S = 10^{-13} \text{A}, V_{D1} = V_{D2} = 0.7 \text{V}, \) and \( I_{D1(min)} = 1 \text{mA} \) to ensure conduction. The transistor parameters are \( \beta_F = h_{fe} = 50, V_{BE} = 0.7 \text{V}, I_S = 10^{-14} \text{A}, \) and \( V_{CE(sat)} = 0.2 \text{V.} \)

(b) Find the biasing voltage \( V_{BB} \) for \( v_O = 0 \) and 11.8 V.

**SOLUTION**

(a) The maximum peak load voltage is

\[
V_{p(max)} = V_{CC} - V_{CE(sat)} = 12 - 0.2 = 11.8 \text{V}
\]

The maximum peak load current is

\[
I_{p(max)} = \frac{V_{p(max)}}{R_L} = \frac{11.8}{50} = 236 \text{mA}
\]

From Eq. (11.71), the maximum DC power from the supply source is

\[
P_S \approx \frac{2I_{p(max)}V_{CC}}{\pi} + I_QV_{CC} = \frac{2 \times 236 \text{mA} \times 12 \text{V}}{\pi} + 2 \text{mA} \times 12 \text{V} = 1.83 \text{W}
\]
From Eq. (11.17), the output power is

\[ P_{L(\text{max})} = \frac{I_{p(\text{max})}V_{p(\text{max})}}{2} = 236 \text{ mA} \times \frac{11.8 \text{ V}}{2} = 1.39 \text{ W} \]

Thus, the maximum power efficiency is

\[ \eta_{\text{max}} = \frac{P_{L(\text{max})}}{P_{S}} = \frac{1.39}{1.83} = 76.1\% \]

The power dissipation of each transistor can be found from

\[ P_{C} = \frac{P_{S} - P_{L(\text{max})}}{2} = \frac{1.83 \text{ W} - 1.39 \text{ W}}{2} = 220 \text{ mW} \]

From Eq. (11.72), we get

\[ R_{1} = R_{2} = \frac{12 \text{ V} - 0.7 \text{ V}}{1 \text{ mA} + (2 \text{ mA} + 236 \text{ mA})/(1 + 50)} = 1.99 \text{ k}\Omega \]

(b) We have

\[ I_{R} = I_{D1(\text{min})} + \frac{I_{Q} + I_{p(\text{max})}}{1 + h_{fe}} = 1 \text{ mA} + \frac{2 \text{ mA} + 236 \text{ mA}}{1 + 50} = 5.667 \text{ mA} \]

For \( v_{O} = 0 \), \( i_{N} = I_{Q} = 2 \text{ mA} \). Thus, the base current of the \( npn \) transistor is

\[ I_{BN} = \frac{I_{N}}{1 + h_{fe}} = \frac{2 \text{ mA}}{1 + 50} = 0.039 \text{ mA} \]

and \( I_{D1} = I_{R} - I_{BN} = 5.667 \text{ mA} - 0.039 \text{ mA} = 5.628 \text{ mA} \)

Therefore, the biasing voltage \( V_{BB} \) becomes

\[ V_{BB} = 2V_{T} \ln \left( \frac{I_{D1}}{I_{S}} \right) = 2 \times 25.8 \text{ mV} \times \ln \left( \frac{5.628 \text{ mA}}{10^{-13} \text{ A}} \right) = 1.277 \text{ V} \]

For \( v_{O} = 11.8 \), \( i_{N} = I_{Q} + I_{p(\text{max})} = 2 \text{ mA} + 236 \text{ mA} = 238 \text{ mA} \). Thus,

\[ I_{BN} = \frac{i_{N}}{1 + h_{fe}} = \frac{238 \text{ mA}}{1 + 50} = 4.67 \text{ mA} \]

and \( I_{D1} = I_{R} - I_{BN} = 5.667 \text{ mA} - 4.67 \text{ mA} = 1 \text{ mA} \)

Therefore, for \( I_{D1} = 1 \text{ mA} \), the biasing voltage \( V_{BB} \) becomes \( V_{BB} = 1.19 \text{ V} \).
Transfer Characteristic

The voltage between the bases of Q_P and Q_N is the same as the voltage drop across the two diodes. That is, 

$$V_{BB} = V_{D1} + V_{D2} \approx 0.7 + 0.7 = 1.4 \text{ V}$$

The base-to-emitter voltage of Q_N is given by

$$V_{BEN} = V_{BB} - V_{EBP} = 1.4 - V_{EBP}$$

(11.73)

Thus, the B-E junctions of both Q_N and Q_P are always forward biased. Because of diodes D_1 and D_2, Q_N and Q_P remain in the active region when \(v_1 = 0 \text{ V}\). The output voltage \(v_O\) is given by

$$v_O = v_1 + V_{BB} - V_{BEN}$$

$$= v_1 + V_{EBP} = v_1 - V_{BEP}$$

(11.74)

which yields the transfer characteristic shown in Fig. 11.21(b). The dead zone is eliminated. However, there is an offset voltage of \(V_{EBP}\), which can be reduced practically to zero by applying feedback similar to that applied to the class B amplifier in Fig. 11.14.

**EXAMPLE 11.6**

**Designing an active current-source biasing circuit for a class AB amplifier**

(a) Design an active current source for the class AB amplifier in Fig. 11.21(a) in order to provide the biasing current of \(I_R = 5.67 \text{ mA}\) needed for Example 11.5. Assume \(V_{CC} = V_{EE} = 12 \text{ V}, V_{BE} = 0.7 \text{ V}, \) and \(R_L = 50 \Omega\).

(b) Use PSpice/SPICE to plot the transfer characteristic and the instantaneous \(i_N, i_R, \) and \(i_O\) for \(v_1 = 5 \sin (2000\pi t)\). The PSpice model parameters for the transistors are

\[ IS=1E^{-14} \quad BF=50 \quad VJE=0.7 \]

and for the diodes are

\[ IS=1E^{-13} \quad BV=100 \]
Power Amplifiers

SOLUTION

(a) The biasing current source $I_R$, shown in Fig. 11.22, can be produced by two pnp transistors $Q_1$ and $Q_2$ and a resistance $R_1$. Thus,

$$I_R \approx I_{ref} = \frac{V_{CC} - V_{EBP}}{R_{ref}}$$  \hspace{1cm} (11.75)

which, for $I_R = 5.67$ mA, gives

$$R_{ref} = \frac{V_{CC} - V_{EBP}}{I_R} = \frac{12 \text{ V} - 0.7 \text{ V}}{5.67 \text{ mA}} \approx 2 \text{ k}\Omega$$

(b) In practice, emitter resistances $R_{E1}$ and $R_{E2}$, shown in Fig. 11.22, are connected to ensure the stability of the biasing point.

The transfer characteristic for the circuit in Fig. 11.22 is shown in Fig. 11.23(a). It gives $V_{O(max)} = V(2) = 11.139$ V and offset voltages of $v_{O1} = 560.4$ mV at $v_I = 0$ and $v_{O2} = 0$ at $v_I = -640.9$ mV. The plots of $i_N$, $i_P$, and $i_Q$ are shown in Fig. 11.23(b), which give $i_{N(peak)} = I(RE1) = 109.66$ mA, $i_{P(peak)} = I(RE2) = -83.92$ mA, and $i_{Q(peak)} = I(RL) = 109.59$ mA to $-83.83$ mA. Thus, because of the offset voltage, the load current is not symmetrical.

FIGURE 11.22  Complementary class AB push-pull amplifier circuit for PSpice simulation

FIGURE 11.23  Transfer characteristic and output voltage for Example 11.6
11.6.5 Biasing with a $V_{BE}$ Multiplier

A $V_{BE}$ multiplier circuit that can automatically adjust the biasing voltage $V_{BB}$ is shown in Fig. 11.24. The circuit consists of a transistor $Q_1$ with a resistor $R_1$ connected between its base and emitter and a feedback resistor $R_F$ connected between the collector and the base. The current source $I_R$ supplies the multiplier circuit and the base current for $Q_N$. Since the voltage across $R_1$ is $V_{BE1}$, the current through $R_1$ is given by

$$I_1 = \frac{V_{BE1}}{R_1} \quad (11.76)$$

The base current of $Q_1$ is generally negligible compared to $I_1$, and the current through $R_F$ is approximately equal to $I_1$. Thus, the biasing voltage becomes

$$V_{BB} = I_1(R_1 + R_F) = \frac{V_{BE1}}{R_1}(R_1 + R_F) = V_{BE1} \left(1 + \frac{R_F}{R_1}\right) \quad (11.77)$$

Therefore, the circuit multiplies $V_{BE1}$ by the factor $(1 + R_F/R_1)$—hence the name $V_{BE}$ multiplier. By selecting the ratio $R_F/R_1$, one can set the value of $V_{BB}$ required to give a desired quiescent current $I_Q$. For $R_F/R_1 = \frac{1}{2}$, $V_{BB} = 2V_{BE1}$. The value of $V_{BE1}$ is related to $i_{C1}$ by

$$V_{BE1} = V_T \ln \left(\frac{i_{C1}}{I_{S1}}\right) \quad (11.78)$$

where $I_{S1}$ is the saturation current of $Q_1$ and

$$i_{C1} = I_R - I_1 - \frac{I_Q + i_O}{1 + h_{fe}} \quad (11.79)$$

Under quiescent conditions, $i_O = 0$ and the base current of $Q_N$ is normally small enough that it can be neglected. That is, $I_Q (1 + h_{fe}) \approx 0$, and $Q_1$ carries the maximum current: $I_{C1(max)} \approx I_R - I_1$. However, at the peak value of $v_O$, the base current of $Q_N$ will be maximum and the current available to the multiplier will be minimum; that is,

$$I_{C1(min)} = I_R - I_1 - \frac{I_Q + i_{O(max)}}{1 + h_{fe}} \quad (11.80)$$

FIGURE 11.24 Biasing of a class AB amplifier with a $V_{BE}$ multiplier
Thus, \( i_{C1} \) can vary widely from \( I_{C1(\text{max})} \) to \( I_{C1(\text{min})} \). However, according to Eq. (11.78), a large change in \( i_{C1} \) will cause only a small change in \( V_{BE1} \). Thus, \( I_1 \) and \( V_{BB} \) will remain almost constant. As with diode biasing, the transistor \( Q_1 \) must be in close contact with \( Q_N \) and \( Q_P \) in order to provide a thermal compensating effect.

**EXAMPLE 11.7**

**Designing a \( V_{BE} \) multiplier for a class AB amplifier**

(a) Design a \( V_{BE} \) multiplier for the class AB amplifier in Fig. 11.24 in order to provide the biasing current of \( I_R = 5.67 \, mA \) needed for Example 11.5. Assume \( h_{fe} = 50, I_S = 10^{-14} A, V_{CC} = V_{EE} = 12 \, V, V_{BE} = 0.7 \, V, \) and \( R_L = 50 \, \Omega \). Assume a minimum current of \( I_{M(\text{min})} = 1 \, mA \) to the multiplier, \( I_Q = 2 \, mA \), and peak load current \( I_{P(\text{max})} = 236 \, mA \).

(b) Use PSpice/SPICE to plot the transfer characteristic and the instantaneous \( i_N, i_P, \) and \( i_O \) for \( v_I = 5 \sin (2000 \pi t) \). The PSpice model parameters for the transistors are 

\[
I_S=1E^{-14} \quad BF=50 \quad V_JE=0.7
\]

and for the diodes are 

\[
I_S=1E^{-13} \quad BV=100
\]

**SOLUTION**

(a) Since the current source must provide the base current when the load current is maximum,

\[
I_R = I_{M(\text{min})} + \frac{I_Q + I_{P(\text{max})}}{1 + h_{fe}} = 1 \, mA + \frac{2 \, mA + 236 \, mA}{1 + 50} = 5.67 \, mA
\]

The biasing voltage \( V_{BB} \) required to yield a quiescent current of \( I_Q = 2 \, mA \) is

\[
V_{BB} = 2V_T \ln \left( \frac{I_Q}{I_S} \right) = 2 \times 25.8 \, mV \times \ln \left( \frac{2 \, mA}{10^{-14} \, mA} \right) = 1.343 \, V
\]

The minimum current through the multiplier must be \( I_{M(\text{min})} = 1 \, mA \). Let \( I_{I(\text{min})} = I_{M(\text{min})} \), \( 2 = 0.5 \, mA \) and \( I_{C(\text{min})} = I_{M(\text{min})} \), \( 2 = 0.5 \, mA \). If \( I_{C(\text{min})} \) is too small, transistor \( Q_1 \) will be off, which is not desirable. Equation (11.77) gives

\[
R_1 + R_F = \frac{V_{BB}}{I_{I(\text{min})}} = \frac{1.343 \, V}{0.5 \, mA} \approx 2.7 \, k\Omega
\]

The current source must be designed to supply \( I_R = 5.67 \, mA \). However, when the output voltage is zero, then \( i_Q = 0, I_R = 5.67 \, mA \) must flow through the multiplier. That is, transistor \( Q_1 \) must carry \( i_{C1} = I_R - I_{I(\text{min})} = 5.67 \, mA - 0.5 \, mA = 5.17 \, mA \), and the corresponding B-E voltage will be

\[
V_{BE1} = V_T \ln \left( \frac{i_{C1}}{I_S} \right) = 25.8 \, mV \times \ln \left( \frac{5.17 \, mA}{10^{-14} \, mA} \right) = 0.696 \, V
\]
Thus, the value of $R_1$ can be found from Eq. (11.76) as

$$R_1 = \frac{V_{BE1}}{I_1} = \frac{0.696 \text{ V}}{0.5 \text{ mA}} = 1.39 \text{ kΩ} \approx 1.4 \text{ kΩ}$$

Therefore, the value of $R_F$ becomes

$$R_F = 2.7 \text{ kΩ} - R_1 = 2.7 \text{ kΩ} - 1.4 \text{ kΩ} = 1.3 \text{ kΩ}$$

(b) The circuit file for the PSpice simulation is similar to that for Example 11.6, except that the diodes are replaced by the $V_{BE}$ multiplier. This configuration is shown in Fig. 11.25.

The transfer characteristic is shown in Fig. 11.26(a). It gives $V_{O(max)} = 11.13$ V and offset voltages of $V_O = 794.3$ mV at $V_I = 0$ and $V_O = 0$ at $V_I = -796$ mV. The plots of the input and output voltages are shown in Fig. 11.26(b). Their waveforms are almost identical, except that the magnitudes are shifted by 763 mV.
11.6.6 Quasi-Complementary Class AB Amplifiers

Because *pnp* transistors have limited current-carrying capability, the complementary output stage is suitable only for delivering load power on the order of a few hundred milliwatts or less. If output power of several watts or more is required, *npn* transistors should be used. A composite *pnp* transistor can be made from a *pnp* transistor $Q_P$ and a high-power *npn* transistor $Q_{N1}$. This arrangement, called a quasi-complementary output stage, is shown in Fig. 11.27(a).

The pair $Q_P$-$Q_{N1}$ is equivalent to a *pnp* transistor, as shown in Fig. 11.27(b). The collector current of $Q_P$ is given by

$$I_{CP} = I_S \exp \left( \frac{V_{EBP}}{V_T} \right)$$

(11.81)

The composite collector current $I_C$ is the emitter current of $Q_{N1}$; that is,

$$I_C = (1 + h_{fe})I_{CP} = (1 + h_{fe})I_S \exp \left( \frac{V_{EBP}}{V_T} \right)$$

(11.82)

![Diagram of a quasi-complementary class AB output stage](image1)

![Equivalent pnp transistor](image2)

![Equivalent PMOS](image3)

**FIGURE 11.27** Quasi-complementary class AB output stage
which is the same as the relationship for a normal pnp transistor. However, the npn transistor carries most of the current and the pnp transistor carries only a small amount of the current. The saturation voltage of the composite pnp will be $V_{CEP(sat)} + V_{BEN1}$, which is higher than that of a normal pnp transistor.

A composite pnp transistor can be replaced by a MOS-bipolar combination [4], known as a composite PMOS, as shown in Fig. 11.27(c). From Eq. (7.7), the overall transfer characteristic of the composite PMOS is given by

$$I_D = -(1 + h_{fe})I_{DP} = -(1 + h_{fe})\frac{h_{n}C_o}{2}\left(\frac{W}{L}\right)(V_{GS} - V_{T})^2$$

Thus, the composite PMOS has a $W/L$ ratio that is $(1 + h_{fe})$ times larger than that of a normal PMOS device.

### 11.6.7 Transformer-Coupled Class AB Amplifiers

The class AB circuit shown in Fig. 11.28 is identical to the class B circuit in Fig. 11.17(a), except that it is biased slightly into conduction so that a quiescent current $I_Q$ flows through $Q_1$ and $Q_2$. This current is achieved by making $V_{BB}$ slightly greater than $V_{BE} = V_{BE1} (= V_{BE2} \approx 0.7 \text{ V})$. Resistors $R_1$ and $R_2$ can be selected to give the desired value of $V_{BB}$. Although transformer-coupled amplifiers offer high power efficiency, they suffer from nonlinearities and distortion introduced by the nonlinear characteristics of the transformers. They are being replaced by direct-coupled all-transistorized circuits.

The nonlinear effects and distortion can be eliminated by applying series-shunt negative feedback, as shown in Fig. 11.29. The amplifier has three stages: a CE stage for voltage gain, an emitter follower for impedance matching, and an output stage for high power output. The series-shunt feedback gives the amplifier the desirable features of low output impedance and high input impedance. The overall voltage gain $A_f$ depends mostly on the feedback network; that is,

$$A_f \approx \frac{1}{\beta} = \frac{1}{1 + \frac{R_F}{R_E + R_F}} = 1 + \frac{R_F}{R_E}$$

where $\beta$ is the feedback factor.

![Transformer-coupled class AB push-pull amplifier](image-url)
11.7 Class C Amplifiers

Class C amplifiers conduct less than 50% of the input signal, and the efficiencies can be as high as 90%, but the output has a high amount of distortion. These amplifiers can find applications in radio frequency transmitters where the distortion can be considerably reduced by using tuned loads on the amplifying devices. Figure 11.30(a) shows the simplified arrangement of a class C amplifier with a parallel-connected load resistance $R_L$. To isolate the load from the DC power supply and for impedance matching with the external load, $R_L$ is generally connected through a transformer. The class C amplifier can also be operated
as a push-pull configuration consisting of an NMOS and a PMOS with a common gate connection for the input signal.

When the gate signal $v_G = V_G + V_m \sin \omega t$ exceeds the MOS threshold voltage $V_t$, the MOS device conducts, and the corresponding drain current flows through the parallel load. The small-signal equivalent circuit is shown in Fig. 11.30(b). In the linear region, the transistor will behave as a switch with a drain–source resistance, which can be found from Eq. (11.2) as given by

$$R_{DS} = \frac{V_{DS}}{i_D} \approx \frac{L}{W \mu_n C_{ox}(V_G + V_m \sin \omega t - V_t)}$$  

The AC gate–source voltage $v_{gs} = V_m \sin \omega t$ will vary the drain current ($i_d = g_m v_{gs}$), which is the source for the $RLC$ load. From Fig. 11.30(b), we can find the small-signal AC output voltage as given by

$$v_o = g_m v_{gs}(R_L \parallel X_L \parallel X_C)$$  

where $X_L(= j \omega L)$ and $X_C(=1/j \omega C)$ are the impedances of $L$ and $C$, respectively. If the gate–source voltage is a pulsed DC voltage of PWM, the drain current will also be of the same form.

The values of $L$ and $C$ can be selected either to oscillate at a specified damping factor or to form a tuned circuit by choosing the values of $L$ and $C$ such that $\omega L = 1/\omega C$ at the tuned frequency $\omega_n$. The tuned circuit will resonate only at particular frequencies, so the unwanted frequencies are dramatically suppressed, and the desirable full signal (sine wave) will be abstracted by the tuned load. Provided that the transmitter is not required to operate over a very wide band of frequencies, this arrangement works extremely well. Other residual harmonics can be removed using a filter.

The load $R_L$ also can be connected to form a series resonant circuit as shown in Fig. 11.30(c). When the transistor is switched on, the drain and the inductor currents will rise through the path formed by $V_{DD}$, $L$, and $M_1$. When the transistor is switched off, the drain current will be zero, and the inductor current will fall through the path formed by $V_{DD}$, $L$, $C$, and $R_L$. The energy stored in the inductor during the transistor on-time will be transferred to the load $R_L$ via the capacitor $C$. The damping factor of series and parallel $RLC$ circuits (see Appendix B) can be found from

$$\delta = \frac{1}{2R_L} \sqrt{\frac{L}{C}}$$  

for series resonant

$$= \frac{1}{2R_L} \sqrt{\frac{L}{C}}$$  

for parallel resonant
The values of $L$ and $C$ are related to the damping ratio $\delta$ and the resonant frequency $f_n$ as given by

$$L = \frac{R_L}{2\delta \omega_n} \quad \text{(for series resonant)} \quad (11.88)$$

$$C = \frac{R_L}{2\delta \omega_n} \quad \text{(for parallel resonant)} \quad (11.89)$$

**EXAMPLE 11.8**

**Finding the values of $L$ and $C$ for a class C amplifier** The gate voltage of the class C amplifier in Fig. 11.30(a) is $v_G = 2 + 1.5 \sin (2\pi f_s t)$ with $f_s = 1$ MHz. The DC supply voltages are $V_{DD} = \pm 15$ V. The load resistance $R_L = 50$ Ω, which is connected through a transformer of turns ratio of $n = 1:1$.

(a) Find the values of $L$ and $C$ to form a tuned resonant circuit.

(b) Use PSpice to plot the voltage $v_D(t)$, the transistor drain current $i_D(t)$, and the transistor drain voltage $v_{DD}$ for varying the gate voltage component $V_m = 3$ V to 4 V with an increment of 0.5 V. Use the PSpice parameters of the NMOS IRF150 whose $V_t = 2.84$ V.

**SOLUTION**

(a) For $n = 1:1$, the effective load resistance is $R = R_L/n^2 = 50$ Ω. From Eq. (11.87), we can find the damping factor for a tuned circuit of $L = C$, $\delta = (1/2R)\sqrt{L/C} = 1/2 \times 50 = 0.01$. For $f_s = 1 \times 10^6$, the resonant frequency $f_0 = f_s = 1 \times 10^6$, which is related to $L$ and $C$ by $f_0 = 1/(2\pi \sqrt{LC})$. From Eqs. (11.88) and (11.89), we get

$$L = \frac{2\delta R}{2\pi f_s} = \frac{2 \times 0.01 \times 50}{2 \times \pi \times 10^6} = 159.2 \text{ nH} \approx 160 \text{ nH}$$

$$C = \frac{1}{2\delta R^2 f_s} = \frac{1}{2 \times 0.01 \times 50 \times 2 \times \pi \times 10^6} = 159.2 \text{ nF} \approx 160 \text{ nF}$$

(b) The PSpice schematic is shown in Fig. 11.31. A small resistance $R_s = 1 \mu\Omega$ is connected to avoid a convergence problem of the zero-resistance DC loop. The PSpice plot of the transistor drain current $i_D(t)$ is shown in Fig. 11.32(a), the transistor drain voltage $v_D(t)$ in Fig. 11.32(b), and the output load voltage $v_{DD}(t)$ in Fig. 11.32(c) for $V_m = 3$ V, 3.5 V, and 4 V. For $v_G(t) < V_t = 2.84$ V, the transistor is off. The output voltage increases with the gate voltage as expected as shown in Fig. 11.32(c). The drain current becomes negative at a higher value because the inductive load $L$ induces a negative voltage ($Ldi/dt$) effect due to the abrupt turning off of the transistor; this negative voltage forward biases the built-in source–channel $pn$ junction. The peak drain voltage reaches $2V_{DD}$ due to the $Ldi/dt$ effect.
FIGURE 11.31  PSpice schematic for Example 11.8

FIGURE 11.32  PSpice voltages and currents of class C amplifier for Example 11.8
11.8 Class D Amplifiers

A class D amplifier as shown in Fig. 11.33(a) operates transistors $M_1$ and $M_2$ as complementary switches—which are either completely turned on or completely turned off with a PWM gate signal—which are generated by comparing the sinusoidal reference (or input) signal voltage $v_r$ with a triangular wave carrier signal $v_{cr}$. The technique for the generation of the PWM waveform is shown in Fig. 11.33(b), which can be implemented with an op-amp comparator. When the gate–source voltage is positive, the NMOS is switched on while the PMOS is turned off, and the voltage at the transistor drain is $+V_{DD}$. When the gate–source voltage is negative, the PMOS is switched on while the NMOS is turned off, and the transistor output is $-V_{SS}$.

The output voltage at the drain terminal is a pulsed waveform switching between $+V_{DD}$ and $V_{SS}$ at the carrier frequency $f_c$, which is a multiple (in the range of 10 to 30) of the signal frequency $f_s$. Since the PWM waveform was generated from a sinusoidal reference signal, a low-pass $LC$ filter as shown in Fig. 11.33(c) is used to restore the original sinusoidal component from the high-frequency modulated voltage. One additional $LC$ filter may be used to reduce the amount of distortion. The class D amplifier is highly efficient (around 95%), which can be even closer to 100%, and finds applications in such portable devices as laptop computers and MP3 players. Class D amplifiers also find applications as high-quality audio amplifiers. The pulse width of the modulated output depends on the ratio of the peak reference signal to the peak carrier signal, commonly known as the modulation index $m$, as given by

$$m = \frac{V_r}{v_{cr}}$$

The characteristic of the $LC$ filter can be determined from the input–output relationship in the Laplace’s domain of $s$ as given by

$$G_f(s) = \frac{V_o(s)}{V_i(s)} = \frac{\omega_n^2}{s^2 + 2\delta \omega_n s + \omega_n^2}$$

where the natural resonant frequency is $\omega_n = 1/\sqrt{LC}$, and the damping factor $\delta = (1/2R)\sqrt{L/C}$. By substituting $s = j\omega$ in Eq. (11.91), we can find the filter transfer function in the frequency domain as given by

$$G_f(j\omega) = \frac{\omega_n^2}{(j\omega)^2 + j\omega 2\delta \omega_n + \omega_n^2} = \frac{\omega_n^2}{\omega_n^2 - \omega^2 + j\omega 2\delta \omega_n}$$

FIGURE 11.33  Class D amplifier
which can be expressed in polar form with normalized frequency ratio \( u = \omega/\omega_n \) as given by

\[
G_f(j\omega) = \frac{1}{1 - u^2 + j2\delta u} = \frac{1}{\sqrt{(1 - u^2)^2 + (2\delta u)^2}} \angle \tan^{-1}\left(\frac{1 - u^2}{2\delta u}\right) \tag{11.93}
\]

which shows that \( u = \omega/\omega_n \) affects the amount of phase shift present at the output. A high \( \omega_n \) will decrease the phase shift, whereas a low \( \omega_n \) will increase it. To minimize the amount of ripple present at the output, \( \omega_n \) has to be smaller than the carrier frequency \( \omega_c = 1/2\pi f_c \), and it should ideally be equal to the reference frequency \( \omega_c = 2\pi f_t \).

**EXAMPLE 11.9**

**Finding the values of \( L \) and \( C \) for a class D amplifier**  The gate voltage of the class D amplifier in Fig. 11.33(a) is generated by modulating a sinusoidal reference \( V_t = 1 \) V at \( f_t = 20 \) kHz with a carrier signal \( V_{cr} = 1 \) V at \( f_{cr} = 30 f_t \). The DC supply voltages are \( V_{DD} = \pm 5 \) V. The load resistance \( R_L = 8 \) \( \Omega \) speaker.

(a) Find the values of \( L \) and \( C \) to form a tuned resonant circuit.

(b) Use PSpice to plot the gate-modulating voltage \( v_G(t) \), the transistor output voltage \( v_D(t) \), and the transistor output voltage \( v_O(t) \) by a modulation index of \( m = 0.5 \) and 0.9. Use the PSpice parameters of the NMOS IRF150 whose \( V_t = 2.84 \) V and the PMOS IRF9140 whose \( V_t = 3.67 \) V.

**SOLUTION**

(a) The lower-order harmonic component is \( f_L = 2f_t = 2 \times 20 \times 10^3 = 40 \times 10^3 \). For the impedance of the filter capacitor, \( C \) must be much smaller than \( R_L \) so that the higher-order harmonic components pass through \( C \). Using a 10:1 ratio, we can write \( R_L >> 1/(2\pi f_L C) \) or

\[
C = \frac{10}{2\pi f_L R_L} = \frac{10}{2\pi \times 40 \times 10^3 \times 8} \approx 2.5 \ \mu F
\]

Let us make the filter cutoff frequency \( f_n = f_L = 1/(2\pi \sqrt{LC}) \), which gives \( L \) for \( C = 2.5 \) \( \mu F \) as

\[
L = \frac{1}{C(2\pi f_n)^2} = \frac{1}{2.5 \times 10^{-6}(2\pi \times 40 \times 10^3)^2} \approx 6.4 \ \mu H
\]

(b) The PSpice schematic is shown in Fig. 11.34. The PSpice plot of the gate modulating voltage \( v_G(t) \) is shown in Fig. 11.35(a), the transistor drain voltage \( v_D(t) \) in Fig. 11.35(b), and the output load voltage \( v_O(t) \) in Fig. 11.35(c) for \( m = 0.8 \) and 1.0. For \( v_G(t) < V_t = 2.84 \) V (NMOS) and \( v_G(t) > V_t = 3.67 \) V (PMOS), the transistors are off. The output voltage increases with the modulation index as expected. It should be noted that the transistor output is \( 2V_{DD} \). From the output file, we can find the THD of the output voltage, which is a measure of the waveform quality, by performing the Fourier analysis: THD = 4.847% for \( m = 0.8 \), and THD = 3.0% for \( m = 1 \).
Power Amplifiers

FIGURE 11.34 PSpice schematic for Example 11.9

FIGURE 11.35 PSpice voltages and currents of a class D amplifier for Example 11.9

(a) Gate modulated voltage

(b) Transistor drain voltage

(c) Output load voltage
As shown in Fig. 11.36, a class E amplifier is similar to the class D amplifier, except that a capacitor \( C_D \) is connected across the transistor. It uses only one transistor as a switch and has low switching losses, yielding a high efficiency of more than 95%. It is used for applications requiring less than 100 W. When the transistor is switched on, the current through the inductor \( L_D \) rises; when the transistor is turned off, \( L_D \) forms a resonant circuit with \( C_D \) and oscillates on. As a result, the transistor output is almost resonant voltage. To minimize switching losses and increase efficiency, the transistor should be gated when the resonance current through the inductor \( L_D \) reaches its minimum low (ideally zero). That means the transistor will be turned on at zero current and will be subjected to minimum switching losses, increasing the amplifier efficiency.

To obtain an almost sinusoidal current through the load, the values of \( L \) and \( C \) are chosen to have a high quality factor, \( Q \geq 7 \), and a low damping ratio, usually \( \delta \leq 0.072 \). The optimum parameters to give the maximum efficiency can be found from [7]

\[
L_D = \frac{0.4001R}{\omega_s} \tag{11.94}
\]

\[
C_D = \frac{2.165}{R\omega_s} \tag{11.95}
\]

\[
\frac{\omega_s L_D}{1 - \frac{1}{\omega_s C_D}} = 0.3533R \tag{11.96}
\]

**EXAMPLE 11.10**

**Finding the values of \( L_D \), \( C_D \), \( L \), and \( C \) for a class E amplifier** The gate voltage of the class E amplifier in Fig. 11.36 is a sinusoidal voltage of 4 V (peak) at 100 kHz. The DC supply voltages are \( V_{DD} = 12 \) V. The load resistance \( R = 10 \) \( \Omega \).

(a) Find the optimum values of \( L_D \), \( C_D \), \( L \), and \( C \).

(b) Use PSpice to plot the gate voltage \( v_G(t) \), the transistor drain current \( i_D(t) \), the transistor output voltage \( v_{DD}(t) \), and the output voltage \( v_O(t) \) for peak gate voltages of 4 V and 4.5 V. Use the PSpice parameters of the NMOS IRF150 whose \( V_t = 2.84 \) V.
SOLUTION

(a) For \( f_s = 100 \text{ kHz} \),

\[ \omega_s = 2\pi f_s = 2\pi \times 100 \text{ kHz} = 6.283 \times 10^5 \text{ rad/s} \]

From Eq. (11.94),

\[ L_D = \frac{0.4001 R}{\omega_s} = \frac{0.4001 \times 10}{6.283 \times 10^5} \approx 6.4 \text{ \mu H} \]

From Eq. (11.95),

\[ C_D = \frac{2.165}{R\omega_s} = \frac{2.165}{10 \times 6.283 \times 10^5} \approx 345 \text{ nF} \]

For choosing \( Q = 7 \),

\[ L_D = \frac{QR}{\omega_s} = \frac{7 \times 10}{6.283 \times 10^5} = 111 \text{ \mu H} \]

From Eq. (11.96),

\[ C_D = \frac{1}{(\omega_s L - 0.3533) R\omega_s} = \frac{1}{(6.283 \times 10^5 \times 111 \times 10^{-6} - 0.3533 \times 10) \times 6.283 \times 10^5} \approx 24 \text{ nF} \]

(b) The PSpice schematic is shown in Fig. 11.37. The PSpice plots of the gate voltage \( v_G(t) \) and the drain current \( i_D \) are shown in Fig. 11.38(a), the transistor output voltage \( v_D(t) \) and the inductor current in Fig. 11.38(b), and the output load voltage \( v_o(t) \) in Fig. 11.38(c) for peak gate voltages of 4 V and 4.5 V. For \( v_G(t) < V_i = 2.84 \text{ V} \) (MOS), the transistors are off. The output voltage increases with the gate voltage as expected, and the transistor output is \( 2V_{DD} \). From the output file, we can find the THD of the output voltage, which is a measure of the waveform quality, by performing the Fourier analysis: THD = 4.279% for \( V_{val} = 4 \text{ V} \) and THD = 3.078% for \( V_{val} = 4.5 \text{ V} \).

\[ v_G = \begin{cases} \text{IRF150} & \text{for } V_{off} = 0 \\ \text{VAMPL} = [\text{Val}] & \text{for } 100 \text{ kHz} \end{cases} \]

\[ v_{o} = \begin{cases} \text{CD} & \text{for } 345 \text{ nF} \\ \text{R} & \text{for } 10 \end{cases} \]

\[ \text{FIGURE 11.37} \quad \text{PSpice schematic for Example 11.10} \]
11.10 Short-Circuit and Thermal Protection

An output stage is normally protected against short-circuiting and excessive temperature rise. A class AB amplifier with such protection is shown in Fig. 11.39.

11.10.1 Short-Circuit Protection

The circuit used in Fig. 11.39 for short-circuit protection consists of transistor Q1 and resistor RE1. If a short circuit occurs at the load while QN is conducting, a large current will flow through RE1, and a voltage V_{RE1} proportional to the short-circuit current will develop across RE1. When voltage V_{RE1} becomes large enough, transistor Q1 will turn on and carry most of the biasing current IR1. Thus, the base current of QN will be reduced to a safe level. The voltage drops across the emitter resistors will reduce the output
voltage by the same amount. Therefore, the values of $R_{E1}$ and $R_{E2}$ should be as low as possible (on the order of milliohms). Their values are determined from

$$R_{RE1} = R_{RE2} = \frac{V_{BE1}}{i_{O\text{(short)}}}$$

(11.97)

where $i_{O\text{(short)}}$ is the permissible short-circuiting current to turn on transistor Q1. For $V_{BE1} = 0.7$ V and $i_{O\text{(short)}} = 200$ mA, $R_{RE1} = R_{RE2} = 0.7 \div 200$ mA $= 3.5$ $\Omega$. Although $R_{E1}$ and $R_{E2}$ reduce the output voltage swing, they will give biasing stability to the quiescent current $I_Q$ and protect $Q_N$ and $Q_P$ against thermal runaway (i.e., excessive junction temperature rise).

### 11.10.2 Thermal Protection

The circuit used in Fig. 11.39 for thermal protection consists of two transistors ($Q_2$ and $Q_3$), three resistors ($R_1$, $R_2$, and $R_3$), and a zener diode. Normally, transistor $Q_2$ is off. If the temperature rises, $V_{BE3}$ will fall because of the negative temperature coefficient of $Q_3$, and the zener voltage $V_Z$ will rise because of the positive temperature coefficient of zener diode $D_Z$. As a result, the voltage at the emitter of $Q_3$ will rise, and the voltage at the base of $Q_2$ will also rise. If the temperature rise is adequate, $Q_2$ will turn on, diverting the reference current $I_R$ from the amplifier and hence shutting down the amplifier. The maximum permissible temperature rise $\Delta T$ can be determined from

$$[\Delta T(K_{DZ} + K_{Q3}) + V_{BE3}] \frac{R_1}{R_1 + R_2} = V_{BE2} = V_T \ln \left( \frac{I_R}{I_S} \right)$$

(11.98)

where $K_{DZ} =$ temperature coefficient of the zener diode, in V $\degree$C

$K_{Q3} =$ temperature coefficient of transistor $Q_3$, in V $\degree$C

$V_{BE3} =$ quiescent B-E voltage of $Q_3$
11.11 Power Op-Amps

Op-amps have some desirable characteristics, such as a very high open-loop gain ($>10^5$), a very high input impedance (up to $10^9\,\Omega$), and a very low input biasing current. However, the AC output power of op-amps is generally low. High power can be obtained from a power amplifier consisting of an op-amp followed by a class AB buffer. The general structure of a power op-amp is shown in Fig. 11.40. The buffer stage consists of transistors $Q_1$, $Q_2$, $Q_3$, and $Q_4$. $R_1$ and $R_2$ bias transistors $Q_1$ and $Q_2$ such that $-V_{BE1} + V_{BE3} \approx 0$ and $V_{BE2} - V_{BE4} = 0$. Transistor $Q_3$ supplies the positive load current until the voltage across $R_3$ is sufficiently large to turn on $Q_5$. Then $Q_5$ supplies additional load current. Similarly, transistors $Q_4$ and $Q_6$ supply the negative load current. The stage formed by $Q_5$ and $Q_6$ supplies the additional load current and acts as a current booster. Emitter resistors $R_{E1}$ and $R_{E2}$ are used for biasing stability.

![Diagram of a power op-amp](image.png)

**FIGURE 11.40** General structure of a power op-amp
11.11.1 IC Power Amplifiers

A variety of power amplifiers that combine a conventional op-amp chip with a current booster are available commercially. Some have internal negative feedback already applied to give a fixed closed-loop voltage gain; others do not have on-chip feedback. We will consider two such representative op-amps: the LH0021 op-amp and the LM380 op-amp, both manufactured by National Semiconductor.

Power Op-Amp LH0021

The schematic of power op-amp LH0021 is shown in Fig. 11.41. The LH0021 is designed to operate from a power supply of ±25 V. It is capable of a peak output voltage swing of about 12 V into a 10-Ω load over the entire frequency range of 15 kHz. The distortion of the output voltage is less than 1.6%. The circuit can be divided into three stages: a differential stage, a gain stage, and an output stage.

The differential input stage consists of transistors Q1 through Q4 biased by Q7, which sinks the base currents of Q3 and Q4. Transistors Q5 and Q6 serve as the current mirror active load.
The gain stage is a common-emitter configuration and consists of Q9 and Q10 connected as a Darlington pair. Transistor Q8 serves as an active current source for this stage. Capacitor C1 is the pole-splitting compensating capacitor connected in shunt-shunt feedback.

The output stage is a complementary class AB push-pull circuit. It consists of transistors Q13, Q14, Q15, Q16, Q17, and Q18. Diodes D3 and D4 provide the biasing voltage for class AB operation in order to minimize crossover distortion. Transistors Q13 and Q14 act as the current booster. Resistors RC1 and RC2 limit the currents through Q13 and Q14, respectively, by turning on Q15 and Q16. Resistor R1 protects Q17 and Q18 by limiting the current flow through them.

A small external capacitor CC is connected to offer a low impedance to a capacitive load. The combination of R1 and the load capacitor does not form a low-pass RC network, and any phase delay of the output voltage can be avoided.

### Power Op-Amp LM380

The schematic of power op-amp LM380 is shown in Fig. 11.42. The LM380 is designed to operate from a single power supply in the range of 12 V to 22 V. The output power can be as high as 5 W onto a 10-Ω load. The distortion of the output voltage is less than 3%. The circuit can be divided into three stages: a differential stage, a gain stage, and an output stage. An external capacitor Cx can be used to bypass the current source to improve the low-frequency response.

The differential input stage consists of pnp transistors Q3 through Q6. Transistor Q3 is biased by Q10, whereas transistor Q4 is biased by a DC current from the output terminal through R2. Under quiescent conditions (i.e., with an input voltage), the biasing currents for Q3 and Q4 will be equal. Thus, the current through and the voltage across R3 will be zero. Transistors Q5 and Q6 serve as the current

![Power Op-Amp LM380 Schematic](image)

**FIGURE 11.42** Power op-amp LM380 (Courtesy of National Semiconductor, Inc.)
mirror active load for this stage. The pnp transistors Q1 and Q2 act as emitter followers for input buffering. Resistors R1 and R2 provide DC paths to the ground for the base currents of Q3 and Q4.

The gain stage consisting of Q12 has the common-emitter configuration. Transistor Q11 serves as a current source active load for the gain stage. Capacitor C is the pole-splitting compensating capacitor intended to yield a wide bandwidth.

The output stage is a quasi-complementary class AB push-pull circuit. It consists of transistors Q8, Q9, and Q6. Diodes D1 and D2 provide the biasing voltage for class AB operation. Emitter resistors R6 and R7 give biasing stability.

Resistor R2 provides DC feedback from the DC output voltage \(v_O\) to the emitter of Q4. If for some reason \(v_O\) increases, then there will be a corresponding increase in the current through R2 and the emitter current \(I_{E4}\) of Q4, causing an increase in the collector current of Q4. As a result, the voltage at the base of Q12 and its base current will increase. This, in turn, will increase the collector current of Q12 and reduce the base current of Q9. Thus, \(v_O\) will be reduced.

To find \(v_O\), let us assume that all transistors are identical and the base currents are negligible compared to the emitter currents. The emitter biasing current of Q3 can be found approximately from

\[
I_{E3} = \frac{V_{CC} - V_{EB10} - V_{EB3} - V_{EB1}}{R_1} = \frac{V_{CC} - 3V_{EB}}{R_1} \tag{11.99}
\]

Also, the emitter current of Q4 can be found from

\[
I_{E4} = \frac{v_O - V_{EB4} - V_{EB2}}{R_2} = \frac{v_O - 2V_{EB}}{R_2} \tag{11.100}
\]

where \(v_O\) is the DC output voltage. For \(I_{E3} = I_{E4}\) (so that no current flows through R3), we get

\[
\frac{V_{CC} - 3V_{EB}}{R_1} = \frac{v_O - 2V_{EB}}{R_2}
\]

which, for \(R_1 = 2R_2\), as shown in Fig. 11.42, gives the DC output voltage as

\[
v_O = \frac{V_{CC} + V_{EB}}{2} = \frac{V_{CC} - V_{BE}}{2} \tag{11.101}
\]

Thus, for \(V_{BE} << V_{CC}\), which is usually the case, the DC output voltage is approximately half the supply voltage \(V_{CC}\); that is, \(v_O \approx V_{CC}/2\).

### 11.11.2 Bridge Amplifier

The output power can be doubled by using two power op-amps, as shown in Fig. 11.43. This arrangement, called a bridge amplifier, is commonly used in high-power applications. The input voltage \(v_I\) is applied to the noninverting input of one amplifier and also to the inverting input of the other, so that the output voltages are 180° out of phase. Thus, the output of the noninverting amplifier is

\[
v_{O1} = \left(1 + \frac{R_F}{R_1}\right)v_I \tag{11.102}
\]

The output voltage of the inverting amplifier is

\[
v_{O2} = -\frac{R_3}{R_2}v_I \tag{11.103}
\]
The voltage across the load becomes

\[ v_O = v_{O1} - v_{O2} = \left( 1 + \frac{R_F}{R_1} \right) v_1 + \frac{R_3}{R_2} v_1 \]

which, for \( R_2 = 1 + R_F \) and \( R_1 = A_f \), becomes

\[ v_O = 2A_f v_1 \]  \hspace{1cm} (11.104)

where \( A_f \) is the closed-loop voltage gain of each amplifier.

**KEY POINTS OF SECTION 11.11**

- IC power amplifiers are known as power op-amps. They consist of a differential stage, a gain stage, and an output stage with a current booster.
- Power op-amps are normally used with feedback and are compensated for frequency response by internal capacitance in the gain stage.

**11.12 Thermal Considerations**

Power transistors dissipate a large amount of power. The power dissipation is converted to heat, which causes the temperature of the collection junction to rise. The physical structure, packaging, and specifications of transistors differ depending on their current-handling capability and power dissipation. The current rating of power transistors can go as high as 500 A, with power dissipation of up to 200 W, especially when the transistors are used as switching elements for power converters [6]. Power transistors must be protected from excessive temperature rise. The junction temperature \( T_J \) must be kept within a specified maximum \( T_{J(max)} \) to avoid damage to the transistor. For silicon transistors, \( T_{J(max)} \) is in the range of 150°C to 200°C.
11.12.1 Thermal Resistance

If a transistor operates in the open air without any cooling arrangement, the heat will be transferred from the transistor junction to the ambient. Thermal resistance is a measure of the heat transfer. It is the temperature drop divided by power dissipation under steady-state conditions. Thus, the units are °C/W. The thermal resistance $\theta_{JA}$ for heat flow from the junction to the ambient is given by

$$ \theta_{JA} = \frac{T_J - T_A}{P_D} \quad \text{(in °C/W)} \quad (11.105) $$

where $T_J =$ junction temperature, in °C
$T_A =$ ambient temperature, in °C

Equation (11.105) represents the heat-transfer process and is analogous to Ohm’s law. Power dissipation corresponds to current, temperature difference to voltage difference, and thermal resistance to electrical resistance. Thus, the thermal process can be represented by an analogous electrical circuit, as shown in Fig. 11.44.

11.12.2 Heat Sink and Heat Flow

To keep the junction temperature below $T_{J_{\text{max}}}$, the transistor is normally mounted on a heat sink, which facilitates the removal of heat from the device to the surrounding air. Typical heat sinks with devices attached are shown in Fig. 11.45. Heat is transferred from the device to the air by one of three methods:

1. Conduction from junction to case with thermal resistance $\theta_{JC}$ and from case to heat sink with thermal resistance $\theta_{CS}$. The values of $\theta_{JC}$ and $\theta_{CS}$ depend on the cross section, length, and temperature difference across the conducting medium. The imperfect matching of adjacent surfaces will increase $\theta_{CS}$. 

![FIGURE 11.44 Electrical equivalent of thermal process](#)

![FIGURE 11.45 Transistors mounted on heat sinks](#)
The value of $\theta_{CS}$ can be made small by coating the mated surfaces of the transistor and sink with a thermally conducting compound.

2. Convection from case to ambient with thermal resistance $\theta_{CA}$ and from heat sink to ambient with thermal resistance $\theta_{SA}$. The values of $\theta_{CA}$ and $\theta_{SA}$ depend on surface condition, type of convecting fluid, velocity and characteristic of the fluid, and temperature difference between surface and fluid.

3. Radiation from cooling fins to the air. The heat transfer will depend on surface emissivity and area, as well as temperature difference between the radiating fins and the air.

The thermal equivalent circuit of a transistor on a heat sink is shown in Fig. 11.46. The power dissipation is related to the junction temperature $T_J$ and the ambient temperature $T_A$ by

$$T_J - T_A = P_D(\theta_{JC} + \theta_{CS} + \theta_{SA})$$  \hspace{1cm} (11.106)

where

- $\theta_{JC}$ = thermal resistance from junction to case, in °C/W
- $\theta_{CS}$ = thermal resistance from case to sink, in °C/W
- $\theta_{SA}$ = thermal resistance from sink to ambient, in °C/W

The values of $\theta_{JC}$ and $\theta_{CS}$ are specified in the manufacturer’s data sheet. The circuit designer has to specify the required value of $\theta_{SA}$ for the heat sink. The value of $\theta_{SA}$ found from Eq. (11.106) may be too small to match any standard heat sink with natural air cooling. In this case, it might be necessary to cool the heat sink using liquid or forced air from a fan.

### 11.12.3 Power Dissipation versus Temperature

The ambient temperature $T_A$ and the case temperature $T_C$ are related to the power dissipation $P_D$ by

$$T_C - T_A = P_D(\theta_{CS} + \theta_{SA})$$  \hspace{1cm} (11.107)

The transistor manufacturer normally specifies the maximum junction temperature $T_J(\text{max})$, the maximum power dissipation $P_D(\text{max})$ at a specified case temperature $T_C(0)$ (usually 25°C), and the thermal resistance $\theta_{JC}$. The manufacturer often provides a power-derating curve, as shown in Fig. 11.47, which gives the maximum allowable power dissipation if the case temperature is higher than 25°C. $T_C(0)$ is the case temperature...
at which the derating begins. \( T_{C_{\text{max}}} \) is the maximum value of the case temperature in degrees Celsius and is equal to \( T_{J_{\text{max}}} \). The power dissipation at a case temperature of \( T_{C} \) can be found from

\[
P_D(T = T_{C}) = P_{D(\text{max})} - \frac{P_{D(\text{max})}}{\theta_{J_{\text{max}}} + T_{C_{0}}} (T_{C} - T_{C_{0}}) \quad \text{for } T_{C} \geq T_{C_{0}}
\]

**EXAMPLE 11.11**

**Finding the power dissipation of a transistor**  The power dissipation of a transistor is specified as 

\[
P_{D(\text{max})} = 150 \text{ W at } T_{C_{0}} = 25^\circ\text{C}.
\]

The transistor is mounted on a heat sink. The thermal resistances are \( \theta_{JC} = 0.5^\circ\text{C/W} \), \( \theta_{CS} = 0.2^\circ\text{C/W} \), and \( \theta_{SA} = 1.5^\circ\text{C/W} \). If \( T_{J_{\text{max}}} = 200^\circ\text{C} \) and \( T_{A} = 45^\circ\text{C} \), calculate the maximum permissible power dissipation of the transistor.

**SOLUTION**

We can find the maximum power dissipation \( P_D \) from Eq. (11.106):

\[
200 - 45 = P_D(0.5 + 0.2 + 1.5) = 2.2P_{D(\text{max})}
\]

which gives

\[
P_D = \frac{200 - 45}{2.2} = 70.5 \text{ W}
\]

From \( P_D = 70.5 \text{ W} \), Eq. (11.107) gives the case temperature as

\[
T_{C} = 45 + 70.5 \times (0.2 + 1.5) = 164.9^\circ\text{C}
\]

The corresponding power dissipation becomes

\[
P_D(T_{C} = 164.9^\circ\text{C}) = 150 - \frac{150}{200 - 25} \times (164.9 - 25) = 30.1 \text{ W}
\]

In other words, for \( P_D = 70.5 \text{ W}, T_{C} = 164.9^\circ\text{C} \), which in turn limits the power dissipation to 30.1 W. Therefore, \( P_D \) cannot be 70.5 W. We need to find the actual power. Substituting \( T_{C} \) from Eq. (11.107) into Eq. (11.108) yields

\[
P_D = R_{D(\text{max})} - \frac{R_{D(\text{max})}}{\theta_{J_{\text{max}}} + T_{C_{0}}} [T_{A} + R_{D(\text{max})}(\theta_{CS} + \theta_{SA})]
\]
which gives

\[ P_D \left[ 1 + \frac{P_{D(\text{max})}}{T_{J(\text{max})} - T_{C0}} (\theta_{CS} + \theta_{SA}) \right] = P_{D(\text{max})} \left[ 1 - \frac{T_A}{T_{J(\text{max})} - T_{C0}} \right] \]

Since

\[ P_D \left[ 1 + \frac{150}{200 - 25} (0.2 + 1.5) \right] = 150 \times \left[ 1 - \frac{45}{200 - 25} \right] \]

the permissible power dissipation is \( P_D = 45.35 \) W.

**NOTE** \( \theta_{JC} = P_{D(\text{max})} / (T_{J(\text{max})} - T_{C0}) = 150 \) \((200 - 25) = 0.857^\circ \) C. But in Example 11.11, \( \theta_{JC} = 0.5^\circ \) C \ W was specified in order to illustrate the method for finding the actual power dissipation.

**KEY POINTS OF SECTION 11.12**

- Power dissipates from a transistor in the form of heat, which causes the temperature of the collection junction to rise. Power transistors or power op-amps are normally mounted on a heat sink, which provides a low thermal impedance for the heat flow.
- The maximum power dissipation is specified by the manufacturer at a specified case temperature \( T_{C0} \) (usually 25°C), which is not normally attainable. It is often necessary to determine the allowable power dissipation from a derating curve.

### 11.13 Design of Power Amplifiers

Since class B and class AB amplifiers eliminate the dead zone, they are generally used as the output stages of practical amplifiers. Thus, the design of a power amplifier is mainly the design of the output stage, and it involves the following steps:

**Step 1.** Identify the specifications of the output stage (e.g., output power \( P_L \), load resistance \( R_L \), and DC supply voltages \( V_{CC} \) and \( V_{EE} \) or \( V_{DD} \) and \( V_{SS} \)).

**Step 2.** Select the type of output operation, usually class B or class AB operation.

**Step 3.** Determine the voltage and current ratings of all transistors.

**Step 4.** Determine the values and power ratings of all resistors. Also, determine the turns ratio(s) of transformers in case of a transformer-coupled load.

**Step 5.** Select the type of DC-biasing circuit. Determine the specifications of active and passive components.

**Step 6.** Select the power transistors that will meet the voltage, current, and power requirements. Find their maximum junction temperature \( T_{J(\text{max})} \) and thermal resistances \( \theta_{JC} \) and \( \theta_{CS} \).

**Step 7.** Determine the power dissipation of the transistors, and find the desired thermal resistance of the heat sink.

**Step 8.** Use PSpice/SPICE to simulate and verify your design, using the standard values of components with their tolerances.
Summary

A power amplifier generally forms the output stage of an audio-frequency amplifier. The requirements of power amplifiers are different from those of small-signal low-power amplifiers. Power amplifiers must deliver an appreciable amount of power to a low-impedance load, while at the same time creating very little distortion in the output signal. Power amplifiers are generally classified into six types: class A, class B, class AB, class C, class D, and class E. The efficiency of a class A amplifier is only 25%, that of a class B type is 50%, and that of a class AB push-pull type is 78.5%. Class D and E amplifiers can give 90% or higher efficiency. Complementary class AB push-pull amplifiers eliminate or reduce the distortion and dead zone in the output signal. The output stage is normally biased by an active current source in order to supply quiescent biasing current at zero input signal. A quasi-complementary amplifier, which uses a composite pnp transistor consisting of a pnp and an npn transistor, can increase the output power. Power IC op-amps generally consist of an op-amp followed by a class AB buffer. A power transistor must be mounted on a heat sink to limit the junction temperature to an acceptable value.

References


Review Questions

1. What are the six types of power amplifiers?
2. What are the major differences among the six types of power amplifiers?
3. What are the advantages of an emitter follower with active current–source biasing?
4. What is the limiting design condition for avoiding clipping on the output voltage?
5. What is a figure of merit for an amplifier?
6. What is a common-emitter class A amplifier?
7. What are the advantages and disadvantages of a common-emitter class A amplifier?
8. What is the maximum efficiency of a common-emitter class A amplifier?
9. What is a figure of merit for a common-emitter class A amplifier?
10. What is a transformer-coupled load class A amplifier?
11. What are the advantages and disadvantages of a transformer-coupled load class A amplifier?
12. What is the maximum efficiency of a transformer-coupled load class A amplifier?
13. What is a figure of merit for a transformer-coupled load class A amplifier?
14. What is a complementary class B push-pull amplifier?
15. What are the advantages and disadvantages of a complementary class B push-pull amplifier?
16. What is the maximum efficiency of a complementary class B push-pull amplifier?
17. What is a figure of merit for a complementary class B push-pull amplifier?
18. What is the cause of crossover distortion in a complementary class B push-pull amplifier?
19. What is a transformer-coupled load class B push-pull amplifier?
20. What are the advantages and disadvantages of a transformer-coupled load class B push-pull amplifier?
21. What is the maximum efficiency of a transformer-coupled load class B push-pull amplifier?
22. What is a figure of merit for a transformer-coupled load class B push-pull amplifier?
23. What are the methods of eliminating or minimizing crossover distortion in a complementary class B push-pull amplifier?
24. What is a complementary class AB push-pull amplifier?
25. What are the advantages and disadvantages of a complementary class AB push-pull amplifier?
26. What is the maximum efficiency of a complementary class AB push-pull amplifier?
27. What is a figure of merit for a complementary class AB push-pull amplifier?
28. What are the methods for DC biasing an output stage?
29. What are the methods for providing short-circuit and thermal protection for an output stage?
30. What is an IC power amplifier?
31. What is the thermal equivalent circuit of a transistor?
32. What is the power derating curve of a power transistor?
33. What is the purpose of a heat sink for a power transistor?
34. What is class C amplifier?
35. What is class D amplifier?
36. What is class E amplifier?

Problems

The symbol $\square$ indicates that a problem is a design problem. The symbol $\mathbb{P}$ indicates that you can check the solution to a problem using PSpice/SPICE or Electronics Workbench.

Assume that the PSpice model parameters for the diodes are

$\text{IS}=100E-15 \quad \text{BV}=100 \quad \text{IBV}=100E-13$

and for the transistors are

$\text{IS}=100E-15 \quad \text{IBV}=100E-14 \quad \text{BF}=100 \quad \text{VJE}=0.8 \quad \text{VA}=100$

(unless specified).

11.4 Class A Amplifiers

11.1 a. Design an emitter follower as shown in Fig. 11.3(b). Assume $V_{CC} = 15 \text{ V}$, $V_{BE} = 0.7 \text{ V}$, $V_{CE(sat)} = 0.5 \text{ V}$, $I_R = 10 \text{ mA}$, and $R_L = 1 \text{ k}\Omega$. Assume identical transistors of $\beta_F = h_{fe} = 100$. Find the voltage, current, and power rating of the transistors.

$\square$ b. Use PSpice/SPICE to check your design by plotting the transfer function.

11.2 For the emitter follower in Prob. 11.1, determine the critical value of load resistance to avoid clipping (or distortion), and calculate the peak-to-peak output voltage swing $V_{pp}$ if $R_L = 2 \text{ k}\Omega$.

11.3 The parameters of the emitter follower in Fig. 11.3(b) are $V_{CC} = 12 \text{ V}$, $R_1 = 2.5 \text{ k}\Omega$, and $R_L = 750 \text{ \Omega}$. The transistors are identical, and their parameters are $V_{BE} = 0.7 \text{ V}$, $V_{CE(sat)} = 0.5 \text{ V}$, and $\beta_F = h_{fe} = 100$. Determine (a) the peak voltage and current of transistor $Q_1$ and (b) the average output power $P_L$. 
11.4 For the emitter follower in Prob. 11.3, calculate (a) the power efficiency, (b) the value of $R_L$ for maximum efficiency, and (c) the corresponding value of efficiency $\eta_{\text{max}}$. 

11.5 a. Design a class A amplifier, as shown in Fig. 11.7(a), for high efficiency. The audio output power is 48 W. The available supply voltage is 12 V. Assume $\beta_F = h_{fe} = 100$. Find the voltage, current, and power rating of the transistors.

b. Use PSpice/SPICE to check your design by plotting the collector current and voltage.

c. The parameters of the transformer-coupled load class A amplifier in Fig. 11.10(a) are $V_{\text{CC}} = 15$ V, $R_1 = 15$ kΩ, and $R_L = 1$ kΩ. The transistors are identical, and their parameters are $V_{BE} = 0.7$ V, $V_{CE(sat)} = 0.5$ V, and $\beta_F = h_{fe} = 100$. Determine (a) the peak voltage and current of transistor Q1, (b) the average output power $P_L$, and (c) the efficiency $\eta$.

11.6 a. Design a transformer-coupled load class A amplifier, as shown in Fig. 11.17(a), to supply an output power of $P_L = 32$ W at a load resistance of $R_L = 8$ Ω. Assume a DC supply voltage of $V_{\text{CC}} = 15$ V. Find the voltage, current, and power rating of the transistors and the transformer turns ratio. Assume $\beta_F = 100$ and $V_{CE(sat)} = 0.5$ V.

b. Use PSpice/SPICE to check your design by plotting the collector current and voltage.

c. The parameters of the transformer-coupled load class A amplifier in Fig. 11.10(a) are $V_{\text{CC}} = 15$ V and $R_L = 16$ Ω. The transistor parameters are $V_{BE} = 0.7$ V, $V_{CE(sat)} = 0.5$ V, and $h_{fe} = 100$. The transformer turns ratio is 2:1. Determine (a) the peak voltage and current of transistor Q1, (b) the average output power $P_L$, and (c) the efficiency $\eta$.

11.7 Class B Push-Pull Amplifiers

11.8 The parameters of the transformer-coupled load class A amplifier in Fig. 11.17(a) are $V_{\text{CC}} = 15$ V and $R_L = 10$ Ω. The transistor parameters are $V_{BE} = 0.7$ V, $V_{CE(sat)} = 0.5$ V, and $h_{fe} = 100$. Find the voltage, current, and power rating of the transistors and the transformer turns ratio. Assume $\beta_F = 100$ and $V_{CE(sat)} = 0.5$ V.

b. Use PSpice/SPICE to check your design by plotting the collector current and voltage.

c. Design a class B push-pull amplifier, as shown in Fig. 11.11(a), to supply an output power of $P_L = 16$ W at a load resistance of $R_L = 4$ Ω. Assume a DC supply voltage of $V_{\text{CC}} = 15$ V and transistors of $\beta_F = h_{fe} = 100$ and $V_{BE} = 0.7$ V. Find the voltage, current, and power rating of the transistors.

b. Use PSpice/SPICE to check your design by plotting the transfer function, the output voltage, and the load current.

11.9 a. Design a complementary class B push-pull amplifier, as shown in Fig. 11.11(a), to supply an output power of $P_L = 16$ W at a load resistance of $R_L = 4$ Ω. Assume a DC supply voltage of $V_{\text{CC}} = 15$ V and transistors of $\beta_F = h_{fe} = 100$ and $V_{BE} = 0.7$ V. Find the voltage, current, and power rating of the transistors.

b. Use PSpice/SPICE to check your design by plotting the transfer function, the output voltage, and the load current.

c. Calculate the power efficiency $\eta$ and power dissipation $P_D$ of each transistor in the complementary class B push-pull output stage in Fig. 11.11(a) if $V_{\text{CC}} = 15$ V and $R_L = 10$ Ω. The parameters of the transistors are $\beta_F = h_{fe} = 150$, $V_{CE(sat)} = 0.2$ V, and $V_{BE} = 0.7$ V.

11.10 Calculate the power efficiency $\eta$ and power dissipation $P_D$ of each transistor in the complementary class B push-pull output stage in Fig. 11.11(a) if $V_{\text{CC}} = 15$ V and $R_L = 10$ Ω. The parameters of the transistors are $\beta_F = h_{fe} = 100$, $V_{CE(sat)} = 0.2$ V, and $V_{BE} = 0.7$ V.

c. Design a transformer-coupled class B push-pull amplifier, as shown in Fig. 11.17(a), to supply an output power of $P_L = 32$ W at a load resistance of $R_L = 8$ Ω. Assume a DC supply voltage of $V_{\text{CC}} = 15$ V and transistors of $\beta_F = h_{fe} = 100$ and $V_{BE} = 0.7$ V.

11.11 The parameters of the transformer-coupled class B push-pull amplifier in Fig. 11.17(a) are $V_{\text{CC}} = 15$ V and $R_L = 8$ Ω. The transistor parameters are $V_{BE} = 0.7$ V, $V_{CE(sat)} = 0.5$ V, and $\beta_F = h_{fe} = 100$. The transformer turns ratio is 2:1. Determine (a) the peak voltage and current of transistor Q1, (b) the average output power $P_L$, and (c) the efficiency $\eta$.

11.12 Complementary Class AB Push-Pull Amplifiers

11.13 a. Design a biasing circuit for the class AB amplifier of Fig. 11.20(c) to supply a maximum output power of $P_{L(\text{max})} = 20$ W. The quiescent biasing current $I_Q$ is 2 mA. Assume a DC supply voltage of $V_{\text{CC}} = 15$ V. The diode parameters are $I_S = 10^{-13}$ A and $I_{D(\text{min})} = 1$ mA to ensure conduction. The transistor parameters are $h_{fe} = 50$, $V_{BE} = 0.7$ V, and $V_{CE(sat)} = 0.2$ V.

b. Find the biasing voltage $V_{BB}$ for $V_O = 0$ and 11.8 V.
11.16 a. Design an active current source for the class AB amplifier in Fig. 11.21(a) in order to provide a biasing current of $I_R = 10 \text{ mA}$. Assume $V_{CC} = 15 \text{ V}$, $V_{BE} = 0.7 \text{ V}$, and $R_L = 50 \text{ } \Omega$. The diode parameters are $I_S = 10^{-13} \text{ A}$ and $I_D(\text{min}) = 1 \text{ mA}$ to ensure conduction. The transistor parameters are $h_{fe} = 50$, $V_{BE} = 0.7 \text{ V}$, and $V_{CE(sat)} = 0.2 \text{ V}$.

b. Use PSpice/SPICE to plot the transfer characteristic and the instantaneous $i_N$, $i_P$, and $i_O$ for $v_I = 5 \sin (2000\pi t)$.

b. Use PSpice/SPICE to plot the transfer characteristic and the instantaneous $i_N$, $i_P$, and $i_O$ for $v_I = 5 \sin (2000\pi t)$.

11.17 a. Design a $V_{BE}$ multiplier for the class AB amplifier in Fig. 11.24 in order to provide a biasing current of $I_R = 10 \text{ mA}$. Assume $V_{CC} = 15 \text{ V}$, $V_{BE} = 0.7 \text{ V}$, and $R_L = 50 \text{ } \Omega$. Assume a minimum current of $I_M(\text{min}) = 1 \text{ mA}$ to the multiplier and $I_Q = 2 \text{ mA}$. The transistor parameters are $\beta_f = h_{fe} = 50$, $V_{BE} = 0.7 \text{ V}$, and $V_{CE(sat)} = 0.2 \text{ V}$.

b. Use PSpice/SPICE to plot the transfer characteristic and the instantaneous $i_N$, $i_P$, and $i_O$ for $v_I = 5 \sin (2000\pi t)$.

11.7–11.9 Class C, D, and E Amplifiers

11.18 The gate voltage of the class C amplifier in Fig. 11.30(a) is $v_G = 2.5 + 1.0 \sin (2\pi f_c t)$ with $f_c = 1.5 \text{ MHz}$. The DC supply voltages are $V_{DD} = \pm 15 \text{ V}$. The load resistance $R_L = 50 \text{ } \Omega$, which is connected through a transformer of a turns ratio of $n = 1:1$.

a. Find the values of $L$ and $C$ to form a tuned resonant circuit.

b. Use PSpice to plot the voltage $v_d(t)$, the transistor drain current $i_D(t)$, and the transistor drain voltage $v_D(t)$ for varying the gain voltage component $V_m = 3 \text{ V}$ to $4 \text{ V}$ with an increment of 0.5 V.

c. Use the PSpice parameters of the NMOS IRF150 whose $V_I = 2.84 \text{ V}$.

11.19 The gate voltage of the class C amplifier in Fig. 11.30(a) is $v_G = 2.5 + 1.0 \sin (2\pi f_c t)$ with $f_c = 1.5 \text{ MHz}$. The DC supply voltages are $V_{DD} = \pm 15 \text{ V}$. The load resistance $R_L = 150 \text{ } \Omega$, which is connected through a transformer of a turns ratio of $n = 2:1$.

a. Find the values of $L$ and $C$ to form a tuned resonant circuit.

b. Use PSpice to plot the voltage $v_d(t)$, the transistor drain current $i_D(t)$, and the transistor drain voltage $v_D(t)$ for varying the gain voltage component $V_m = 3 \text{ V}$ to $4 \text{ V}$ with an increment of 0.5 V.

c. Use the PSpice parameters of the NMOS IRF150 whose $V_I = 2.84 \text{ V}$.

11.20 The gate voltage of the class C amplifier in Fig. 11.30(a) is $v_G = 2.5 + 1.5 \sin (2\pi f_c t)$ with $f_c = 1.5 \text{ MHz}$. The DC supply voltages are $V_{DD} = \pm 15 \text{ V}$. The load resistance $R_L = 50 \text{ } \Omega$, which is connected through a transformer of a turns ratio of $n = 2:1$.

a. Find the values of $L$ and $C$ to form a tuned resonant circuit.

b. Use PSpice to plot the voltage $v_d(t)$, the transistor drain current $i_D(t)$, and the transistor drain voltage $v_D(t)$ for varying the gain voltage component $V_m = 3 \text{ V}$ to $4 \text{ V}$ with an increment of 0.5 V.

c. Use the PSpice parameters of the NMOS IRF150 whose $V_I = 2.84 \text{ V}$.

11.21 The gate voltage of the class D amplifier in Fig. 11.33(a) is generated by modulating a sinusoidal reference $V_r = 1 \text{ V}$ at 25 kHz with a carrier signal $V_{cr} = 1 \text{ V}$ at $f_{cr} = 25f_c$. The DC supply voltages are $V_{DD} = \pm 5 \text{ V}$. The load resistance $R_L = 8 \text{ } \Omega$ speaker.

a. Find the values of $L$ and $C$ to form a tuned resonant circuit.

b. Use PSpice to plot the gate-modulating voltage $v_G(t)$, the transistor output voltage $v_D(t)$, and the transistor output voltage $v_D(t)$ by a modulation index of $m = 0.5$ and 0.9.

c. Use the PSpice parameters of the NMOS IRF150 whose $V_I = 2.84 \text{ V}$ and the PMOS IRF9140 whose $V_I = 3.67 \text{ V}$.

11.22 The gate voltage of the class D amplifier in Fig. 11.33(a) is generated by modulating a sinusoidal reference $V_r = 1 \text{ V}$ at 40 kHz with a carrier signal $V_{cr} = 1 \text{ V}$ at $f_{cr} = 30f_c$. The DC supply voltages are $V_{DD} = \pm 5 \text{ V}$. The load resistance $R_L = 16 \text{ } \Omega$ speaker.
11.10 Short-Circuit and Thermal Protection

11.27 Design a thermal protection circuit as shown in Fig. 11.39 to limit the maximum temperature rise to \( \Delta T = 110^\circ C \). The temperature coefficients of the diode and the transistors are \( K_{DZ} = K_{Q3} = 2.5 \text{ mV } ^\circ C \). The normal biasing current is \( I_R = 5 \text{ mA} \), and the collector current of Q3 is 1 mA. Assume \( V_{BEQ2} = 0.7 \text{ V} \) and \( V_{CC} = 15 \text{ V} \).

11.11 Power Op-Amps

11.28 Use LM380 power op-amps to design a bridge amplifier as shown in Fig. 11.43. The output power is \( P_L = 20 \text{ W} \) at \( R_L = 4 \Omega \). Assume \( V_{CC} = 15 \text{ V} \) and \( V_{1} = 100 \text{ mV} \) (peak).

11.29 The gain and output stages of an LM380 power op-amp are shown in Fig. P11.29. Determine the small-signal differential voltage gain \( A_{id} = V_{o} / V_{t} \), the differential input resistance \( R_{id} \), and the output resistance \( R_o \). The diode parameters are \( I_S = 10^{-13} \text{ A} \) and \( I_{D(min)} = 1 \text{ mA} \) to ensure conduction. The transistor parameters are \( \beta_f = h_{fe} = 50 \), \( V_{BE} = 0.7 \text{ V} \), \( V_A = 40 \text{ V} \), and \( V_{CE(sat)} = 0.2 \text{ V} \). Assume a DC supply voltage of \( V_{CC} = 15 \text{ V} \) and \( R_L = 10 \text{ k}\Omega \).
11.30 The power op-amp shown in Fig. 11.40 uses DC supply voltage $V_{CC} = -V_{EE} = 15$ V and transistors of type Q2N2222 having $I_s = 14.34 \times 10^{-15}$ and $\beta_F = 255.9$.

a. Determine the output voltage $v_o$ and the input resistance $R_i$ for an input voltage of $v_{in} = 5$ V.

b. Use PSpice to verify the results in part (a) and determine the high cutoff frequency $f_H$ by plotting the frequency response.

11.31 Design the bridge amplifier shown in Fig. 11.43 to obtain a voltage gain $A_f = 250$ V/V. Assume $V_{CC} = -V_{EE} = -12$ V. Use PSpice to verify the results.

11.12 Thermal Considerations

11.32 The power dissipation of a transistor is specified as $P_{D(max)} = 250$ W at $T_{CO} = 25^\circ$C. The transistor is mounted on a heat sink. The thermal resistances are $\theta_{JC} = 0.7^\circ$C W, $\theta_{CS} = 0.2^\circ$C W, and $\theta_{SA} = 0.8^\circ$C W. If $T_{J(max)} = 200^\circ$C and $T_A = 45^\circ$C, calculate the maximum permissible power dissipation of the transistor.

11.33 The power dissipation of a transistor is specified as $P_{D(max)} = 290$ W at $T_{CO} = 25^\circ$C. The transistor is mounted on a heat sink. The thermal resistances of the transistor are $\theta_{JC} = 0.6^\circ$C W and $\theta_{CS} = 0.2^\circ$C W. If $T_{J(max)} = 200^\circ$C, $T_C = 150^\circ$C, and $T_A = 45^\circ$C, calculate the required thermal resistance of the heat sink $\theta_{SA}$. 
CHAPTER 12

ACTIVE FILTERS

Learning Outcomes
After completing this chapter, students should be able to do the following:

• List the advantages of active filters over passive filters.
• Describe the characteristics and types of active filters.
• Determine the pole locations for the Butterworth function.
• Analyze active filters.
• Select an active filter to meet desired frequency requirements.
• Design an active filter to meet desired specifications.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_i, V_o$</td>
<td>Input and output voltages of a filter</td>
</tr>
<tr>
<td>BW</td>
<td>Bandwidth of a filter</td>
</tr>
<tr>
<td>$f_{clk}$</td>
<td>Clock frequency, in hertz</td>
</tr>
<tr>
<td>$p_1, \ldots, p_n, z_1, \ldots, z_n$</td>
<td>Poles and zeros of a transfer function</td>
</tr>
<tr>
<td>$H(s), D(s), N(s)$</td>
<td>Transfer function, and denominator and numerator of a transfer function</td>
</tr>
<tr>
<td>$H_n$</td>
<td>Transfer function of an $n$th-order filter</td>
</tr>
<tr>
<td>$Q, K$</td>
<td>Quality factor and constant parameter of a filter</td>
</tr>
</tbody>
</table>
12.1 Introduction

In electrical engineering, a filter is a frequency-selective circuit that passes a specified band of frequencies and blocks or attenuates signals of frequencies outside this band. These signals are usually voltages. Filters that employ only passive elements such as capacitors, inductors, and resistors are called passive filters. Filters that make use of the properties of op-amps in addition to resistors and capacitors are called active filters or more often analog filters, in contrast to digital filters. Both analog and digital filters can be implemented in the same IC chip. This chapter introduces active filters and deals with the analysis and design of simple circuit topologies. Because of their practical importance, analog filters are often covered in a single course [1, 2].

12.2 Active versus Passive Filters

Both active and passive filters are used in electronic circuits. However, active filters offer the following advantages over passive filters:

- **Flexibility of gain and frequency adjustment:** Since op-amps can provide a voltage gain, the input signal in active filters is not attenuated as it is in passive filters. It is easy to adjust or tune active filters.
- **No loading effect:** Because of the high input resistance and low output resistance of op-amps, active filters do not cause loading of the input source or the load.
- **Cost and size:** Active filters are less expensive than passive filters because of the availability of low-cost op-amps and the absence of inductors.
- **Parasitics:** Parasitics are reduced in active filters because of their smaller size.
- **Digital integration:** Analog filters and digital circuitry can be implemented on the same IC chip.
- **Filtering functions:** Active filters can realize a wider range of filtering functions than passive filters.
- **Gain:** An active filter can provide gain, whereas a passive filter often exhibits a significant loss.

Active filters also have some disadvantages:

- **Bandwidth:** Active components have a finite bandwidth, which limits the applications of active filters to the audio-frequency range. Passive filters do not have such an upper-frequency limitation and can be used up to approximately 500 MHz.
- **Drifts:** Active filters are sensitive to component drifts due to manufacturing tolerances or environmental changes; in contrast, passive filters are less affected by such factors.
- **Power supplies:** Active filters require power supplies, whereas passive filters do not.
- **Distortion:** Active filters can handle only a limited range of signal magnitudes; beyond this range, they introduce unacceptable distortion.
- **Noise:** Active filters use resistors and active elements, which produce electrical noise.

### Symbol Meaning

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_o$, $f_c$, $f_L$, $f_H$</td>
<td>Cutoff, center, low cutoff, and high cutoff frequencies of a filter</td>
</tr>
<tr>
<td>$R_n$, $C_n$</td>
<td>Normalized resistance and capacitance of a filter</td>
</tr>
<tr>
<td>$\theta_p$, $\theta_z$</td>
<td>Pole and zero angles</td>
</tr>
</tbody>
</table>
In general, the advantages of active filters outweigh their disadvantages in voice and data communication applications. Active filters are used in almost all sophisticated electronic systems for communication and signal processing, such as television, telephone, radar, space satellite, and biomedical equipment. However, passive filters are still widely used.

## 12.3 Types of Active Filters

Let $V_i \angle 0$ be the input voltage to the filtering circuit shown in Fig. 12.1. The output voltage $V_o$ and its phase shift $\theta$ will depend on the frequency $\omega$. If two voltages are converted to Laplace’s domain of $s$, the ratio of the output voltage $V_o(s)$ to the input voltage $V_i(s)$ is referred to as the **voltage transfer function** $H(s)$:

$$H(s) = \frac{V_o(s)}{V_i(s)}$$

---

**FIGURE 12.1** Filtering circuit
The transfer function \( H(s) \) can be expressed in the general form

\[
H(s) = \frac{a_m s^m + \cdots + a_2 s^2 + a_1 s + a_0}{s^n + \cdots + b_2 s^2 + b_1 s + b_0} \quad (\text{for } n \geq m)
\]

(12.1)

whose coefficients are determined so as to meet the desired filter specifications. Substituting \( s = j \omega \) will give \( H(j \omega) \), which will have a magnitude and a phase delay. The denominator and the numerator of Eq. (12.1) are polynomials in \( s \) with real coefficients. If these polynomials are factored, \( H(s) \) can be written as [3, 4]

\[
H(s) = \frac{N(s)}{D(s)} = \frac{a_m(s + z_1)(s + z_2) \cdots (s + z_m)}{(s + p_1)(s + p_2) \cdots (s + p_n)}
\]

(12.2)

The expression \( z_1, z_2, \ldots, z_m \) is referred to as the zeros of \( H(s) \) because \( H(s) = 0 \) when \( s = -z_m \) and the expression \( p_1, p_2, \ldots, p_n \) is referred to as the poles of \( H(s) \) because \( H(s) = \infty \) when \( s = -p_n \). The poles and zeros can be plotted in the complex \( s \)-plane as shown in Fig. 12.1(b), where \( s = \sigma + j \omega \) is considered as one set. Since the coefficients of \( H(s) \) are all real, the poles and zeros occur in conjugate pairs.

For stability all poles must lie in the left half-plane. When the poles lie on the \( j \omega \)-axis and are simple, the network oscillates; when the poles lie in the right half-plane, the responses grow exponentially with time. Thus, for stability reasons, the characteristic polynomial \( D(s) \) of a realizable system must have only left half-plane poles. Substituting \( s = j \omega \) will give \( H(j \omega) \), which will have a magnitude and a phase delay as given by

\[
H(j \omega) = \frac{a_m(z_1 + j \omega)(z_2 + j \omega) \cdots (z_m + j \omega)}{(p_1 + j \omega)(p_2 + j \omega) \cdots (p_n + j \omega)} = \frac{a_m}{(p_1 + \omega^2)(p_2 + \omega^2) \cdots (p_n + \omega^2)} \angle \theta_H
\]

(12.3)

where \( \theta_H \) is the phase angle of the transfer function as given by

\[
\theta_H = \left( \tan^{-1} \frac{\omega}{z_1} + \tan^{-1} \frac{\omega}{z_2} + \cdots + \tan^{-1} \frac{\omega}{z_m} \right) - \left( \tan^{-1} \frac{\omega}{p_1} + \tan^{-1} \frac{\omega}{p_2} + \cdots + \tan^{-1} \frac{\omega}{p_n} \right)
\]

(12.4)

Therefore, the transfer function will cause a delay or a lead in the output response. To minimize this delay or lead, the zeros should be the mirror image of the poles as shown in Fig. 12.1(c).

Depending on the desired specification of magnitude or phase delay, active filters can be classified as low-pass filters, high-pass filters, band-pass filters, band-reject filters, or all-pass filters. The ideal characteristics of these filters are shown in Fig. 12.2. A low-pass (LP) filter passes frequencies from DC to a desired frequency \( f_0 (= \omega_0 / 2 \pi) \) and attenuates high frequencies. \( f_0 \) is known as the cutoff frequency. The low-frequency range from 0 to \( f_0 \) is known as the passband or bandwidth (BW), and the high-frequency range from \( f_0 \) to \( \infty \) is known as the stop band. A high-pass (HP) filter is the complement of the low-pass filter; the frequency range from 0 to \( f_0 \) is the stop band, and the range from \( f_0 \) to \( \infty \) is the passband.

![Figure 12.2](image-url)

**FIGURE 12.2** Ideal filter characteristics
A band-pass (BP) filter passes frequencies from $f_1$ to $f_2$ and stops all other frequencies. A band-reject (BR) filter is the complement of a BP filter; the frequencies from $f_1$ to $f_2$ are stopped, and all other frequencies are passed. BR filters are sometimes known as band-stop filters. An all-pass (AP) filter passes all frequencies from 0 to $\infty$, but it provides a phase delay.

It is impossible to create filters with the ideal characteristics shown in Fig. 12.2. Instead of abrupt changes from pass to stop behavior and from stop to pass behavior, practical filters exhibit a gradual transition from stop band to passband. Realistic filter characteristics are shown in Fig. 12.3 [(a)–(d)]. All characteristics are combined in Fig. 12.3(e). The cutoff frequency corresponds to the frequency at which the gain is 70.7% of its maximum value. The sharpness of the transition or the rate at which the characteristic changes is known as the roll-off or the falloff rate. If the frequency is plotted on a logarithmic scale, the plot is known as a Bode plot, and the roll-off, or asymptotic slope, is measured in multiples of ±6 dB per octave or ±20 dB per decade.

**FIGURE 12.3** Realistic filter characteristics

- **(a) Low-pass filter**
- **(b) High-pass filter**
- **(c) Band-pass filter**
- **(d) Band-reject filter**
- **(e) Combined characteristics**

**KEY POINTS OF SECTION 12.3**

- Depending on the frequency characteristic, filters can be classified as low-pass, high-pass, band-pass, band-reject, or all-pass.
- It is not possible to create filters with the ideal characteristics of abrupt changes from pass to stop behavior and from stop to pass behavior. Practical filters exhibit a gradual transition from stop band to passband.
12.4 First-Order Filters

A first-order filter is the simplest type. It is commonly used and serves as the building block for a wide variety of active filters. Both the numerator and the denominator of Eq. (12.1) are $1$, $m = n = 1$. Thus, the first-order filter has the transfer function as given by [5, 6]

$$H(s) = \frac{a_1 s + a_0}{s + b_0} = K \frac{s + z_1}{s + p_1}$$

(12.5)

where $p_1$ and $z_1$ are the pole and zero of the transfer function and $K$ is the voltage gain between the output and the input. For $s = -z_1$, $H(s = -z_1) = 0$, and for $s = -p_1$, $H(s = -p_1) = \infty$. The pole–zero location for the transfer function is shown in Fig. 12.4. For $p_1 > z_1$, the zero is always closer to the origin than the pole. When $s$ becomes very large, then $H$ approaches the value of $K$. For $s = 0$, $H$ becomes less than $K$. That is,

$$H(s = \infty) = 1 \quad \text{and} \quad H(s = 0) = \frac{Kz_1}{p_1}$$

For $z_1 > p_1$, the pole is always closer to the origin than the zero, and $H$ becomes greater than $K$. Therefore, the pole–zero locations are important parameters to characterize the transfer function and its response. The output will be either $K$ or less than $K$; that is, the output is attenuated.

By letting $s = j\omega$, Eq. (12.2) gives the frequency characteristics as

$$H(j\omega) = \frac{z_1 + j\omega}{p_1 + j\omega} = \frac{Kz_1}{p_1} \times \frac{1 + (j\omega/z_1)}{1 + (j\omega/p_1)}$$

(12.6)

which gives the magnitude as

$$|H(j\omega)| = \frac{Kz_1}{p_1} \sqrt{\frac{1 - (\omega/z_1)^2}{1 - (\omega/p_1)^2}}$$

(12.7)

The phase angle $\theta$ is given by

$$\theta = \theta_z - \theta_p = \tan^{-1}\left(\frac{\omega}{z_1}\right) - \tan^{-1}\left(\frac{\omega}{p_1}\right)$$

(12.8)

![FIGURE 12.4 Pole and zero locations of first-order filters](image-url)
The phase angle $\theta$ can be leading, having a positive value, or lagging, having a negative value. The phase angle for zero becomes $\theta_z = \tan^{-1}(\omega/z_1) = 45^\circ$ at $\omega = z_1$, as shown in Fig. 12.5(a), and the phase angle for pole is $\theta_p = \tan^{-1}(\omega/z_1) = 45^\circ$ at $\omega = z_p$, as shown in Fig. 12.5(b). Depending on the values of $z_1$ and $p_1$, Eq. (12.3) can exhibit the characteristics of a lead low-pass filter, a lag low-pass filter, a lead HP filter, and a lag HP filter. The conditions for $z_1$ and $p_1$ to obtain these filters are shown in Table 12.1. A pole 1 in the denominator gives the low-pass characteristic, whereas the addition of a zero at the origin gives the HP characteristic. If $p_1 > z_1$, it makes an HP characteristic, whereas $p_1 < z_1$ makes a low-pass

**TABLE 12.1** Poles and zeros for high- and low-pass filters

<table>
<thead>
<tr>
<th>Pole and Zero</th>
<th>$T(s)$</th>
<th>Classification</th>
</tr>
</thead>
<tbody>
<tr>
<td>$s + z_1$</td>
<td>$s + p_1$ ($p_1 &gt; z_1$)</td>
<td>Lag high-pass</td>
</tr>
<tr>
<td>$s$</td>
<td>$s + p_1$</td>
<td>$z = 0$</td>
</tr>
<tr>
<td>$s + z_1$</td>
<td>$s + p_1$ ($z_1 &gt; p_1$)</td>
<td>Lead low-pass</td>
</tr>
<tr>
<td>$p_1$</td>
<td>$s + p_1$</td>
<td>$z = 0$</td>
</tr>
</tbody>
</table>

**FIGURE 12.5** Poles and zeros for $s = z_1$ and $s = z_p$
characteristic. The characteristic of a BP filter, which requires two poles, can be obtained by cascading an 
HP filter with a low-pass filter to obtain a transfer function as given by

\[ H(s) = K \frac{s + z_1}{(s + p_1)(s + p_2)} \]  

\((12.9)\)

**KEY POINTS OF SECTION 12.4**

- The locations of the poles and zeros can reveal types and characteristics of the filters.
- A first-order function can exhibit only low-pass and high-pass filters. A first-order band-pass filter 
can be made by cascading a low-pass filter with a high-pass filter.

### 12.5 The Biquadratic Function

For an active filter with \( n > 2 \), Eq. (12.1) will become complex. Thus, a second-order transfer function 
(i.e., a function with \( n = 2 \)) is most commonly used. For \( n = 2 \), Eq. (12.1) becomes

\[ H(s) = \frac{a_2s^2 + a_1s + a_0}{s^2 + b_1s + b_0} = K \frac{(s + z_1)(s + z_2)}{(s + p_1)(s + p_2)} \]  

\((12.10)\)

The poles (or zeros) can be determined from the roots of a quadratic equation from

\[ p_1, p_2 = \frac{-b_1 \pm \sqrt{b_1^2 - 4b_0}}{2} = -\frac{b_1}{2} \pm \sqrt{\left(\frac{b_1}{2}\right)^2 - b_0} \]  

\((12.11)\)

where \( b_0 \) is generally greater than \( b_1^2/4 \) and the poles are complex such that \( p_1, p_2 = \alpha_p \pm j\beta_p \), which 
can be substituted in Eq. (12.7) to give the transfer function of the form as given by

\[ H(s) = K \frac{(s + \alpha_z + j\beta_z)(s + \alpha_z - j\beta_z)}{(s + \alpha_p + j\beta_p)(s + \alpha_p - j\beta_p)} = K \frac{s^2 + 2\alpha_z s + \omega_z^2}{s^2 + 2\alpha_p s + \omega_p^2} \]  

\((12.12)\)

\[ = K \frac{s^2 + (\omega_p/Q_z)s + \omega_z^2}{s^2 + (\omega_p/Q_p)s + \omega_p^2} \]  

\((12.13)\)

where \( Q_p = \omega_p/2\alpha_p \) and \( Q_z = \omega_z/2\alpha_z \) are the pole and zero quality factors, respectively. The imaginary part \( \beta_p \) is related to \( \alpha_p \) and \( \omega_p \) by \( \beta_p = \alpha_p^2 + \beta_p^2 \). This can be solved for \( \beta_p \) as 
given by

\[ \beta_p = \sqrt{\omega_p^2 - \alpha_p^2} = \omega_p \sqrt{1 - \frac{1}{4Q_p^2}} \]  

\((12.14)\)

The pole angle \( \phi_p \) as shown in Fig. 12.6(a) can be found from

\[ \phi_p = \cos^{-1}\left(\frac{\alpha_p}{\omega_p}\right) = \cos^{-1}\left(\frac{1}{2Q_p}\right) \]  

\((12.15)\)
which shows that the phase delay depends on the quality factor $Q_p$, whose value is chosen to be greater than 1. If $Q_p > 5$, Eq. (12.14) is approximated to $\beta_p \approx \omega_p$ with an error less than 1%. As a rule of thumb, $Q_p = 7$ is generally used for second filter designs.

Equation (12.10) is known as the **biquadratic function**, which serves as the building block for a wide variety of active filters and has the general form given by

$$H(s) = K \frac{k_2 s^2 + k_1 (\omega_o/Q) s + k_0 \omega_o^2}{s^2 + (\omega_o/Q) s + \omega_o^2}$$  \((12.16)\)

where  
- $\omega_o = \text{undamped natural (or resonant) frequency}$
- $Q = \text{quality factor or figure of merit}$
- $K = \text{DC gain}$

Constants $k_2$, $k_1$, and $k_0$ are $\pm 1$ or 0. Table 12.2 shows their possible values for each type of filter. Substituting $s = j\omega$ into Eq. (12.16) will give the frequency domain $H(j\omega)$, which will have a magnitude and a phase delay:

$$H(j\omega) = \frac{-k_2 \omega^2 + jk_1 (\omega_o/Q) \omega + k_0 \omega_o^2}{-\omega^2 + j(\omega_o/Q) \omega + \omega_o^2} = \frac{(k_0 \omega_o^2 - k_2 \omega^2) + jk_1 (\omega_o/Q) \omega}{(\omega_o^2 - \omega^2) + j(\omega_o/Q) \omega}$$  \((12.17)\)

where $\omega = 2\pi f$, in radians per second, and $f$ = supply frequency, in hertz.

Normalizing the frequency $\omega$ with respect to the natural frequency $\omega_o$ so that $u = \omega/\omega_o$, Eq. (12.15) becomes

$$H(ju) = \frac{(k_0 - k_2 u^2) + (jk_1 u/\omega)}{(1 - u^2) + (ju/\omega)}$$  \((12.18)\)

where $\omega_o = \text{natural frequency, in radians per second}$.

It can be shown (Appendix B) that $Q$ is related to the bandwidth $\text{BW}$ and to $\omega_o$ by

$$Q = \frac{\omega_o}{\text{BW}} = \frac{\omega_o}{\omega_H - \omega_L}$$  \((12.19)\)

where $\omega_H = \text{high cutoff frequency, in radians per second}$, and $\omega_L = \text{low cutoff frequency, in radians per second}$. 

---

**FIGURE 12.6** Pole angles and their loci
TABLE 12.2 Biquadratic filter functions

<table>
<thead>
<tr>
<th>Filter</th>
<th>$k_2$</th>
<th>$k_1$</th>
<th>$k_0$</th>
<th>Transfer Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Low-pass</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>$H_{LP} = \frac{K\omega_0^2}{s^2 + (\omega_0/Q)s + \omega_0^2}$</td>
</tr>
<tr>
<td>High-pass</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>$H_{HP} = \frac{Ks^2}{s^2 + (\omega_0/Q)s + \omega_0^2}$</td>
</tr>
<tr>
<td>Band-pass</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>$H_{BP} = \frac{K(\omega_0/Q)s}{s^2 + (\omega_0/Q)s + \omega_0^2}$</td>
</tr>
<tr>
<td>Band-reject</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>$H_{BR} = \frac{K(s^2 + \omega_0^2)}{s^2 + (\omega_0/Q)s + \omega_0^2}$</td>
</tr>
<tr>
<td>All-pass</td>
<td>1</td>
<td>-1</td>
<td>1</td>
<td>$H_{AP} = \frac{s^2 - (\omega_0/Q)s + \omega_0^2}{s^2 + (\omega_0/Q)s + \omega_0^2}$</td>
</tr>
</tbody>
</table>

Depending on the values of the constants $k_0$, $k_1$, and $k_2$, there are four possible cases whose transfer function and phase angles in frequency domain can be described as follows:

**Case 1.** For $k_0 = 1$ and $k_1 = k_2 = 0$, there is no zero. Equation (12.18) gives the low-pass characteristic as shown in Table 12.3(a), and its transfer function in frequency domain is given by

$$H_{LP}(s) = K\frac{1}{(1 - u^2) + (ju/Q)} = \left| \frac{K}{\sqrt{(1 - u^2)^2 + (u/Q)^2}} \right| \angle \tan^{-1}\left(\frac{u/Q}{1 - u^2}\right) \quad (12.20)$$

**Case 2.** For $k_1 = 1$ and $k_0 = k_2 = 0$, there is one zero on the $j\omega$-axis. Equation (12.18) gives the BP characteristic as shown in Table 12.3(b), and its transfer function in frequency domain is given by

$$H_{BP}(s) = K\frac{ju/Q}{(1 - u^2) + (ju/Q)} = \left| \frac{(u/Q)K}{\sqrt{(1 - u^2)^2 + (u/Q)^2}} \right| \angle \left[\frac{\pi}{2} - \tan^{-1}\left(\frac{u/Q}{1 - u^2}\right)\right] \quad (12.21)$$

**Case 3.** For $k_0 = k_2 = 1$ and $k_1 = 0$, there are two zeros at the origin. Equation (12.18) gives the BR characteristic as shown in Table 12.3(c), and its transfer function in frequency domain is given by

$$H_{BR}(s) = K\frac{(1 - u^2)}{(1 - u^2) + (ju/Q)} = \left| \frac{(1 - u^2)K}{\sqrt{(1 - u^2)^2 + (u/Q)^2}} \right| \angle \tan^{-1}\left(\frac{u/Q}{1 - u^2}\right) \quad (12.22)$$

**Case 4.** For $k_2 = 1$ and $k_0 = k_1 = 0$, there are two zeros at the origin. Equation (12.18) gives the HP characteristic as shown in Table 12.3(d), and its transfer function in frequency domain is given by

$$H_{HP}(s) = K\frac{-u^2}{(1 - u^2) + (ju/Q)} = \left| \frac{u^2K}{\sqrt{(1 - u^2)^2 + (u/Q)^2}} \right| \angle \left[\pi - \tan^{-1}\left(\frac{u/Q}{1 - u^2}\right)\right] \quad (12.23)$$
### Key Points of Section 12.5

- The quality factor $Q$ is a measure of the bandwidth of a filter. The lower the value of $Q$, the more selective the filter will be.
- The denominator of all quadratic filter functions is the same, but the numerator depends on the type of filter.
12.6 Butterworth Filters

The denominator of a filter transfer function determines the poles and the falloff rate of the frequency response. Notice from Table 12.2 that the denominator of the biquadratic function has the same form for all types of filters. Butterworth filters [7] are derived from the magnitude-squared function

\[
|H_n(j\omega)|^2 = \frac{1}{(1 + \omega/\omega_0)^{2n}} \tag{12.25}
\]

which gives the magnitude of the transfer function as

\[
|H_n(j\omega)| = \frac{1}{\sqrt{1 + (\omega/\omega_0)^{2n}}} \tag{12.26}
\]

Plots of this response, known as the Butterworth response, are shown in Fig. 12.7 for \(n = 1, 2, 4, 6, 8,\) and 10. This type of response has the following properties:

1. \(|H_n(j0)| = 1\) for all \(n\) (voltage gain at zero frequency—that is, DC voltage gain at \(\omega = 0\)).
2. \(|H_n(j\omega_0)| = 1 \sqrt{2} = 0.707\) for all \(n\) (voltage gain at \(\omega = \omega_0\)).
3. \(|H_n(j\omega_0)|\) exhibits \(n\)-pole roll-off for \(\omega > \omega_0\).
4. It can be shown that all but one of the derivatives of \(|H_n(j\omega)|\) equal zero near \(\omega = 0\). That is, the maximally flat response is at \(\omega = 0\).
5. For \(n > 10\), the response becomes close to the ideal characteristic of abrupt change from passband to stop band.

By substituting \(\omega = s \ j\) into Eq. (12.25), we can express the transfer function for Butterworth filters in the \(s\) domain by

\[
|H_n(s)|^2 = \frac{1}{1 + (-1)^n(s/\omega_0)^{2n}} \tag{12.27}
\]

\[
= \frac{1}{D_n(s)D_n(-s)} \tag{12.28}
\]

where \(D_n(s)\) is a polynomial in \(s\), all of whose roots have negative real parts, and \(|D_n(s)| = |D_n(-s)|\).
12.6.1 Butterworth Function for $n = 1$

For $n = 1$ and letting $\omega_o = 1$, Eq. (12.27) becomes

$$|H_1(s)|^2 = \frac{1}{1 - s^2} = \frac{1}{D_1(s)D_1(-s)}$$  \hspace{1cm} (12.29)

Factorizing $1 - s^2$, we get $D_1(s)$ and $D_1(-s)$ as

$$D_1(s)D_1(-s) = 1 - s^2 = (1 + s)(1 - s)$$

which gives

$$D_1(s) = (1 + s) \quad \text{and} \quad D_1(-s) = (1 - s)$$

Since $|D_1(s)| = |D_1(-s)|$, Eq. (12.24) gives the Butterworth function with negative real parts. That is, for $D_1(s)$ only, we get the general form

$$H_1(s) = \frac{1}{(s/\omega_o) + 1} = \frac{\omega_o}{s + \omega_o}$$

The pole angles are $\phi_p = 0^\circ$ and $180^\circ$. For $n = 1$, it becomes a first-order function that is not generally used. For the Butterworth filter, the minimum is a second-order function, $n \geq 2$.

12.6.2 Butterworth Function for $n = 2$

If we let $\omega_o = 1$, for $n = 2$ Eq. (12.27) becomes

$$|H_2(s)|^2 = \frac{1}{1 + s^4} = \frac{1}{D_2(s)D_2(-s)}$$  \hspace{1cm} (12.30)

Factoring $1 + s^4$ gives $D_2(s)D_2(-s)$ as

$$D_2(s)D_2(-s) = s^4 + 1 = \left(s - \frac{1-j}{\sqrt{2}}\right)\left(s - \frac{1+j}{\sqrt{2}}\right)\left(s - \frac{1-j}{\sqrt{2}}\right)\left(s - \frac{1+j}{\sqrt{2}}\right)$$

which gives

$$D_2(s) = \left(s - \frac{1-j}{\sqrt{2}}\right)\left(s - \frac{1+j}{\sqrt{2}}\right) = s^2 + \sqrt{2}s + 1$$

and  \hspace{1cm} $D_2(-s) = \left(s - \frac{1-j}{\sqrt{2}}\right)\left(s - \frac{1+j}{\sqrt{2}}\right) = s^2 - \sqrt{2}s + 1$

Since $|D_2(s)| = |D_2(-s)|$, Eq. (12.30) gives the Butterworth function with negative real parts. That is, for $D_2(s)$ only, we get the general form

$$H_2(s) = \frac{1}{(s/\omega_o)^2 + \sqrt{2}(s/\omega_o) + 1} = \frac{\omega_o^2}{s^2 + \sqrt{2}\omega_o s + \omega_o^2}$$  \hspace{1cm} (12.31)

which has $Q = 1 \sqrt{2} = 0.707$. Thus, for $n = 2$, a Butterworth filter will exhibit the frequency characteristic of a second-order system (Appendix B), and the frequency response will fall at a rate of $-40$ dB decade or $-12$ dB octave.
The angles for all $k (=2n = 4)$ poles are
\[ \phi_k = \frac{180^\circ + 360^\circ \times k}{2 \times n} = 45^\circ, 135^\circ, 225^\circ, 315^\circ \]

### 12.6.3 Butterworth Function for $n = 3$

If we let $\omega_o = 1$, for $n = 3$ Eq. (12.27) becomes
\[ |H_3(s)|^2 = \left| \frac{1}{1 - s^6} \right| = \left| \frac{1}{D_3(s)D_3(-s)} \right| \]
Factoring $1 - s^6$, we get
\[ D_3(s)D_3(-s) = 1 - s^6 = (s^2 + s + 1)(s^2 - s + 1)(s + 1)(-s + 1) \]
which gives $D_3(s)$, whose roots have negative real parts, as
\[ D_3(s) = (s^2 + s + 1)(s + 1) = s^3 + 2s^2 + 2s + 1 \]
The transfer function for $n = 3$ is given by
\[ H_3(s) = \frac{1}{\left(\frac{s}{\omega_o}\right)^3 + 2\left(\frac{s}{\omega_o}\right)^2 + 2\left(\frac{s}{\omega_o}\right) + 1} \]
\[ = \frac{\omega_o^3}{s^3 + 2\omega_0s^2 + 2\omega_0^2s + \omega_0^3} \]

Thus, for $n = 3$, a Butterworth filter will exhibit the frequency characteristic of a third-order system, and the frequency response will fall at a rate of $-60$ dB decade or $-18$ dB octave.

The angles for all $k (= 2n = 6)$ poles are
\[ \phi_k = \frac{360^\circ \times k}{2 \times n} = 0^\circ, 60^\circ, 120^\circ, 180^\circ, 240^\circ, 300^\circ \]

### 12.6.4 Butterworth Function for Higher-Order Filters

A Butterworth filter will exhibit the frequency characteristic of an $n$th-order system and will fall at a rate of $20n$ dB/decade or $6n$ dB/octave. The denominator $D_n(s)$ depends on the LHS poles that will be phase shifted by $180^\circ$ with respect to the poles of $D_n(s)$ on the RHS. Each pole $p_k$ has it complex conjugate $\bar{p}_k$ as shown in Fig. 12.8 in the $s$-plane for $n = 1$ to 5. Taking the LHS real-axis as the reference, we can find the LHS pole angles for an $n$th-order filter from the following expression:
\[ \phi_k = 180^\circ - 90^\circ \left( \frac{2k + n - 1}{n} \right) \quad (\text{for } k = 1, 2, \ldots, n) \]

Assuming $\omega_o = 1$, the corresponding real and imaginary parts of the poles are given by
\[ \alpha_k = \cos \phi_k \quad \text{and} \quad \beta_k = \sin \phi_k \]
Substituting $\alpha_k$ in Eq. (12.12), we can find the function $D(s)$, which is the reciprocal of $H(s)$, from

$$D_n(s) = (1 + s) \prod_k (s^2 + 2\alpha_k s + 1)$$

(for odd values of $k = 1, 3, \ldots, n$) \hfill (12.36)

$$= \prod_k (s^2 + 2\alpha_k s + 1)$$

(for even values of $k = 2, 4, \ldots, n$)

**EXAMPLE 12.1**

**Finding the transfer function for the Butterworth response** Find the transfer function for the fifth-order Butterworth response of a BP filter.

**SOLUTION**

$n = 5$. From Eq. (12.35) we get the pole angles $\phi_k = 0^\circ, \pm 36^\circ,$ and $\pm 72^\circ$. The real values are $\alpha_k = \cos \phi_k = 1, 0.809, 0.309, 0.809,$ and $0.309$. Therefore,

$$D_5(s) = (1 + s)(s^2 + 2\alpha_1 s + 1)(s^2 + 2\alpha_2 s + 1)$$

$$= (1 + s)(s^2 + 2 \times 0.309s + 1)(s^2 + 2 \times 0.809s + 1)$$
Therefore, the transfer function for a BP filter is given by

\[
H(s) = \frac{Ks^5}{1 + (s/\omega_o)[(s/\omega_o)^2 + 2 \times 0.309s/\omega_o + 1][(s/\omega_o)^2 + 2 \times 0.809s/\omega_o + 1]}
\]

\[
= \frac{Ks^5\omega_o^5}{(1 + s)(s^2 + 2 \times 0.309s + \omega_o^2)(s^2 + 2 \times 0.809s + \omega_o^2)}
\]

The values of \(Q_k, \alpha_k, \) and \(\beta_k\) of the factorial function \(\Pi_k(s^2 + 2\alpha_k s + 1)\) for even values of \(k = 2, 4, \ldots, n\) are as follows:

<table>
<thead>
<tr>
<th>(k)</th>
<th>(\phi_k)</th>
<th>(\alpha_k)</th>
<th>(\beta_k)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>72</td>
<td>0.309</td>
<td>0.951</td>
</tr>
<tr>
<td>2</td>
<td>36</td>
<td>0.809</td>
<td>0.588</td>
</tr>
<tr>
<td>3</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>-36</td>
<td>0.809</td>
<td>-0.588</td>
</tr>
<tr>
<td>5</td>
<td>-72</td>
<td>0.309</td>
<td>-0.951</td>
</tr>
</tbody>
</table>

**KEY POINTS OF SECTION 12.6**

- Butterworth filters can give a maximally flat response.
- For \(n > 10\), the response becomes close to the ideal characteristic of abrupt change from passband to stop band. However, a filter with \(n = 2\) is quite satisfactory for most applications.

### 12.7 Transfer Function Realization

The poles and zeros of a transfer function for active filters are generally obtained from series and parallel connection impedances as shown in Fig. 12.9. The transfer function describing the relationship between the output and input voltages can be described by [8]

\[
H(s) = \frac{V_o(s)}{V_i(s)} = \frac{Z_2(s)}{Z_1(s) + Z_2(s)}
\]  

(12.37)

where the impedances \(Z_1(s)\) and \(Z_2(s)\) consist of series and parallel connections of \(Rs\) and \(Cs\). The basic elements for generating poles and zeros are listed in Table 12.4.

The transfer function is a dimensionless quantity. It is convenient to normalize all circuit elements so that they become unitless quantities. The relationships among the circuit elements remain the same, irrespective

**FIGURE 12.9** Impedances for obtaining transfer function
of their values. Normalization allows scaling of all values and has the advantages of (a) removing the
dimensions from the circuit variables, (b) the relative relationship independent of impedance level, and
(c) allowing the designer to select convenient and practical values of circuit elements. Frequency normaliza-
tion is done by dividing the frequency variable by a normalizing frequency, usually \( \omega_0 \), and the impedances
are normalized by dividing all impedances in the circuit by a normalizing resistance \( R_n \). The normalized \( s 
\) becomes \( s_n = s/\omega_0 \), and the normalized frequency becomes \( \omega_n = \omega/\omega_0 \). Thus, the normalized impedances
for resistance \( R \) and capacitance \( C \) become

\[
R_n = \frac{R}{R_o} \quad \text{and} \quad \frac{1}{s_n C_n} = \left( \frac{1}{s/\omega_0} \right) \left( \frac{1}{\omega_0 C R_o} \right)
\]

Therefore, the normalized element values are given by

\[
R_n = \frac{R}{R_o} \quad \text{and} \quad C_n = C \omega_o R_o \tag{12.38}
\]

### 12.8 Low-Pass Filters

Depending on the order of the biquadratic polynomial in Eq. 12.16, low-pass filters can be classified into
two types: first-order and second-order.

#### 12.8.1 First-Order Low-Pass Filters

The transfer function of a first-order low-pass filter has the general form

\[
H(s) = \frac{K \omega_0}{s + \omega_0} \tag{12.39}
\]
A typical frequency characteristic is shown in Fig. 12.10(a). A first-order filter that uses an $RC$ network for filtering is shown in Fig. 12.10(b). The op-amp operates as a noninverting amplifier, which has the characteristics of a very high input impedance and a very low output impedance.

The voltage ($V_x$ in Laplace’s domain of $s$) at the noninverting terminal of the op-amp can be found by the voltage divider rule:

$$V_x(s) = \frac{1/sC}{R + 1/sC} V_i(s) = \frac{1}{1 + sRC} V_i(s)$$

The output voltage of the noninverting amplifier is

$$V_o(s) = \left(1 + \frac{R_F}{R_1}\right) V_x(s) = \left(1 + \frac{R_F}{R_1}\right) \frac{1}{1 + sRC} V_i(s)$$

which gives the voltage transfer function $H(s)$ as

$$H(s) = \frac{V_o(s)}{V_i(s)} = \frac{K}{1 + sRC} \quad (12.40)$$

where the DC gain is

$$K = 1 + \frac{R_F}{R_1} \quad (12.41)$$

Substituting $s = jo\omega$ into Eq. (12.40), we get

$$H(j\omega) = \frac{V_o(j\omega)}{V_i(j\omega)} = \frac{K}{1 + j\omega RC} \quad (12.42)$$

which gives the cutoff frequency $f_o$ at 3-dB gain as

$$f_o = \frac{1}{2\pi RC} \quad (12.43)$$

The magnitude and phase angle of the filter gain can be found from

$$|H(j\omega)| = \frac{K}{[1 + (\omega/\omega_o)^2]^{1/2}} = \frac{K}{[1 + (f/f_o)^2]^{1/2}} \quad (12.44)$$

and

$$\phi = -\tan^{-1}(f/f_o) \quad (12.45)$$

where $f$ = frequency of the input signal, in hertz.
EXAMPLE 12.2

Designing a first-order low-pass filter

(a) Design a first-order low-pass filter to give a high cutoff frequency of \( f_0 = 1 \text{ kHz} \) with a pass-band gain of 4. If the desired frequency is changed to \( f_n = 1.5 \text{ kHz} \), calculate the new value of \( R_n \).

(b) Use PSpice/SPICE to plot the frequency response of the filter designed in part (a) from 10 Hz to 10 kHz.

SOLUTION

(a) The high cutoff frequency is \( f_0 = 1 \text{ kHz} \). Choose a value of \( C \) less than or equal to \( \frac{1}{2\pi f_0} \): let \( C = 0.01 \text{ \mu F} \). Using Eq. (12.43), calculate the value of \( R \):

\[
R = \frac{1}{2\pi f_0 C} = \frac{1}{2\pi \times 1 \text{ kHz} \times 0.01 \text{ \mu F}} = 15,916 \text{ \Omega} \quad \text{(use a 20-k\Omega potentiometer)}
\]

Choose values of \( R_1 \) and \( R_F \) to meet the pass-band gain \( K \). From Eq. (12.41), \( K = 1 + \frac{R_F}{R_1} \). Since \( K = 4 \),

\[
\frac{R_F}{R_1} = 4 - 1 = 3
\]

If we let \( R_1 = 10 \text{ k\Omega} \), \( R_F = 30 \text{ k\Omega} \).

Calculate the frequency scaling factor, \( \text{FSF} = \frac{f_0}{f_n} \):

\[
\text{FSF} = \frac{f_0}{f_n} = \frac{1 \text{ kHz}}{1.5 \text{ kHz}} = 0.67
\]

Calculate the new value of \( R_n = \text{FSF} \times R \):

\[
R_n = \text{FSF} \times R = 0.67 \times 15,916 = 10,664 \text{ \Omega} \quad \text{(use a 15-k\Omega potentiometer)}
\]

**NOTE:** The exact numbers of the resistances are used to demonstrate the validity of the calculated values with the PSpice simulation results. In a practical design, the commercially available standard values will be used (i.e., 16 k\Omega instead of 15,916 \Omega; see Appendix E).

(b) A low-pass filter with the calculated values of the circuit parameters and the LF411 op-amp is shown in Fig. 12.11.

The plot of the voltage gain is shown in Fig. 12.12, which gives \( K = 4.0 \) (expected value is 4) and \( f_0 \approx 998 \text{ Hz} \) (expected value is 1 kHz) at \( |H(j\omega)| = 0.707 \times 4 = 2.828 \). Thus, the results are close to the expected values. If we use a linear op-amp model in Fig. 3.8, the results will differ slightly.

**FIGURE 12.11** Low-pass filter for PSpice simulation
12.8.2 Second-Order Low-Pass Filters

The roll-off of a first-order filter is only $-20\,\text{dB/decade}$ in the stop band. A second-order filter exhibits a stop-band roll-off of $-40\,\text{dB/decade}$ and thus is preferable to a first-order filter. In addition, a second-order filter can be the building block for higher-order filters ($n = 4, 6, \ldots$). Substituting $k_2 = k_1 = 0$ and $k_0 = 1$ into Eq. (12.16), we get the general form

$$H(s) = \frac{K\omega_0^2}{s^2 + (\omega_0/Q)s + \omega_0^2} \quad (12.46)$$

where $K$ is the DC gain. A typical frequency characteristic is shown in Fig. 12.13(a); for high values of $Q$, overshoots will be exhibited at the resonant frequency $f_o$. For frequencies above $f_o$, the gain rolls off at the rate of $-40\,\text{dB/decade}$. A first-order filter can be converted to a second-order filter by adding an additional $RC$ network, known as the Sallen–Key circuit, as shown in Fig. 12.13(b). The input $RC$ network is shown in Fig. 12.13(c); the equivalent circuit appears in Fig. 12.13(d). The transfer function of the filter network is

$$H(s) = \frac{V_o(s)}{V_i(s)} = \frac{K/R_2R_3C_2C_3}{s^2 + s(R_3C_3 + R_2C_3 + R_2C_2 - K R_2C_2)/R_2R_3C_2C_3 + 1/R_2R_3C_2C_3} \quad (12.47)$$

where $K = (1 + R_F/R_1)$ is the DC gain. (See Prob. 12.16 for the derivation.)

Equation (12.47) is similar in form to Eq. (12.46). Setting the denominator equal to zero gives the characteristic equation

$$s^2 + s \left( \frac{R_3C_3 + R_2C_3 + R_2C_2 - KR_2C_2}{R_2R_3C_2C_3} \right) + \frac{1}{R_2R_3C_2C_3} = 0 \quad (12.48)$$

which will have two real parts and two equal roots. Setting $s = j\omega$ in Eq. (12.48) and then equating the real parts to zero, we get

$$-\omega^2 + \frac{1}{R_2R_3C_2C_3} = 0$$
which gives the cutoff frequency as

$$f_o = \frac{\omega_o}{2\pi} = \frac{1}{2\pi \sqrt{R_2 R_3 C_2 C_3}}$$

To simplify the design of second-order filters, equal resistances and capacitances are normally used—that is, \(R_1 = R_2 = R_3 = R\), \(C_2 = C_3 = C\). Then Eq. (12.47) can be simplified to

$$H(s) = \frac{K\omega_o^2}{s^2 + (3 - K)\omega_o s + \omega_o^2}$$

Comparing the denominator of Eq. (12.50) with that of Eq. (12.46) shows that \(Q\) can be related to \(K\) by

$$Q = \frac{1}{3 - K}$$

or

$$K = 3 - \frac{1}{Q}$$

The frequency response of a second-order system at the 3-dB point will depend on the damping factor \(\zeta\) such that \(Q = 1 - 2\zeta\). A \(Q\)-value of \(1 - \sqrt{2} (= 0.707)\), which represents a compromise between the peak magnitude and the bandwidth, causes the filter to exhibit the characteristics of a flat passband as well as a stop band, and gives a fixed DC gain of \(K = 1.586\):

$$K = 1 + \frac{R_F}{R_1} = 3 - \sqrt{2} = 1.586$$
However, more gain can be realized by adding a voltage-divider network, as shown in Fig. 12.14, so that only a fraction \( x \) of the output voltage is fed back through the capacitor \( C_2 \); that is,

\[
x = \frac{R_4}{R_4 + R_5}
\]  

which will modify the transfer function of Eq. (12.50) to

\[
H(s) = \frac{K\omega_0^2}{s^2 + (3 - xK)\omega_0s + \omega_0^2}
\]

and the quality factor \( Q \) of Eq. (12.51) to

\[
Q = \frac{1}{3 - xK}
\]

Thus, for \( Q = 0.707 \), \( xK = 1.586 \), allowing a designer to realize more DC gain \( K \) by choosing a lower value of \( x \), where \( x < 1 \).

EXAMPLE 12.3

**Designing a second-order low-pass filter**

(a) Design a second-order low-pass filter as in Fig. 12.14, to give a high cutoff frequency of \( f_H = f_0 = 1 \) kHz, a pass-band gain of \( K = 4 \), and \( Q = 0.707, 1, 2, \) and \( \infty \).

(b) Use PSpice/SPICE to plot the frequency response of the output voltage of the filter designed in part (a) from 10 Hz to 10 kHz.

**SOLUTION**

(a) To simplify the design calculations, let \( R_1 = R_2 = R_3 = R_4 = R \) and let \( C_2 = C_3 = C \). Choose a value of \( C \) less than or equal to 1 \( \mu \)F: Let \( C = 0.01 \) \( \mu \)F. For \( R_2 = R_3 = R \) and \( C_2 = C_3 = C \), Eq. (12.49) is reduced to

\[
f_o = \frac{1}{2\pi RC}
\]
which gives the value of $R$ as

$$R = \frac{1}{2\pi f_0 C} = \frac{1}{2\pi \times 1 \text{ kH} \times 0.01 \mu\text{F}} = 15,916 \Omega$$ (use a 20-kΩ potentiometer)

Then $R_F = (K - 1)R_1 = (4 - 1) \times 15,916 = 47,748 \Omega$

For $Q = 0.707$ and $K = 4$, Eq. (12.56) gives $x = 1.586$, $K = 1.586$, $4 = 0.396$. From Eq. (12.54), we get

$$\frac{R_5}{R_4} = \frac{1}{x} - 1 = \frac{1 - x}{x}$$  \hspace{1cm} (12.57)

which, for $x = 0.396$ and $R_4 = R = 15,916 \Omega$, gives

$$R_5 = 1.525 \times 15,916 = 24,275 \Omega$$ (use a 30-kΩ potentiometer)

For $Q = 1$ and $K = 4$, Eq. (12.56) gives $3 - xK = 1$ or $x = 2$. $K = 0.5$, and

$$R_5 = R = 15,916 \Omega$$

For $Q = 2$ and $K = 4$, Eq. (12.56) gives $3 - xK = 1$ or $x = 2.5$. $K = 0.625$, and

$$R_5 = 0.6R = 9550 \Omega$$

For $Q = \infty$ and $K = 4$, Eq. (12.56) gives $3 - xK = 1$ or $x = 3$. $K = 0.75$, and

$$R_5 = 0.333R = 5305 \Omega$$

NOTE: The exact numbers of the resistances are used to demonstrate the validity of the calculated values with the PSpice simulation results. In a practical design, the commercially available standard values will be used (i.e., 48 kΩ instead of 47,748 Ω and 25 kΩ instead of 24,275 Ω; see Appendix E).

(b) The low-pass filter, with the designed values of the circuit parameters and a simple DC model of the op-amp, is shown in Fig. 12.15.
12.8.3 Butterworth Low-Pass Filters

The Butterworth response requires that $|H(j0)| = 1$ (or 0 dB); the transfer function in Eq. (12.50) for the Sallen–Key circuit gives $|H(j0)| = K$ to achieve a Butterworth response with Sallen–Key topology. Therefore, we must reduce the gain by $1/K$. Consider the portion of the circuit to the left of the terminals a and b in Fig. 12.13(b). The resistance $R_2$ is in series with the input voltage $V_i$, as shown in Fig. 12.17(a). The gain reduction can be accomplished by adding a voltage-divider network consisting of $R_a$ and $R_b$, as shown in Fig. 12.17(b). The Sallen–Key circuit for the Butterworth response appears in Fig. 12.17(c). The values of $R_a$ and $R_b$ must be such that $R_{in} = R_2$ and the voltage across $R_b$ is $V_i/K$; that is,

\[
\frac{R_aR_b}{R_a + R_b} = R_2
\]

\[
\frac{R_b}{R_a + R_b} = \frac{1}{K}
\]

Solving for $R_a$ and $R_b$, we get

\[
R_a = KR_2 \quad \text{for } |H(j0)| = 1(0 \text{ dB})
\]

\[
R_b = \frac{K}{K - 1}R_2 \quad \text{for } |H(j0)| = 1(0 \text{ dB})
\]

Equations (12.60) and (12.61) will ensure a zero-frequency gain of 0 dB at all values of $Q$. For example, if $K = 4$ and $R_2 = 15,916 \, \Omega$,

\[
R_a = 4 \times 15,916 = 63,664 \, \Omega \quad \text{and} \quad R_b = \frac{4 \times 15,916}{4 - 1} = 21,221 \, \Omega
\]
It is more desirable, however, to have a gain of 0 dB at the resonant frequency $\omega_0$—that is, $|H(j\omega_0)| = 1$ (or 0 dB). Substituting $s = j\omega_0$ into Eq. (12.50) gives a gain magnitude of $K(3 - K)$, which gives the required gain reduction of $(3 - K)K$; that is,

$$\frac{R_b}{R_a + R_b} = \frac{3 - K}{K} \tag{12.62}$$

Solving Eqs. (12.58) and (12.62) for $R_a$ and $R_b$, we get

$$R_a = R_2 \frac{K}{3 - K} \quad \text{for } |H(j\omega_0)| = 1 \text{ (or 0 dB)} \tag{12.63}$$

$$R_b = R_2 \frac{K}{2K - 3} \quad \text{for } |H(j\omega_0)| = 1 \text{ (or 0 dB)} \tag{12.64}$$

Therefore, we can design an active filter to yield a gain of 0 dB at either $\omega = 0$ or $\omega = \omega_0$. In the case where $|H(j\omega_0)| = 1$ (or 0 dB) is specified, the zero-frequency gain will be reduced by a factor $(3 - K)K$; that is,

$$|H(0)| = 3 - K \quad \text{for } |H(j\omega_0)| = 1 \text{ (or 0 dB)} \tag{12.65}$$

For $Q = \sqrt{2}$ and $K = 3 - 1$ $Q = 1.586$, Eq. (12.65) gives $|H(0)| = 3 - K = 1.414$, provided that we design the filter for $|H(j\omega_0)| = 1$ (or 0 dB).

**EXAMPLE 12.4**

**Designing a second-order low-pass Butterworth filter for $|H(j\omega_0)| = 1$**

(a) Design a second-order Butterworth low-pass filter as in Fig. 12.17(c) to yield $|H(j\omega_0)| = 1$ (or 0 dB), a cutoff frequency of $f_o = 1$ kHz, and $Q = 0.707$.

(b) Use PSpice/SPICE to plot the frequency response of the output voltage of the filter designed in part (a) from 10 Hz to 10 kHz.
SOLUTION

(a) For the Butterworth response, $Q = 0.707$, and Example 12.3 gives $C = 0.01 \mu F$ and $R = 15,916 \Omega$. From Eq. (12.52),

$$K = 3 - \frac{1}{Q} = 3 - \frac{1}{0.707} = 1.586$$

and $R_F = (K - 1)R_1 = (1.586 - 1) \times 15,916 = 9327 \Omega$

From Eq. (12.63),

$$R_4 = \frac{RK}{3 - K} = \frac{15,916 \times 1.586}{3 - 1.586} = 17,852 \Omega$$

From Eq. (12.64),

$$R_b = \frac{RK}{2K - 3} = \frac{15,916 \times 1.586}{2 \times 1.586 - 3} = 146,760 \Omega$$

From Eq. (12.65),

$$|H(j0)| = 3 - K = 3 - 1.586 = 1.414$$

NOTE: The exact numbers of the resistances are used to demonstrate the validity of the calculated values with the PSpice simulation results. In a practical design, the commercially available standard values will be used (i.e., 147 kΩ instead of 146,760 Ω and 18 kΩ instead of 17,852 Ω).

(b) For PSpice simulation, the circuit in Fig. 12.15 can be modified by removing $R_4$ and $R_5$, replacing $R_2$ by $R_a$, and adding $R_b$. This modified circuit is shown in Fig. 12.18.

The PSpice plot of the voltage gain is shown in Fig. 12.19, which gives $|H(j\omega_0)| = 1.0$ at $f_0 = 1 \text{kHz}$ and $|H(j0)| = 1.414$, both of which correspond to the expected values.

NOTE: The simulation was run with the DC linear model described in Sec. 6.3.
KEY POINTS OF SECTION 12.8

- A second-order filter with a fall rate of 40 dB decade is preferred over a first-order filter with a fall rate of 20 dB decade. First- and second-order filters can be used as building blocks for higher-order filters.
- The Sallen–Key circuit is a commonly used second-order filter. This circuit can be designed to exhibit the characteristics of a flat passband as well as a stop band and can be modified to give pass-band gain as well as the Butterworth response.

12.9 High-Pass Filters

High-pass filters can be classified broadly into two types: first-order and second-order. Higher-order filters can be synthesized from these two basic types. Since the frequency scale of a low-pass filter is 0 to \( f_o \) and that of a high-pass filter is \( f_o \) to \( \infty \), their frequency scales have a reciprocal relationship. Therefore, if we can design a low-pass filter, we can convert it to a high-pass filter by applying an \( RC-CR \) transformation. This transformation can be accomplished by replacing \( R_n \) by \( C_n \) and \( C_n \) by \( R_n \). The op-amp, which is modeled as a voltage-controlled voltage source, is not affected by this transformation. The resistors that are used to set the DC gain of the op-amp circuit are not affected either.

12.9.1 First-Order High-Pass Filters

The transfer function of a first-order high-pass filter has the general form

\[
H(s) = \frac{sK}{s + \omega_o}
\]  

(12.66)
A typical high-pass frequency characteristic is shown in Fig. 12.20(a). A first-order high-pass filter can be formed by interchanging the frequency-dependent resistor and capacitor of the low-pass filter of Fig. 12.10(b). This arrangement is shown in Fig. 12.20(b). The voltage at the noninverting terminal of the op-amp can be found by the voltage divider rule. That is,

$$V_o(s) = \frac{R}{R + 1/sC} V_i(s) = \frac{S}{s + 1/RC} V_i(s)$$

The output voltage of the noninverting amplifier is

$$V_o(s) = \left(1 + \frac{R_F}{R_1}\right)V_i(s) = \left(1 + \frac{R_F}{R_1}\right)\frac{s}{s + 1/RC} V_i(s)$$

which gives the voltage gain as

$$H(s) = \frac{V_o(s)}{V_i(s)} = \frac{sK}{s + 1/RC}$$

where $K = 1 + R_F$ $R_1$ is the DC voltage gain.

Substituting $s = j\omega$ into Eq. (12.67), we get

$$H(j\omega) = \frac{V_o(j\omega)}{V_i(j\omega)} = \frac{j\omega K}{j\omega + 1/RC} = \frac{j\omega K}{j\omega + \omega_o}$$

which gives the cutoff frequency $f_o$ at 3-dB gain as

$$f_o = \frac{\omega_o}{2\pi} = \frac{1}{2\pi RC}$$

as in Eq. (12.43). The magnitude and phase angle of the filter gain can be found from

$$|H(j\omega)| = \frac{(\omega/\omega_o)K}{\left[1 + (\omega/\omega_o)^2\right]^{1/2}} = \frac{(f/f_o)K}{\left[1 + (f/f_o)^2\right]^{1/2}}$$

and

$$\phi = 90^\circ - \tan^{-1}(f/f_o)$$

This filter passes all signals with frequencies higher than $f_o$. However, the high-frequency limit is determined by the bandwidth of the op-amp itself. The gain–bandwidth product of a practical $\mu$741-type op-amp is 1 MHz.
12.9.2 Second-Order High-Pass Filters

A second-order high-pass filter has a stop-band characteristic of 40 dB/decade rise. The general form of a second-order high-pass filter is

\[ H(s) = \frac{s^2K}{s^2 + (\omega_0/Q)s + \omega_0^2} \quad (12.72) \]

where \( K \) is the high-frequency gain. Figure 12.21(a) shows a typical frequency response. As in the case of the first-order filter, a second-order high-pass filter can be formed from a second-order low-pass filter by interchanging the frequency-dominant resistors and capacitors. Figure 12.21(b) shows a second-order high-pass filter derived from the Sallen–Key circuit of Fig. 12.13(b). The transfer function can be derived by applying the \( RC \)-to-\( CR \) transformation and substituting \( 1/s \) for \( s \) in Eq. (12.47). For \( R_1 = R_2 = R_3 = R\)

![Graph of high-pass characteristic](image)

**FIGURE 12.21** Second-order high-pass filter

**EXAMPLE 12.5**

**Designing a first-order high-pass filter**  Design a first-order high-pass filter with a cutoff frequency of \( f_o = 1 \) kHz and a pass-band gain of 4.

**SOLUTION**

High-pass filters are formed simply by interchanging \( R \) and \( C \) of the input \( RC \) network, so the design and frequency scaling procedures for low-pass filters are also applicable. Since \( f_o = 1 \) kHz, we can use the values of \( R \) and \( C \) that were determined for the low-pass filter of Example 12.2—that is,

\[ C = 0.01 \mu F \]

\[ R = 15,916 \Omega \quad \text{(use a 20-k\Omega potentiometer)} \]

Similarly, we use \( R_1 = 10 \) k\Ω and \( R_F = 30 \) k\Ω to yield \( K = 4 \).

A PSpice simulation that confirms the design values can be run by interchanging the locations of \( R \) and \( C \) in Fig. 12.11 so that the statements for \( R \) and \( C \) read as follows:

\[ C \quad 1 \
\[ R \quad 0 

\[ 0.01UF \quad ; \text{For } C \text{ connected between nodes 1 and 2} \]

\[ 15916 \quad ; \text{For } R \text{ connected between nodes 2 and 0} \]
and $C_2 = C_3 = C$, the transfer function becomes

$$H(s) = \frac{s^2 K}{s^2 + (3 - K)\omega_0 s + \omega_0^2}$$  \hspace{2cm} (12.73)$$

and Eq. (12.49) gives the cutoff frequency as

$$f_0 = \frac{\omega_0}{2\pi} = \frac{1}{2\pi \sqrt{R_2 R_3 C_2 C_3}} = \frac{1}{2\pi RC}$$  \hspace{2cm} (12.74)$$

$Q$ and $K$ of the circuit remain the same. A voltage-divider network can be added, as shown in Fig. 12.22, so that only a fraction $x$ of the output voltage is fed back through resistor $R_2$. The transfer function of Eq. (12.73) then becomes

$$H(s) = \frac{s^2 K}{s^2 + (3 - xK)\omega_0 s + \omega_0^2}$$  \hspace{2cm} (12.75)$$

**EXAMPLE 12.6**

**Designing a second-order high-pass filter**

(a) Design a second-order high-pass filter as in Fig. 12.22, with a cutoff frequency of $f_0 = 1$ kHz, a pass-band gain of $K = 4$, and $Q = 0.707, 1, 2, \text{ and } \infty$.

(b) Use PSpice/SPICE to plot the frequency response of the output voltage of the filter designed in part (a) from 10 Hz to 100 kHz.

**SOLUTION**

(a) Since high-pass filters are formed simply by interchanging the Rs and Cs of the input RC network and since $f_0 = 1$ kHz, we can use the values of $R$ and $C$ that were determined for the second-order low-pass filter of Example 12.3—that is, $C = 0.01 \mu F$; and

$$R_4 = R = 15,916 \ \Omega \quad \text{(use a 20-k\Omega potentiometer)}$$

For $Q = 0.707$,

$$R_5 = 24,275 \ \Omega \quad \text{(use a 30-k\Omega potentiometer)}$$

For $Q = 1$,

$$R_5 = R = 15,916 \ \Omega$$
Active Filters

For $Q = 2$,

$$R_S = 0.6R = 9550 \ \Omega$$

For $Q = \infty$,

$$R_S = 0.3333R = 5305 \ \Omega$$

**NOTE:** The exact numbers of the resistances are used to demonstrate the validity of the calculated values with the PSpice simulation results. In a practical design, the commercially available standard values will be used (i.e., 10 k$\Omega$ instead of 9550 $\Omega$; see Appendix E).

(b) Figure 12.23 shows the circuit obtained by interchanging the locations of $R$ and $C$ in Fig. 12.15. The PSpice plots are shown in Fig. 12.24. As expected, the voltage gain shows increased peaking for a higher value of $Q$. The PSpice statements for $R$ and $C$ read as follows:

- $C_2 1 \ 2 0.01\mu F$ ; For $C_2$ connected between nodes 1 and 2
- $R_2 2 \ 8 15916$ ; For $R_2$ connected between nodes 2 and 8
- $C_3 2 \ 3 0.01\mu F$ ; For $C_3$ connected between nodes 2 and 8
- $R_3 3 \ 0 15916$ ; For $R_3$ connected between nodes 3 and 0

**NOTE:** We will notice a high-end roll-off due to the internal capacitances of the $\mu$A741 op-amp.

**FIGURE 12.23** Second-order high-pass filter for PSpice simulation

**FIGURE 12.24** PSpice plots of frequency response for Example 12.6
### 12.9.3 Butterworth High-Pass Filters

Since the frequency scale of a low-pass filter is the reciprocal of that of a high-pass filter, the Butterworth response of Eq. (12.26) can also be applied to high-pass filters. The magnitude of the transfer function becomes

$$|H_n(j\omega)| = \frac{1}{\sqrt{1 + (\omega_0/\omega)^2n}}$$  \hspace{1cm} (12.76)

where $|H_n(j\infty)| = 1$ for all $n$, rather than $|H_n(j0)| = 1$. The Butterworth response requires that $|H(j\infty)| = 1$ (or 0 dB); however, the transfer function in Eq. (12.73) gives $|H(j\infty)| = K$. Therefore, we must reduce the gain by $1/K$. The gain reduction can be accomplished by adding to Fig. 12.25(a) a voltage-divider network consisting of $C_a$ and $C_b$, as shown in Fig. 12.25(b). The complete circuit is shown in Fig. 12.25(c). The values of $C_a$ and $C_b$ must be such that $C_{\text{in}} = C_2$ and the voltage across $C_b$ is $V_i/K$; that is,

$$C_a + C_b = C_2$$ \hspace{1cm} (12.77)

$$\frac{C_a}{C_a + C_b} = \frac{1}{K}$$ \hspace{1cm} (12.78)

Solving for $C_a$ and $C_b$, we get

$$C_a = \frac{C_2}{K} \quad \text{[for } |H(j\infty)| = 1 \text{ (or 0 dB)]}$$ \hspace{1cm} (12.79)

$$C_b = C_2 \frac{K - 1}{K} \quad \text{[for } |H(j\infty)| = 1 \text{ (or 0 dB)]}$$ \hspace{1cm} (12.80)

Equations (12.79) and (12.80) will ensure a high-frequency gain of 0 dB at all values of $Q$. For $C_2 = 0.01 \mu F$ and $K = 4$, we get

$$C_a = \frac{0.01 \mu F}{4} = 2.5 \text{ nF} \quad \text{and} \quad C_b = \frac{0.01 \mu F \times (4 - 1)}{4} = 7.5 \text{ nF}$$

![Butterworth second-order high-pass filter](image)
As with low-pass filters, however, it is more desirable to have a gain of 0 dB at the resonant frequency \( \omega_o \)—that is, for \(|H(j\omega_o)| = 1 \) (or 0 dB). Substituting \( s = j\omega_o \) into Eq. (12.73) yields a gain magnitude of 
\[
K (3 - K)
\]
which gives the required gain reduction of \((3 - K) K\). Thus, Eq. (12.78) becomes

\[
\frac{C_a}{C_a + C_b} = \frac{3 - K}{K}
\]

(12.81)

Solving Eqs. (12.77) and (12.81) for \( C_a \) and \( C_b \), we get

\[
C_a = C_2 \frac{3 - K}{K} \quad \text{[for } |H(j\omega_o)| = 1 \text{ (or 0 dB)]}
\]

(12.82)

\[
C_b = C_2 \frac{2K - 3}{K} \quad \text{[for } |H(j\omega_o)| = 1 \text{ (or 0 dB)]}
\]

(12.83)

Therefore, we can design a Butterworth high-pass filter to yield a gain of 0 dB at either \( \omega = \infty \) or \( \omega = \omega_o \). However, in the case where \(|H(j\omega_o)| = 1 \) (or 0 dB) is specified, the high-frequency gain will be reduced by a factor \((3 - K) K\); that is,

\[
|H(j\infty)| = 3 - K \quad \text{[for } |H(j\omega_o)| = 1 \text{ (or 0 dB)]}
\]

(12.84)

For \( Q = \sqrt{2} \) and \( K = 3 - 1 \), \( Q = 1.586 \), Eq. (12.84) gives \(|H(j\omega_o)| = 3 - K = 1.414\), provided that we design the filter for \(|H(j\omega_o)| = 1 \) (or 0 dB).

### Example 12.7

**Designing a second-order high-pass Butterworth filter for \(|H(j\infty)| = 1\)**

(a) Design a second-order Butterworth high-pass filter as in Fig. 12.25(c) to yield \(|H(j\infty)| = 1 \) (or 0 dB), a cutoff frequency of \( f_o = 1 \text{ kHz} \), and \( Q = 0.707 \).

(b) Use PSpice/SPICE to plot the frequency response of the output voltage of the filter designed in part (a) from 10 Hz to 100 kHz.

**Solution**

(a) For \( Q = 0.707 \), Example 12.6 gives \( C = 0.01 \mu F \) and \( R = 15,916 \Omega \). From Eq. (12.52),

\[
K = \frac{3 - 1}{Q} = \frac{3 - 1}{0.707} = 1.586
\]

and

\[
R_F = (K - 1)R_1 = (1.586 - 1) \times 15,916 = 9327 \Omega
\]

From Eq. (12.79),

\[
C_a = \frac{C}{K} = \frac{0.01 \mu F}{1.586} = 6.305 \text{ nF}
\]

From Eq. (12.80),

\[
C_b = \frac{C(K - 1)}{K} = \frac{0.01 \mu F \times (1.586 - 1)}{1.586} = 3.695 \text{ nF}
\]
NOTE: The exact numbers of the resistances and capacitances are used to demonstrate the validity of the calculated values with the PSpice simulation results. In a practical design, the commercially available standard values will be used (i.e., $9.3 \, \Omega$ instead of $9327 \, \Omega$ and $C_b = 4 \, \text{nF}$ instead of $3.695 \, \text{nF}$; see Appendix E).

(b) For PSpice simulation, the circuit of Fig. 12.18 can be transformed into the circuit of Fig. 12.26 by removing $R_4$ and $R_5$, interchanging the locations of $R$ and $C$, replacing $C_2$ by $C_a$, and adding $C_b$ between nodes a and b. The PSpice statements for $R$ and $C$ read as follows:

- CA 1 2 6.305nF ; For CA connected between nodes 1 and 2
- CB 2 0 3.695nF ; For CB connected between nodes 2 and 0
- R2 2 6 15916 ; For R2 connected between nodes 2 and 6
- C3 2 3 0.01UF ; For C3 connected between nodes 2 and 3
- R3 3 0 15916 ; For R3 connected between nodes 3 and 0

The PSpice plot of the gain is shown in Fig. 12.27, which gives $|H(j\omega)| = 1.0$ at $\omega = \infty$. 

FIGURE 12.26 Second-order high-pass Butterworth filter for PSpice simulation

FIGURE 12.27 PSpice plot of frequency response for Example 12.7
12.10 Band-Pass Filters

A band-pass filter has a passband between two cutoff frequencies $f_L$ and $f_H$ such that $f_H > f_L$. Any frequency outside this range is attenuated. The transfer function of a BP filter has the general form

$$H_{BP}(s) = \frac{K_{PB}(\omega_C/Q)s}{s^2 + (\omega_C/Q)s + \omega_C^2}$$  \hspace{1cm} (12.85)$$

where $K_{PB}$ is the pass-band gain and $\omega_C$ is the center frequency in radians per second. There are two types of band-pass filters: wide band pass and narrow band pass. Although there is no dividing line between the two, it is possible to identify them from the value of the quality factor $Q$. A filter may be classified as wide band pass if $Q \leq 10$ and narrow band pass if $Q > 10$. The higher the value of $Q$, the more selective the filter or the narrower its bandwidth (BW) will be. Thus, $Q$ is a measure of the selectivity of a filter. The relationship of $Q$ to 3-dB bandwidth and center frequency $f_C$ is given by

$$Q = \frac{\omega_C}{BW} = \frac{f_C}{f_H - f_L}$$  \hspace{1cm} (12.86)$$

For a wide-band-pass filter, the center frequency $f_C$ can be defined as

$$f_C = \sqrt{f_Lf_H}$$  \hspace{1cm} (12.87)$$

where $f_L$ is the low cutoff frequency, in hertz, and $f_H$ is the high cutoff frequency, in hertz. In a narrow-band-pass filter, the output peaks at the center frequency $f_C$.

12.10.1 Wide-Band-Pass Filters

The frequency characteristic of a wide-band-pass filter is shown in Fig. 12.28(a), where $f_H > f_L$. This characteristic can be obtained by implementing Eq. (12.85), which may not give a flat midband gain over a wide bandwidth. An alternative arrangement is to use two filters: one low-pass filter and one high-pass filter. The output is obtained by multiplying the low-frequency response by the high-frequency response, as shown in Fig. 12.28(b); this solution can be implemented simply by cascading the first-order (or second-order)
high-pass and low-pass sections. The order of the band-pass filter depends on the order of the high-pass and low-pass sections. This arrangement has the advantage that the falloff, rise, and midband gain can be set independently. However, it requires more op-amps and components.

Figure 12.28(c) shows a ±20 dB decade wide-band-pass filter implemented with first-order high-pass and first-order low-pass filters. In this case, the magnitude of the voltage gain is equal to the product of the voltage gain magnitudes of the high-pass and low-pass filters. From Eqs. (12.40) and (12.67), the transfer function of the wide-midband filter for first-order implementation becomes

\[
H(s) = \frac{K_{PB} \omega_H s}{(s + \omega_L)(s + \omega_H)}
\]  

(12.88)

Using Eqs. (12.46) and (12.72) gives the transfer function for second-order implementation:

\[
H(s) = \frac{K_{PB} \omega_H^2 s^2}{[s^2 + (\omega_L/Q)s + \omega_L^2][s^2 + (\omega_H/Q)s + \omega_H^2]}
\]  

(12.89)

where \(K_{PB} = \) overall pass-band gain = high-pass gain \(K_H \times \) low-pass gain \(K_L\).
EXAMPLE 12.8

Designing a wide-band-pass filter

(a) Design a wide-band-pass filter with \( f_L = 10 \text{ kHz} \), \( f_H = 1 \text{ MHz} \), and a pass-band gain of \( K_{PB} = 16 \).

(b) Calculate the value of \( Q \) for the filter.

(c) Use PSpice/SPICE to plot the frequency response of the filter designed in part (a) from 100 Hz to 10 MHz.

SOLUTION

(a) Let the gain of the high-pass section be \( K_H = 4 \). For the first-order high-pass section, \( f_L = 10 \text{ kHz} \). Following the steps in Example 12.5, we let \( C = 1 \text{ nF} \). Then

\[
R = \frac{1}{2\pi \times 10 \text{ kHz} \times 1 \text{ nF}} = 15.915 \text{ k}\Omega
\]

and

\[
K_H = 1 + \frac{R_F}{R_1} = 4 \quad \text{or} \quad \frac{R_F}{R_1} = 4 - 1 = 3
\]

If we let \( R_1 = 10 \text{ k}\Omega \), \( R_F = 3R_1 = 30 \text{ k}\Omega \).

For the first-order low-pass section, \( f_H = 1 \text{ MHz} \) and the desired gain is \( K_L = K_{PB} \quad K_H = 16 \quad 4 = 4 \). Following the steps in Example 12.2, we let \( C' = 10 \text{ pF} \). Then

\[
R' = \frac{1}{2\pi \times 1 \text{ MHz} \times 10 \text{ pF}} = 15.915 \text{ k}\Omega
\]

and

\[
K_L = 1 + \frac{R'_F}{R'_1} = 4 \quad \text{or} \quad \frac{R'_F}{R'_1} = 4 - 1 = 3
\]

If we let \( R'_1 = 10 \text{ k}\Omega \), \( R'_F = 3R'_1 = 30 \text{ k}\Omega \).

(b) From Eq. (12.87),

\[
f_C = \sqrt{10 \text{ kHz} \times 1 \text{ MHz}} = 100 \text{ kHz}
\]

and

\[
\text{BW} = 1 \text{ MHz} - 10 \text{ kHz} = 990 \text{ kHz}
\]

From Eq. (12.86), we can find

\[
Q = \frac{100\text{kHz}}{1\text{MHz} - 10\text{kHz}} = 0.101
\]

NOTE: The exact numbers of the resistances are used to demonstrate the validity of the calculated values with the PSpice simulation results. In a practical design, the commercially available standard values will be used (i.e., 16 k\Omega instead of 15.915 k\Omega; see Appendix E).

(c) The wide-band-pass filter with the calculated values is shown in Fig. 12.29.

The frequency response (with a linear model in Fig. 3.8) is shown in Fig. 12.30, which gives \( K_{PB} = 15.842 \) (expected value is 16), \( f_L = 10.04 \text{ kHz} \) (expected value is 10 kHz), and \( f_H = 997 \text{ kHz} \) (expected value is 1 MHz). For a low value of bandwidth, the response due to the high-pass filter may not reach the expected gain before the low-pass filter becomes effective. As a result, the pass-band gain may be much lower than 16.
12.10.2 Narrow-Band-Pass Filters

A typical frequency response of a narrow-band-pass filter is shown in Fig. 12.31(a). This characteristic can be derived by setting a high $Q$-value for the band-pass filter shown in Fig. 12.31(b). This filter uses only one op-amp in the inverting mode. Because it has two feedback paths, it is also known as a multiple feedback filter. For a low $Q$-value, it can also exhibit the characteristic of a wide-band-pass filter.

A narrow-band-pass filter is generally designed for specific values of $f_C$ and $Q$ or $f_C$ and BW. The op-amp, along with $C_2$ and $R_2$, can be regarded as an inverting differentiator such that $V_o(s) = (-sC_2R_2)V_x(s)$; the equivalent filter circuit is shown in Fig. 12.31(c). The transfer function of the filter network is

$$H_{BP}(s) = \frac{V_o(s)}{V_i(s)} = \frac{(-1/R_1C_1)s}{s^2 + (1/R_2)(1/C_1 + 1/C_2)s + 1/R_1R_2C_1C_2} \quad (12.90)$$

NOTE: If we use the linear model, the high-end roll-off will be sooner due to the internal capacitances of the $\mu$A741 op-amp.
which is similar in form to Eq. (12.85). (See Prob. 12.29 for the derivation.) For $C_1 = C_2 = C$, Eq. (12.90) gives

$$
\omega_C = \frac{1}{\sqrt{R_1 R_2 C_1 C_2}} = \frac{1}{C \sqrt{R_1 R_2}} \quad (12.91)
$$

$$
Q = \frac{1}{2} \sqrt{\frac{R_2}{R_1}} \quad (12.92)
$$

$$
K_{PB} \left( \frac{\omega_C}{Q} \right) = \frac{1}{R_1 C_1} \quad (12.93)
$$

Solving these equations, we can find the component values:

$$
R_1 = \frac{Q}{2 \pi f_C C K_{PB}} \quad (12.94)
$$

$$
R_2 = \frac{Q}{\pi f_C C} \quad (12.95)
$$

$$
K_{PB} = \frac{R_2}{2 R_1} = 2 Q^2 \quad (12.96)
$$

Resistance $R_1$ can be replaced by $R_A$, and resistance $R_B$ can be connected between nodes a and 0 so that the design specification $|H_{BP}(j\omega_C)| = 1$ (or 0 dB) is met for the Butterworth response. The method of calculating the values of $R_A$ and $R_B$ for a gain reduction of $1 \text{ K}_{PB} (=1 \text{ } 2Q^2)$ is explained in Sec. 12.8.

Notice from Eq. (12.96) that, for a known value of $Q$, the value of $K_{PB}$ is fixed. It is, however, possible to have different values of $K_{PB}$ and $Q$ by choosing only the value of $R_B$ without changing the value of $R_1$. The new value of the gain $K_{PB}$ is related to $2Q^2$ by

$$
\frac{K_{PB}}{2Q^2} = \frac{R_B}{R_1 + R_B}
$$

which gives the value of $R_B$ as

$$
R_B = \frac{R_1 K_{PB}}{2Q^2 - K_{PB}} = \frac{Q}{2 \pi f_C C(2Q^2 - K_{PB})} \quad (12.97)
$$
provided that

\[ K_{PB} < 2Q^2 \]  \hspace{1cm} (12.98)

Also, the center frequency \( f_C \) can be changed to a new value \( f'_C \) without changing the pass-band gain (or bandwidth) simply by changing \( R_B \) to \( R'_B \), so that

\[ R'_B = R_B \left( \frac{f'_C}{f_C} \right)^2 \]  \hspace{1cm} (12.99)

**EXAMPLE 12.9**

**Designing a narrow-band-pass filter**

(a) Design a narrow-band-pass filter as in Fig. 12.31(b) such that \( f_C = 1 \text{ kHz}, \ Q = 4, \) and \( K_{PB} = 8 \).

(b) Calculate the value of \( R_B \) required to change the center frequency from 1 kHz to 1.5 kHz.

(c) Use PSpice/SPICE to plot the frequency response of the narrow-band-pass filter designed in part (a) from 100 Hz to 1 MHz.

**SOLUTION**

(a) \( f_C = 1 \text{ kHz} \) and \( Q = 4 \). Let \( C_1 = C_2 = C = 0.0047 \mu \text{F} \). Check that the condition in Eq. (12.98) is satisfied; that is, \( 2Q^2 = 2 \times 4^2 = 32 \), which is greater than \( K_{PB} = 8 \). Thus, we must use \( R_B \) in Fig. 12.32. Using Eqs. (12.94), (12.95), (12.96), and (12.97), we get

\[ R_1 = \frac{Q}{2\pi f_C K_{PB}} = \frac{4}{2\pi \times 1 \text{ kHz} \times 0.0047 \mu \text{F} \times 8} = 16.93 \text{ k}\Omega \]

\[ R_2 = \frac{Q}{\pi f_C C} = \frac{4}{\pi \times 1 \text{ kHz} \times 0.0047 \mu \text{F}} = 270.9 \text{ k}\Omega \]

\[ K_{PB} = \frac{R_2}{2R_1} = \frac{270.9 \text{ k}\Omega}{2 \times 16.93 \text{ k}\Omega} = 8 \]

\[ R_B = \frac{Q}{2\pi f_C (2Q^2 - K_{PB})} = \frac{4}{2\pi \times 1 \text{ kHz} \times 0.0047 \mu \text{F} \times (2 \times 4^2 - 8)} = 5.64 \text{ k}\Omega \]

**FIGURE 12.32** Narrow-band-pass filter for PSpice simulation
From Eq. (12.99), we find that the new value of $R_B$ is

$$R_B' = R_B \left( \frac{f_C}{f} \right)^2 = 5.64 \text{k}\Omega \left( \frac{1 \text{kHz}}{1.5 \text{kHz}} \right)^2 = 2.51 \text{k}\Omega$$

NOTE: The exact numbers of the resistances are used to demonstrate the validity of the calculated values with the PSpice simulation results. In a practical design, the commercially available standard values will be used (i.e., 17 k\Omega instead of 16.93 k\Omega and 270 k\Omega instead of 270.9 k\Omega; see Appendix E).

(c) The narrow-band-pass filter with the designed values is shown in Fig. 12.32. The frequency response is shown in Fig. 12.33, which gives $f_C \approx 1 \text{kHz}$ (expected value is 1 kHz) and $K_{PB} = 8$.

KEY POINTS OF SECTION 12.10

- The wide-band-pass characteristic can be obtained by cascading a high-pass filter with a low-pass filter.
- A narrow-band-pass filter has a sharply tuned center frequency and can be implemented with only one op-amp in inverting mode of operation.

12.11 Band-Reject Filters

A band-reject filter attenuates signals in the stop band and passes those outside this band. It is also called a band-stop or band-elimination filter. The transfer function of a second-order band-reject filter has the general form

$$H_{BR}(s) = \frac{K_{PB}(s^2 + \omega_C^2)}{s^2 + (\omega_C/Q)s + \omega_C^2} \quad (12.100)$$
where $K_{PB}$ is the pass-band gain. Band-reject filters can be classified as wide band reject or narrow band reject. A narrow-band-reject filter is commonly called a *notch filter*. Because of its higher $Q (>10)$, the bandwidth of a narrow-band-reject filter is much smaller than that of a wide-band-reject filter.

### 12.11.1 Wide-Band-Reject Filters

The frequency characteristic of a wide-band-reject filter is shown in Fig. 12.34(a). This characteristic can be obtained by adding a low-pass response to a high-pass response, as shown in Fig. 12.34(b); the solution can be implemented by summing the responses of a first-order (or second-order) high-pass section and low-pass section through a summing amplifier. This arrangement is shown in Fig. 12.34(c). The order of the band-reject filter depends on the order of the high-pass and low-pass sections. For a band-reject response to be realized, the cutoff frequency $f_L$ of the high-pass filter must be larger than the cut-off frequency $f_H$ of the low-pass filter. In addition, the pass-band gains of the high-pass and low-pass sections must be equal. With an inverting summer ($A_3$), the output will be inverted. Note that $R_{OM}$ in Fig. 12.34(c) has no function but to minimize the op-amp offsets (see Chapter 14).

*FIGURE 12.34*  Wide-band-reject filter
EXAMPLE 12.10

Designing a wide-band-reject filter

(a) Design a wide-band-reject filter as shown in Fig. 12.34(c) with \( f_L = 100 \text{ kHz} \), \( f_H = 10 \text{ kHz} \), and a pass-band gain of \( K_{PB} = 4 \).

(b) Calculate the value of \( Q \) for the filter.

(c) Use PSpice/SPICE to plot the frequency response of the filter designed in part (a) from 10 Hz to 10 MHz.

SOLUTION

(a) In Example 12.8, we designed a wide-band-pass filter with \( f_L = 10 \text{ kHz} \) and \( f_H = 1 \text{ MHz} \). In this example, we have \( f_L = 100 \text{ kHz} \) and \( f_H = 10 \text{ kHz} \); that is, \( f_H > f_L \). However, we can follow the design steps in Example 12.8 to find the component values, provided that we interchange the high-pass and low-pass sections.

Thus, for the high-pass section of \( f_L = 100 \text{ kHz} \), \( C = 100 \text{ pF} \) and \( R = 15.915 \text{ k}\Omega \), and for the low-pass section of \( f_H = 10 \text{ kHz} \), \( C' = 1 \text{ nF} \) and \( R' = 15.915 \text{ k}\Omega \). For a pass-band gain of \( K_{PB} = 4 \), use \( R_1 = R'_1 = 10 \text{ k}\Omega \) and \( R_2 = R'_2 = 30 \text{ k}\Omega \). For the summing amplifier, set a gain of 1. Choose \( R_2 = R_3 = R_4 = 10 \text{ k}\Omega \).

(b) From Eq. (12.87),

\[
f_C = \sqrt{10 \text{ kHz} \times 100 \text{ kHz}} = 31.623 \text{ kHz}
\]

and \( \text{BW} = 100 \text{ kHz} - 10 \text{ kHz} = 90 \text{ kHz} \)

From Eq. (12.86), we can find

\[
Q = \frac{31.623 \text{ kHz}}{(100 \text{ kHz} - 10 \text{ kHz})} = 0.351
\]

(c) The circuit for PSpice simulation of the wide-band-reject filter is shown in Fig. 12.35.
12.11.2 Narrow-Band-Reject Filters

A typical frequency response of a narrow-band-reject filter is shown in Fig. 12.37(a). This filter, often called a notch filter, is commonly used in communication and biomedical instruments to eliminate undesired frequencies such as the 60-Hz power line frequency hum. A twin-T network, which is composed of two T-shaped networks, as shown in Fig. 12.37(b), is commonly used for a notch filter. One network is made up of two resistors and a capacitor; the other uses two capacitors and a resistor. To increase the $Q$ of a twin-T network, it is used with a voltage follower. It can be shown [9] that the transfer function of a twin-T network is given by

$$H_{NF}(s) = \frac{K_{PB}(s^2 + \omega_0^2)}{s^2 + \left(\omega_0/Q\right)s + \omega_0^2}$$

(12.101)

The frequency response is shown in Fig. 12.36, which gives $f_c \approx 31.376$ kHz (expected value is 31.623 kHz) and $K_{PB} = 4$ (expected value is 4).

NOTE: If we use the linear op-amp model, there will be a serious high-end roll-off with μA741 op-amp.

FIGURE 12.36 PSpice plot of frequency response for Example 12.10 (using linear op-amp model)

FIGURE 12.37 Narrow-band-reject filter
where
\[ \omega_n = 1 \, RC \]
\[ \omega_0 = 1 \, \sqrt{3RC} \]
\[ Q = \sqrt{3/4} \]
\[ K_{PB} = 1 \]

Therefore, the notch-out frequency, which is the frequency at which maximum attenuation occurs, is given by
\[
f_n = \frac{1}{2\pi RC} \quad (12.102)
\]

**EXAMPLE 12.11**

**Designing a narrow-band-reject (notch) filter**

(a) Design a notch filter as in Fig. 12.37(b) with \( f_n = 60 \, \text{Hz} \).

(b) Use PSpice/SPICE to plot the frequency response of the filter designed in part (a) from 1 Hz to 1 kHz.

**SOLUTION**

(a) \( f_n = 60 \, \text{Hz} \). Choose a value of \( C \) less than or equal to 1 \( \mu \text{F} \): Let \( C = 0.047 \, \mu \text{F} \). Then, from Eq. (12.102),
\[
R = \frac{1}{2\pi f_n C} = \frac{1}{2\pi \times 60 \, \text{Hz} \times 0.047 \, \mu \text{F}} = 56.44 \, \text{k}\Omega
\]
(\text{use a 59-k}\Omega \text{ standard resistor of 10\% tolerance})
\[
R_3 = \frac{R}{2} = 28.22 \, \text{k}\Omega
\]
(\text{use two 59-k}\Omega \text{ resistors in parallel})
\[
C_3 = 2C = 0.094 \, \mu \text{F}
\]
(\text{use two 0.047-\mu F capacitors in parallel})

(b) The circuit of the notch filter for PSpice simulation is shown in Fig. 12.38.
12.12 All-Pass Filters

An all-pass filter passes all frequency components of the input signals without attenuation. However, this filter provides predictable phase shifts for different frequencies of the input signals. Transmission lines (e.g., telephone wires) usually cause phase changes in the signals; all-pass filters are commonly used to compensate for these phase changes. An all-pass filter is also called a delay equalizer or a phase corrector.

Figure 12.40(a) shows the characteristic of an all-pass filter; the circuit diagram is shown in Fig. 12.40(b). The output voltage in Laplace’s domain can be obtained by using the superposition theorem:

\[
V_o(s) = -\frac{R_F}{R_1} V_i(s) + \frac{1/sC}{R + 1/sC} \left(1 + \frac{R_F}{R_1}\right) V_i(s) \tag{12.103}
\]

If we assume \( R_F = R_1 \), Eq. (12.103) can be reduced to

\[
V_o(s) = -V_i(s) + \frac{2}{1 + sRC} V_i(s)
\]
which gives the voltage gain as

\[ H(s) = \frac{V_o(s)}{V_i(s)} = \frac{1 - sRC}{1 + sRC} \]  \hfill (12.104)

Substituting \( s = j\omega \) into Eq. (12.104) gives the magnitude of the voltage gain as

\[ |H(j\omega)| = 1 \]

and the phase angle \( \phi \) as

\[ \phi = -2 \tan^{-1}(\omega RC) = -2 \tan^{-1}(2\pi f RC) \]  \hfill (12.105)

Equation (12.105) indicates that, for fixed values of \( R \) and \( C \), the phase angle \( \phi \) can change from \( 0^\circ \) to \(-180^\circ \) as the frequency \( f \) of the input signal is varied from 0 to \( \infty \). For example, if \( R = 21 \text{ k}\Omega \) and \( C = 0.1 \text{ \mu F} \), we will get \( \phi = -64.4^\circ \) at 60 Hz. If the positions of \( R \) and \( C \) are interchanged, the phase shift \( \phi \) will be positive. That is, the output signal leads the input signal.

**KEY POINT OF SECTION 12.12**

- An all-pass filter does not give any gain attenuation, but it provides predictable phase shifts for different frequencies of the input signals.

## 12.13 Switched-Capacitor Filters

Switched-capacitor filters use on-chip capacitors and MOS switches to simulate resistors. The cutoff frequencies are proportional to and determined by the external clock frequency. In addition, the cutoff or center frequency can be programmed to fall anywhere within an extremely wide range of frequencies—typically more than a 200,000:1 range. Switched-capacitor filters are becoming increasingly popular since they require no external reactive components, capacitors, or inductors. They offer the advantages of low cost, fewer external components, high accuracy, and excellent temperature stability. However, they generate more noise than standard active filters.
12.13.1 Switched-Capacitor Resistors

In all the filters discussed so far, discrete resistors and capacitors were connected to one or more op-amps to obtain the desired cutoff frequencies and voltage gain. Use of discrete resistors is avoided in integrating circuits to reduce chip size; instead, resistor behavior is simulated by using active switches. A resistor is usually simulated by a capacitor and switches. The value of this simulated resistor is inversely proportional to the rate at which the switches are opened or closed.

Consider a capacitor with two switches, as shown in Fig. 12.41. The switches are actually MOS transistors that are alternately opened and closed. When \( S_1 \) is closed and \( S_2 \) is open, the input voltage is applied to the capacitor. Therefore, the total charge on the capacitor is

\[
q = V_i C
\]  

(12.106)

When \( S_1 \) is open and \( S_2 \) is closed, the charge \( q \) flows to the ground. If the switches are ideal (i.e., they open and close instantaneously and have zero resistance when they close), the capacitor \( C \) will charge and discharge instantly. The charging current \( I_{in} \) and the discharging current \( I_{out} \) of the capacitor are shown in Fig. 12.42. If the switches are opened and closed at a faster rate, the current pulses will have the same magnitude but will occur more often. That is, the average current will be higher at a higher switching rate. The average current flowing through the capacitor of Fig. 12.41 is given by

\[
I_{av} = \frac{q}{T} = \frac{V_i C}{T}
\]  

(12.107)

\[
= V_i \frac{C}{f_{clk}}
\]

where
\[ q = \text{capacitor's charge} \]
\[ T = \text{time between closings of } S_1 \text{ or closings of } S_2, \text{ in seconds} \]
\[ f_{clk} = \frac{1}{T} = \text{clock frequency, in hertz} \]

The equivalent resistance seen by the input voltage is

\[
R = \frac{V_i}{I_{av}} = \frac{V_i}{V_i \frac{C}{f_{clk}}} = \frac{1}{f_{clk}}
\]  

(12.108)

which indicates that the value of \( R \) is a function of \( C \) and \( f_{clk} \). For a fixed value of \( C \), the value of \( R \) can be adjusted by adjusting \( f_{clk} \). Therefore, a switched-capacitor resistor, also known as a clock-tunable resistor, can be built in IC form with a capacitor and two MOS switches. Note that any change in \( V_i \) must occur at a rate much slower than \( f_{clk} \), especially when \( V_i \) is an AC signal.
12.13.2 Switched-Capacitor Integrators

A simulated resistor can be used as a part of an IC to build a switched-capacitor integrator, as shown in Fig. 12.43. The switches $S_1$ and $S_2$ must never be closed at the same time. That means the clock waveform driving the MOS switches must not overlap if the filter is to operate properly.

12.13.3 Universal Switched-Capacitor Filters

A universal filter combines many features in an op-amp and can be used to synthesize any of the normal filter types: band-pass, low-pass, high-pass, notch, and all-pass. Universal filters are available commercially (e.g., the type FLT-U2 manufactured by Datel-Intersil). A switched-capacitor filter is a type...
of universal active filter. It has the characteristics of a second-order filter and can be cascaded to provide very steep attenuation slopes. Figure 12.44 is a block diagram of the internal circuitry for National Semiconductor’s MF5. The basic filter consists of an op-amp, two positive integrators, and a summing node. An MOS switch, controlled by a logic voltage on pin 5 (SA), connects one of the inputs of the first integrator either to the ground or to the output of the second integrator, thus allowing more application flexibility. The MF5 includes a pin (9) that sets the ratio of the clock frequency \( f_{\text{clk}} \) to the center frequency \( f_C \) at either 50/100 or 100/1. The maximum recommended clock frequency is 1 MHz, which results in a maximum center frequency \( f_C \) of 20 kHz at 50/100 or 10 kHz at 100/1, provided the product \( Qf_0 \) is less than 200 kHz. An extra uncommitted op-amp is available for additional signal processing. A very convenient feature of the MF5 is that \( f_0 \) can be controlled independently of \( Q \) and the pass-band gain. Without affecting the other characteristics, one can tune \( f_0 \) simply by varying \( f_{\text{clk}} \). The selection of the external resistor values is very simple, so the design procedure is much easier than for typical \( RC \) active filters.

EXAMPLE 12.12

Designing a second-order Butterworth filter using a universal filter Using the MF5, design a second-order Butterworth low-pass filter with a cutoff frequency of 1 kHz and a pass-band gain of \(-4\). Assume a power supply of \( \pm 5 \) V and a CMOS clock.

SOLUTION

Step 1. Choose the mode in which the MF5 filter will be operated. Let us choose the simplest one: mode 1, which has low-pass, band-pass, and notch output and inverts the output signal polarity.
Step 2. Determine the values of the external resistors. The MF5 requires three external resistors that determine $Q$ and the gain of the filter. The external resistors are connected as shown in Fig. 12.45. For mode 1, the relationship among $Q$, $K_{LP}$, and the external resistors is given by the data sheet (on which only three out of six possible modes are shown) as

$$Q = \frac{f_C}{BW} = \frac{R_3}{R_2} \quad (12.109)$$

$$K_{LP} = -\frac{R_2}{R_1} \quad (12.110)$$

In this mode, the input impedance of the filter is equal to $R_1$ since the input signal is applied to INV (pin 3) through $R_1$. To provide a fairly high input impedance, let $R_1 = 10 \, \text{k} \Omega$. From Eq. (12.110), we get

$$R_2 = -K_{LP}R_1 = -(-4) \times 10 \, \text{k} \Omega = 40 \, \text{k} \Omega$$

For a second-order Butterworth low-pass filter, $Q = 0.707$. Therefore, Eq. (12.109) gives

$$R_3 = QR_2 = 0.707 \times 40 \, \text{k} \Omega = 28.28 \, \text{k} \Omega$$

Step 3. Choose the power supplies and complete their connections. Since a power supply of $\pm 5 \, \text{V}$ is required, $V_+$ (pin 6) is connected to $+5 \, \text{V}$, $V_-$ (pin 10) is connected to $-5 \, \text{V}$, and AGND (pin 11) is connected to the ground. To eliminate any ripples, two $0.1\, \mu\text{F}$ capacitors are connected across the power supplies.

Step 4. Choose the clock frequency $f_{\text{clk}}$. The 50/100 (pin 9) must be connected to $V_+$ (pin 6) for a ratio of 50:1 or to $V_-$ (pin 10) for a ratio of 100:1. Let us choose an $f_{\text{clk}}$-to-$f_0$ ratio of 50:1. That means the 50/100 (pin 9) must be connected to $V_+$ (pin 6). Since the cutoff frequency is $1 \, \text{kHz}$, the external clock frequency is $f_{\text{clk}} = 50 \times 1 \, \text{kHz} = 50 \, \text{kHz}$.

Step 5. For a CMOS clock, the L Sh (pin 7) should be connected to the ground (pin 11). The low-pass filter $S_A$ (pin 5) is connected to $V_+$ (pin 6), and $S_1$ (pin 4) is connected to the ground (pin 11). The complete circuit for the second-order low-pass filter is shown in Fig. 12.45.
12.14 Filter Design Guidelines

Designing filters requires selecting the values of \( R \) and \( C \) that will satisfy two requirements: the bandwidth and the gain. Normally, more than two resistors and capacitors are necessary, and the designer has to assume values for some of them, so there is no unique solution to the design problem. The general guidelines for designing an active filter are as follows:

**Step 1.** Decide on the design specifications, which may include the cutoff frequencies \( f_L \) and \( f_H \), pass-band gain \( K_{PB} \), bandwidth \( BW \), damping factor of \( \zeta = 0.707 \) for a flat response, \( |H(j\omega_0)| = 0.707 \), and \( |H(j0)| = 1 \).

**Step 2.** Assume a suitable value for the capacitor. The recommended values of \( C \) are 1 \( \mu \)F to 5 pF. (Mylar or tantalum capacitors are recommended because they give better performance than other types of capacitors.)

**Step 3.** Having assumed a value for the capacitor, find the value for the resistor that will satisfy the bandwidth or frequency requirement.

**Step 4.** If the value of \( R \) does not fall within the practical range of 1 k\( \Omega \) to 500 k\( \Omega \), choose a different value of \( C \).

**Step 5.** Find values for the other resistances that will satisfy the gain requirements and fall in the range of 1 k\( \Omega \) to 500 k\( \Omega \).

**Step 6.** If necessary, change the filter’s cutoff frequency. The procedure for converting the original cutoff frequency \( f_o \) to the new cutoff frequency \( f_n \) is called *frequency scaling*. It is accomplished by multiplying the value of \( R \) or \( C \) (but not both) by the ratio of the original frequency \( f_o \) to the new cutoff frequency \( f_n \). The new value of \( R \) or \( C \) can be found from

\[
R_n \text{ (or } C_n) = \frac{\text{Original cutoff frequency } f_o}{\text{New cutoff frequency } f_n} \cdot R \text{ (or } C) \quad (12.111)
\]

**KEY POINTS OF SECTION 12.14**

- Designing filters requires selecting values of \( R \) and \( C \) to meet the specifications for bandwidth and DC gain. Normally more than two resistors and capacitors are used, and values must be assumed for some of them (normally \( C \)). There is no unique solution to a design problem.
- Once a filter has been designed for one cutoff frequency, new values of \( R \) and \( C \) can be determined by multiplying the value of \( R \) or \( C \) (but not both) by the ratio of the original frequency \( f_o \) to the new cutoff frequency \( f_n \).
Summary

Active filters offer many advantages over passive filters. The many types of active filters—low-pass, high-pass, band-pass, band-reject, and all-pass—are based on the frequency characteristics. A second-order filter has a sharper stop band and is preferable to a first-order filter. An all-pass filter gives a phase shift that is proportional to the input signal frequency.

Universal filters are very popular because of their flexibility in synthesizing frequency characteristics with a very high accuracy. A switched-capacitor filter is a type of universal filter that uses on-chip capacitors and MOS switches to simulate resistors. Its cutoff frequency is proportional to and determined by the external clock frequency.

References


Review Questions

1. What is an active filter?
2. What are the advantages of active filters over passive ones?
3. What are the types of active filters?
4. What are the passband and the stop band of a filter?
5. What is a cutoff frequency?
6. What is the Butterworth response of a filter?
7. What are the differences between first-order and second-order filters?
8. What is frequency scaling of filters?
9. What is a notch filter?
10. What is a notch-out frequency?
11. What is an all-pass filter?
12. What is a universal filter?
13. What is a switched-capacitor resistor?
14. What is a switched-capacitor filter?
15. What is a clock-tunable resistor?
Problems

The symbol $\bullet$ indicates that a problem is a design problem. The symbol $P$ indicates that you can check the solution to a problem using PSpice/SPICE or Electronics Workbench. For PSpice/SPICE simulation, assume op-amps with parameters $R_i = 2 \, \text{M} \Omega$, $R_o = 75 \, \Omega$, and $A_o = 2 \times 10^5$.

12.4 First-Order Filters

12.1 Determine (a) the transfer function of the network shown in Fig. P12.1 and (b) its poles and zeros.

**FIGURE P12.1**

12.2 Determine (a) the transfer function of the network shown in Fig. P12.2 and (b) its poles and zeros.

**FIGURE P12.2**

12.3 Determine (a) the transfer function of the network shown in Fig. P12.3 and (b) its poles and zeros.

**FIGURE P12.3**

12.4 Determine (a) the transfer function of the network shown in Fig. P12.4 and (b) its poles and zeros.

**FIGURE P12.4**
12.5 The Biquadratic Function

Determine (a) the pole and zero quality factors \( Q_p \) and \( Q_z \), (b) the pole and zero resonant frequencies \( \omega_p \) and \( \omega_z \), (c) the pole factor \( \beta_p \), and (d) the pole angle \( \phi_p \). The transfer function has the general form as given by

\[
H(s) = \frac{5s^2 + 15s + 100}{s^2 + 20s + 200}
\]

12.6 Plot the frequency and phase responses of the low-pass transfer function \( H_{LP}(j\omega) \) in Eq. (12.20) versus \( \omega = 0.5 \text{ to } 5 \) for \( Q = 0.5, 0.707, \text{ and } 1 \) and \( K = 1 \).

12.7 Plot the frequency and phase responses of the high-pass transfer function \( H_{HP}(j\omega) \) in Eq. (12.21) versus \( \omega = 0.5 \text{ to } 5 \) for \( Q = 0.5, 0.707, \text{ and } 1 \) and \( K = 1 \).

12.8 Plot the frequency and phase responses of the band-pass transfer function \( H_{BP}(j\omega) \) in Eq. (12.22) versus \( \omega = 0.5 \text{ to } 5 \) for \( Q = 0.5, 0.707, \text{ and } 1 \) and \( K = 1 \).

12.9 Plot the frequency and phase responses of the band-reject transfer function \( H_{BP}(j\omega) \) in Eq. (12.23) versus \( \omega = 0.5 \text{ to } 5 \) for \( Q = 0.5, 0.707, \text{ and } 1 \) and \( K = 1 \).

12.6 Butterworth Filters

12.10 Determine the transfer function for fourth-order Butterworth response of a band-pass filter.

12.11 Determine the transfer function for sixth-order Butterworth response of a band-pass filter.

12.12 Determine the transfer function for seventh-order Butterworth response of a band-pass filter.

12.13 Determine the transfer function for eighth-order Butterworth response of a band-pass filter.

12.14 Determine the transfer function for ninth-order Butterworth response of a band-pass filter.

12.8 Low-Pass Filters

12.15 Design a first-order low-pass filter as in Fig. 12.10(b) to give a low cutoff frequency of \( f_o = 2 \text{ kHz} \) with a pass-band gain of 1. If the desired frequency is changed to \( f_n = 1.5 \text{ kHz} \), calculate the new value of \( R_n \).

12.16 Derive the transfer function \( H(s) \) of the network in Fig. 12.13(d).

12.17 Design a second-order low-pass filter as in Fig. 12.14 to give a low cutoff frequency of \( f_o = 10 \text{ kHz} \), a pass-band gain of \( K = 5 \), and \( Q = 0.707, 1 \text{ and } \infty \).

12.18 Design a second-order Butterworth low-pass filter as in Fig. 12.17(c) to yield \( |H(j\omega_o)| = 1 \text{ (or 0 dB)} \), a cutoff frequency of \( f_o = 10 \text{ kHz} \), and \( Q = 0.707 \).

12.19 Design a second-order Butterworth filter as in Fig. 12.14 to yield \( |H(0)| = 1 \text{ (or 0 dB)} \), a cutoff frequency of \( f_o = 10 \text{ kHz} \), and \( Q = 0.707 \).

12.20 Design a third-order Butterworth low-pass filter as in Fig. P12.20 to give a high cutoff frequency of \( f_o = 10 \text{ kHz} \) and a pass-band gain of 10. The transfer function has the general form

\[
H_3(s) = \frac{10\omega_o^3}{s^3 + 2\omega_o s^2 + 2\omega_o^2 s + \omega_o^3}
\]
Design a fourth-order Butterworth low-pass filter as in Fig. P12.21 to give a high cutoff frequency of \( f_o = 10 \text{ kHz} \) and a pass-band gain of 25. The transfer function has the general form

\[
H_d(s) = \frac{25\omega_0^4}{(s^2 + \sqrt{2}\omega_0s + \omega_0^2)^2}
\]

Design a first-order high-pass filter as in Fig. 12.20(b) to give a low cutoff frequency of \( f_o = 400 \text{ Hz} \) and a pass-band gain of \( K = 2 \). If the desired frequency is changed to \( f_n = 1 \text{ kHz} \), calculate the new value of \( R_n \).

Design a second-order high-pass filter as in Fig. 12.22 to give a low cutoff frequency of \( f_o = 2 \text{ kHz} \) and a pass-band gain of 2. If the desired frequency is changed to \( f_n = 3.5 \text{ kHz} \), calculate the new value of \( R_n \).

Design a second-order Butterworth high-pass filter as in Fig. 12.25(c) to yield \( |H(j\omega)| = 1 \) (or 0 dB), a cutoff frequency of \( f_o = 10 \text{ kHz} \), and \( Q = 0.707 \).

Design a second-order Butterworth high-pass filter as in Fig. 12.25(c) to yield \( |H(j\omega_0)| = 1 \) (or 0 dB), a cutoff frequency of \( f_o = 10 \text{ kHz} \), and \( Q = 0.707 \).

Design a third-order Butterworth high-pass filter as in Fig. P12.26 to give a low cutoff frequency of \( f_o = 10 \text{ kHz} \) and a pass-band gain of 10. The transfer function has the general form

\[
H_3(s) = \frac{10s^3}{s^3 + 2\omega_0s^2 + 2\omega_0^2s + \omega_0^3}
\]
12.10 Band-Pass Filters

12.27 Design a wide-band-pass filter with $f_L = 400$ Hz, $f_H = 2$ kHz, and a pass-band gain of $K_{PB} = 4$. Calculate the value of $Q$ for the filter.

12.28 Design a wide-band-pass filter with $f_L = 1$ kHz, $f_H = 10$ kHz, and a pass-band gain of $K_{PB} = 20$. Calculate the value of $Q$ for the filter.

12.29 Derive the transfer function $H(s)$ of the network in Fig. 12.31(c).

12.30 Design a band-pass filter as in Fig. P12.30 to give $f_C = 5$ kHz, $Q = 20$, and $K_{PB} = 40$.

12.31 a. Design a narrow-band-pass filter as in Fig. 12.31(b) such that $f_C = 2$ kHz, $Q = 20$, and $K_{PB} = 10$.

b. Calculate the value of $R_B$ required to change the center frequency from 2 kHz to 5.5 kHz.

12.11 Band-Reject Filters

12.32 Design a wide-band-reject filter as in Fig. 12.34(c) to give $f_H = 400$ kHz, $f_L = 2$ kHz, and $K_{PB} = 10$. Calculate the value of $Q$ for the filter.

12.33 Design a wide-band-reject filter with a falloff rate of 40 dB/decade to give $f_H = 400$ kHz, $f_L = 2$ kHz, and $K_{PB} = 40$. 

---

**FIGURE P12.26**

**FIGURE P12.30**
12.34 Design an active notch filter as in Fig. 12.37(b) with $f_n = 400$ Hz.

12.35 Derive the transfer function $H(s)$ of the network in Fig. 12.37(b).

12.36 Design an active notch filter as in Fig. P12.36 with $f_n = 400$ Hz and $Q = 5$.

**FIGURE P12.36**

12.12 *All-Pass Filters*

12.37 Design an all-pass filter as in Fig. 12.40(b) so that the phase shift is $\phi = \pm 150^\circ$ at 60 Hz.

12.38 Using the MF5, design a second-order Butterworth low-pass filter with a cutoff frequency of 2 kHz and a pass-band gain of $-2$. Assume a power supply of ±5 V and a CMOS clock.
CHAPTER 13

OSCILLATORS

Learning Outcomes
After completing this chapter, students should be able to do the following:

- Describe the operating principle of oscillators for generating sinusoidal voltage and the conditions for sustained oscillations.
- Describe the characteristics and types of oscillators.
- Select an oscillator circuit to meet specific frequency requirements.
- Design an oscillator circuit to meet desired frequency specifications.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>( A, A_f )</td>
<td>Open-loop and closed-loop voltage gains of an amplifier</td>
</tr>
<tr>
<td>( f_o, \omega_o )</td>
<td>Oscillation frequencies in hertz and radians per second</td>
</tr>
<tr>
<td>( g_m, r_{\pi}, r_o )</td>
<td>Transconductance, input resistance, and output resistance of a transistor</td>
</tr>
<tr>
<td>( h_{fe} )</td>
<td>Hybrid current gain parameter of a BJT transistor</td>
</tr>
<tr>
<td>( K_p, V_t )</td>
<td>MOSFET constant and threshold voltage</td>
</tr>
</tbody>
</table>
13.1 Introduction

We know from Sec. 10.11 that an amplifier with negative feedback will be unstable if the magnitude of the loop gain is greater than or equal to 1 and its phase shift is $\pm 180^\circ$. Under these conditions, the feedback becomes positive and the output of the amplifier oscillates. An oscillator is a circuit that generates a repetitive waveform of fixed amplitude at a fixed frequency without any external input signal. A waveform of this characteristic can be obtained by applying positive feedback in amplifiers. Positive feedback provides enough feedback signal to maintain oscillations. Although oscillations are very undesirable in linear amplifier circuits, oscillators are designed specifically to produce controlled and predictable oscillation. Thus, the strategy for designing oscillators is quite different from that for designing linear amplifiers. Occasionally, oscillators have inputs that are used to control the frequency or to synchronize the oscillations with an external reference. Oscillators are used in many electronic circuits, such as in radios, televisions, computers, and communications equipment.

13.2 Principles of Oscillators

An oscillator is an amplifier with positive feedback. The block diagram of an amplifier with positive feedback, shown in Fig. 13.1(a), suggests the following relationships:

- $v_c = v_i + v_f$
- $v_o = Av_c$
- $v_f = \beta v_o$

FIGURE 13.1 Oscillator block diagram
Using these relationships, we get the closed-loop voltage gain $A_f$:

$$A_f = \frac{v_o}{v_i} = \frac{A}{1 - A\beta} \quad (13.1)$$

### 13.2.1 Conditions for Oscillations

$A_f$ in Eq. (13.1) can be made very large by making $1 - A\beta = 0$. That is, an output of reasonable magnitude can be obtained with a very small-value input signal, tending to zero, as shown in Fig. 13.1(b). Thus, the amplifier will be unstable when $1 - A\beta = 0$, which gives the loop gain as

$$A\beta = 1 \quad (13.2)$$

Expressing Eq. (13.2) in polar form, we get

$$A\beta = 1 \angle 0^\circ \quad \text{or} \quad 1 \angle 360^\circ \quad (13.3)$$

The above discussion leads to the following design criteria for oscillators:

1. The magnitude of the loop gain $|A\beta|$ must be unity or slightly larger at the desired oscillation frequency.
2. The total phase shift $\phi$ of the loop gain must be equal to $0^\circ$ or $360^\circ$ at the same frequency.
3. The first two conditions must not be satisfied at other frequencies. This condition is normally met by carefully selecting the component values.
4. The first two conditions must continue to be satisfied as parameter values change in response to component tolerance, temperature change, aging, and device replacement. Meeting this criterion often requires special design considerations.

If an amplifier provides a phase shift of $180^\circ$, the feedback circuit must provide an additional phase shift of $180^\circ$ so that the total phase shift around the loop is $360^\circ$. The type of waveform generated by an oscillator depends on the types of components used in the circuit; hence the waveform may be sinusoidal, square, or triangular. The frequency of oscillation is determined by the feedback components:

- **RC** components generate a sinusoidal waveform at audio frequencies—that is, in the range from several hertz (Hz) to several kilohertz (kHz).
- **LC** components generate a square wave at radio frequencies—that is, in the range from 100 kHz to 100 MHz.
- Crystals generate a triangular or sawtooth wave over a wide range—that is, in the range from 10 kHz to 10 MHz.

Oscillators can be classified into many types depending on the feedback components, amplifiers, and circuit topologies used [1]. This chapter covers the following types of oscillators: phase-shift oscillators, quadrature oscillators, three-phase oscillators, Wien-bridge oscillators, Colpitts oscillators, Hartley oscillators, crystal oscillators, and active-filter tuned oscillators.
EXAMPLE 13.1

**Finding the gain and phase for oscillation** A block diagram of an oscillator is shown in Fig. 13.2. Determine the values of the gain $A$ and the phase angle $\theta$ that will produce a steady-state oscillation.

**SOLUTION**

Applying the condition of Eq. (13.3), we know that the loop gain must be

$$A \angle 180^\circ \times 0.01 \angle 0^\circ \times 0.5 \angle \theta \times 10 \angle 0^\circ = 1 \angle 360^\circ$$

from which we get

$$A \times 0.01 \times 0.5 \times 10 = 1 \quad \text{or} \quad A = \frac{1}{0.01 \times 0.5 \times 10} = 20$$

and $180 + 0 + \theta + 0 = 360$ or $\theta = 360 - 180 = 180^\circ$

![Figure 13.2 Oscillator circuit](image)

EXAMPLE 13.2

**Finding the frequency and conditions to sustain oscillation** The amplifier shown in Fig. 13.3(a) has a voltage gain of $A = 50$, input resistance $R_i = 10 \, k\Omega$, and output resistance $R_o = 200 \, \Omega$. Find the resonant frequency $\omega_0$ and the values of $R$ and $R_3$ that will sustain the oscillation.

**SOLUTION**

The voltage-controlled current source representation is shown in Fig. 13.3(b), where transconductance $G_m = A \, R_o$, $R_1$ is in parallel with $R$, and we let $R_1 = R_1 || R$. Also, $R_3$ is in series with $R_3$, and we let $R_F = R_o + R_3$. The equivalent circuit is shown in Fig. 13.4, which represents the amplifier as an ideal voltage amplifier. The feedback transfer function $\beta$ of the feedback circuit is given by

$$\beta(j\omega) = \frac{V_f}{V_i} = \frac{j\omega L (-j\omega C) || R_1}{j\omega L (-j\omega C) || R_1 || R_1 + R_F}$$
Oscillators

which can be simplified to give the loop gain as

$$A\beta(j\omega) = \frac{j\omega A}{R_F(1 - \omega^2 LC) + j\omega L(1 + R_F/R_1)}$$ (13.4)

This will provide a $0^\circ$ phase shift at the resonant frequency $\omega_0$ given by

$$\omega_0 = \frac{1}{\sqrt{LC}} \quad \text{(in rad/s)}$$ (13.5)

At this frequency, the magnitude of $A\beta(j\omega)$ becomes

$$|A\beta(j\omega)| = \frac{A}{1 + R_F/R_1}$$ (13.6)

which must be equal to 1 and gives the condition for oscillation as

$$\frac{R_F}{R_1} = A - 1$$ (13.7)

FIGURE 13.3  LC-feedback amplifier

FIGURE 13.4  Equivalent circuit for Example 13.2
13.2.2 Frequency Stability

The ability of an oscillator to oscillate at an exact frequency is called frequency stability. The oscillating frequency is a function of circuit components (e.g., LC components) and can change in response to temperature changes, device replacement, or parasitic elements. Good frequency stability can be obtained by making the phase shift a strong function of frequency at resonance. That is, \( |\frac{d\phi}{d\omega}| \) (at \( \omega = \omega_0 \)) is made large so that only a slight change in \( \phi \) is required to correct any phase shift and restore the loop gain to zero phase shift.

The quality factor (or figure of merit) \( Q \) of a circuit also determines the frequency stability. The higher the \( Q \) factor, the better the stability will be because the variation in phase shift with frequency near resonance is greater. Crystal oscillators are far more stable than RC or LC oscillators, especially at higher frequencies. The equivalent electrical circuit of a crystal has a very high \( Q \)-value, leading to a high value of \( |A/\omega| \). LC and crystal oscillators are generally used for the generation of high-frequency signals; RC oscillators are used mostly for audio-frequency applications.

13.2.3 Amplitude Stability

Like the frequency, the gain of practical amplifiers can change in response to changes in parameters such as temperature, age, and operating point. Therefore, \( |A/\omega| \) might drop below unity. If the magnitude of \( A/\omega \) falls below unity, an oscillating circuit ceases oscillating. In practice, an oscillator is designed with a value of \( |A/\omega| \) that is slightly higher than unity—say, by 5%—at the oscillating frequency. The greater the value of \( A/\omega \), the greater will be the amplitude of the output signal and the amount of its distortion. This distortion will usually lower the gain \( A \) to the value required to sustain oscillation.

For good stability, the change in the gain \( A \) with a change in the amplitude of output voltage \( v_o \) should be made large; an increase in amplitude must result in a decrease in gain. That is, \( dA dv_o \) must be a large negative number. An oscillator is often stabilized by adding nonlinear limiting devices or elements such as diodes.

KEY POINTS OF SECTION 13.2

- To sustain oscillations, the magnitude of the loop gain \( |A/\omega| \) must be unity or slightly larger at the desired oscillation frequency, and the total phase shift \( \phi \) of the loop gain must be equal to 0° or 360° at the same frequency.
- For good frequency stability, the phase shift must be made a strong function of frequency at resonance; that is, the \( Q \) factor should be high.
- For good stability, the change in the gain with a change in the amplitude of output voltage \( v_o \) should be made large; an increase in amplitude must result in a decrease in gain.
13.3 Audio-Frequency Oscillators

An audio-frequency (AF) oscillator uses RC components for generating a sinusoidal waveform in the frequency range from several hertz (Hz) to several kilohertz (kHz). There are different ways to make an amplifier oscillate. We will consider the following types of AF oscillators: phase-shift oscillator, quadrature oscillator, three-phase oscillator, Wien-bridge oscillator, and ring oscillator. The op-amp Wien bridge is the most common and easy to implement. The MOS oscillators allow CMOS implementation.

13.3.1 Phase-Shift Oscillators

A phase-shift oscillator consists of an inverting amplifier with a positive feedback circuit [1]. The amplifier gives a phase shift of 180°, and the feedback circuit gives another phase shift of 180°, so the total phase shift around the loop is 360°. A phase-shift oscillator consisting of an inverting op-amp amplifier with positive feedback is shown in Fig. 13.5(a). The feedback circuit provides voltage feedback from the output back to the input of the amplifier. Any signal that appears at the inverting terminal is shifted by 180° at the output. Therefore, an additional 180° shift is required for oscillation at a specific frequency \( f_o \) in order to give a total phase shift around the loop of 360°. Since the feedback network consists of resistors and capacitors, as shown in Fig. 13.5(b), this type of oscillator is also known as an RC oscillator. The transfer function of the feedback network in Laplace’s domain of \( s \) is given by

\[
\beta(s) = \frac{V_o(s)}{V_i(s)} = \frac{R^3C^3s^3}{R^3C^3s^3 + 6R^2C^2s^2 + 5RCs + 1} \tag{13.8}
\]

(see Prob. 13.3). The closed-loop voltage gain of the op-amp circuit is

\[
A(s) = \frac{V_o(s)}{V_i(s)} = -\frac{R_F}{R_1} \tag{13.9}
\]

![Phase-shift oscillator](a) Oscillator ![Feedback network](b) Feedback network

**FIGURE 13.5** Phase-shift oscillator
Since \( A\beta = 1 \) for an oscillator, from Eqs. (13.8) and (13.9), we get

\[
\begin{align*}
- \frac{R_F}{R_1} & \left[ \frac{R^3 C^3 s^3}{R^3 C^3 s^3 + 6R^2 C^2 s^2 + 5RC s + 1} \right] = 1 \\
\end{align*}
\]  

(13.10)

Substituting \( s = j\omega \) into Eq. (13.10) and canceling the elements in the denominator, we get

\[
- R_F (-jR^3 C^3 \omega^3) = R_1 (-jR^3 C^3 \omega^3 - 6R^2 C^2 \omega^2 + j5RC \omega + 1)
\]

Equating the real parts to zero, we get

\[
R_1 (-6R^2 C^2 \omega^2 + 1) = 0
\]

which gives the oscillation frequency \( \omega_0 \) as

\[
\omega_0 = \omega = 2\pi f_o = \frac{1}{\sqrt{6RC}} \quad \text{(in rad/s)}
\]  

(13.11)

where \( f_o \) is the frequency in hertz. Equating the imaginary parts on both sides yields

\[
- R_F (-jR^3 C^3 \omega^3) = R_1 (-jR^3 C^3 \omega^3 + j5RC \omega)
\]

which gives

\[
R_F = R_1 \left[ \frac{5}{R^2 C^2 \omega^2} - 1 \right]
\]  

(13.12)

Substituting the value of \( \omega = \omega_0 \) from Eq. (13.11) into Eq. (13.12) yields

\[
\frac{R_F}{R_1} = 29
\]  

(13.13)

which gives the condition for sustained oscillations. This relationship does not control the peak amplitude of the output voltage. The oscillation frequency \( \omega_0 \) in Eq. (13.11) is inversely proportional to the \( RC \) product, assuming that both resistances and capacitances are equal. Theoretically, the frequency can be varied by varying either \( R \) or \( C \). In practice, it is usually easier to vary \( R \) on a continuous basis and to vary \( C \) on a discrete basis. Identical capacitors are switched into the circuit at each frequency range. Also, identical resistances, which together are referred to as a gauged potentiometer, are mounted on the same shaft and are used to vary the frequency on a continuous basis in each frequency range.

Note that setting the loop gain to unity is not a reliable method for designing an oscillator. To stabilize an oscillator, usually it is necessary to limit the output voltage by introducing nonlinearity. Stability can be achieved by adding two zener diodes in series with the resistance \( R_B \), as shown in Fig. 13.6(a). As long as the magnitude of the voltage \( v_f \) across resistance \( R_1 \) is less than the zener breakdown voltage \( V_Z \), the zener diodes act as an open circuit, and the gain of the amplifier is

\[
|A_1| = \frac{R_F}{R_1}
\]  

(13.14)
As soon as the magnitude of $v_f$ starts to increase above $V_Z$, the zener diodes conduct, and the resistor $R_B$ suddenly becomes in parallel with $R_F$ so that the gain is reduced. The new gain becomes

$$|A_2| = \frac{R_F}{R_1}$$  \hspace{1cm} (13.15)

which is less than $|A_1|$. Furthermore, if the output amplitude starts to decrease, the gain $|A|$ is increased again. The $v_o$–$v_f$ characteristic of the amplifier is shown in Fig. 13.6(b).

**EXAMPLE 13.3**

**Designing a phase-shift oscillator**

(a) Design the phase-shift oscillator shown in Fig. 13.5(a) so that the oscillating frequency is $f_o = 400$ Hz.

(b) Use PSpice/SPICE to plot the transient response of the output voltage $v_o(t)$ in part (a) from 0 to 4 ms. Assume $V_{CC} = V_{EE} = 12$ V.

**SOLUTION**

(a) The following steps can be used to complete the design.

Step 1. Choose a suitable value of $C$: Let $C = 0.1 \mu F$. 
Step 2. Calculate the value of $R$ from Eq. (13.11):

$$R = \frac{1}{2\pi \sqrt{6f_0C}} = \frac{1}{2\pi \times \sqrt{6 \times 400 \text{ Hz} \times 0.1 \mu \text{F}}} = 1624 \Omega$$

Choose $R = 1.7 \text{k}\Omega$ (use a 2.7-k\Omega potentiometer).

Step 3. To prevent the loading of the op-amp by the $RC$ network, choose $R_1$ much larger than $R$ by making $R_1 \gg 10R$. Therefore, let

$$R_1 = 10R = 10 \times 1.7 \text{k}\Omega = 17 \text{k}\Omega$$

Step 4. Choose the value of $R_F$ from Eq. (13.13):

$$R_F = 29R_1 = 29 \times 17 \text{k}\Omega = 493 \text{k}\Omega$$

Choose a 500-k\Omega potentiometer $R_F$ to account for tolerance.

(b) The phase-shift oscillator with the calculated values of the circuit parameters is shown in Fig. 13.7. The PSpice plot of the output voltage $v_o = V(\text{RF}:2)$ is shown in Fig. 13.8, which gives the peak-to-peak voltage $V_{pp} = 6.78 + 6.65 = 13.43 \text{ V}$ at $f_o = 1/4.3125 \text{ m} - 1.875 \text{ m} = 410 \text{ Hz}$ (expected value is 400 Hz).

**NOTES:**

1. If you observe carefully, you will notice that the amplitude of the output voltage is falling slowly and the oscillation will not be sustained for a long time. A nonlinear device is often necessary to stabilize the oscillator.

2. An initial voltage of 1 V has been assigned to the capacitor $C$ in order to start the oscillator, and the UIC (use initial condition) is used in transient analysis. Otherwise, PSpice will first calculate the biasing values and then use those values to find the solutions, and the circuit will not oscillate. In practice, random noise or transients can cause the oscillations to begin, and they are sustained by the feedback of the appropriate signal.
13.3.2 Quadrature Oscillators

A quadrature oscillator, as shown in Fig. 13.9(a), generates two signals (sine and cosine) that are in quadrature—that is, out of phase by 90°. The actual location of sine and cosine signals is arbitrary. In Fig. 13.9(a), the output of amplifier $A_1$ is labeled as sine and that of amplifier $A_2$ as cosine. This oscillator requires a dual op-amp. Amplifier $A_2$ operates as an inverting integrator and provides a phase shift of $-270°$ (or 90°); amplifier $A_1$, in combination with the feedback network, operates as a noninverting integrator and provides the remaining $-90°$ (or 270°) to give the total phase shift of 360° that is required to satisfy the condition of oscillation. The transfer function of the feedback network shown in Fig. 13.9(b) in Laplace’s domain of $s$ is given by

$$\beta(s) = \frac{V_f(s)}{V_o(s)} = \frac{1/C_s}{R + 1/C_s} = \frac{1}{1 + RC_s} \quad (13.16)$$

If $V_{o1}$ is the voltage at the output of amplifier $A_1$, the voltage $V_x$ at its inverting terminal is given by

$$V_x = \frac{RV_{o1}}{R + 1/C_s} = \frac{RC_sV_{o1}}{1 + RC_s} \quad (13.17)$$

Since the differential voltage between the op-amp terminals is very small ($V_d \approx 0$), we can write

$$V_f = V_x - V_d \approx V_x$$
That is,
\[
\frac{V_o(s)}{1 + RCs} = \frac{RCsV_{o1}}{1 + RCs}
\]  \hspace{1cm} (13.18)
which gives the transfer function of amplifier A_1, including the feedback network, as
\[
\beta A_1(s) = \frac{V_{o1}(s)}{V_o(s)} = \frac{1}{RCs}
\]  \hspace{1cm} (13.19)
In the frequency domain, Eqs. (13.16) and (13.19) become, respectively,
\[
\beta(j\omega) = \frac{1}{1 + j\omega RC}
\]  \hspace{1cm} (13.20)
and
\[
\beta A_1(j\omega) = \frac{1}{j\omega RC} = -\frac{j}{\omega RC}
\]  \hspace{1cm} (13.21)
The transfer function of amplifier A_2 is
\[
A_2(j\omega) = \frac{V_o}{V_{o1}(j\omega)} = -\frac{1}{j\omega RC} = \frac{j}{\omega RC}
\]
which gives a phase shift of 90°. Thus, \(\beta G_1(j\omega)\) must give a phase shift of \(-90°\).
To provide a phase shift of \(-90°\), \(|\beta G_1(j\omega)|\) in Eq. (13.21) must equal unity. That is, \(\omega RC = 1\), and the frequency of oscillation is given by
\[
f_o = \frac{1}{2\pi RC} \quad \text{(in Hz)}
\]  \hspace{1cm} (13.22)
At this frequency, the magnitude of \(\beta(j\omega)\) in Eq. (13.20) becomes
\[
\beta = |\beta(j\omega)| = \left| \frac{1}{1 + j1} \right| = \frac{1}{\sqrt{2}}
\]  \hspace{1cm} (13.23)
The loop gain becomes
\[
\beta A(j\omega) = A_2(j\omega)\beta A_1(j\omega) = \frac{j}{\omega RC} \times \frac{1}{j\omega RC} = \frac{1}{\omega^2 RC} = 1
\]
Therefore, the overall closed-loop gain \(A_v\) of amplifiers A_1 and A_2 is given by
\[
A_v = \frac{1}{\beta} = \sqrt{2} = 1.4142
\]  \hspace{1cm} (13.24)
which implies a constant gain of 1.4142. The design of a quadrature oscillator is very simple. For \(f_o = 200\) Hz and assuming \(C = 0.1 \mu F\), Eq. (13.22) gives \(R = 7958 \Omega\) (use a 10-k\(\Omega\) potentiometer). This oscillator can be stabilized by connecting two zener diodes back to back across one of the integrating capacitors, as shown in Fig. 13.9(a) by the shaded area.
13.3.3 Three-Phase Oscillators

A three-phase oscillator generates three sinusoidal voltages of equal magnitude, but displaced by 120° from each other. They have the same form as the voltages in a three-phase power system and are normally used for generating control signals synchronized to the power system. A three-phase oscillator consisting of three “lossy” integrator circuits connected in cascade with unity feedback is shown in Fig. 13.10. The transfer function of each of the integrators is given by

\[ A_1(s) = A_2(s) = A_3(s) = -\frac{R_F (1/C_s)}{1 + R_F C_s} \]  \hspace{1cm} (13.25)

Since \( \beta = 1 \) for unity feedback, the loop gain is given by

\[ \beta A(s) = A_1(s)A_2(s)A_3(s) = -\frac{(R_F/R)^3}{(R_F C_s)^3 + 3(R_F C_s)^2 + 3(R_F C_s) + 1} \]  \hspace{1cm} (13.26)

Then the characteristic equation, which is the numerator of \( 1 - \beta A(s) = 0 \), is

\[ (R_F C_s)^3 + 3(R_F C_s)^2 + 3(R_F C_s) + 1 + \left(\frac{R_F}{R}\right)^3 = 0 \]  \hspace{1cm} (13.27)

Substituting \( s = j\omega \) into Eq. (13.27) and then equating the imaginary part to zero, we get the frequency of oscillation \( \omega_o \) as

\[ \omega_o = \frac{\sqrt{3}}{R_F C} \text{ (in rad/s)} \]  \hspace{1cm} (13.28)
Equating the real part of Eq. (13.27) to zero at this frequency gives the condition of oscillation as \((R_F/R)^3 = 8\); that is,
\[
\frac{R_F}{R} = 2 \quad \text{(13.29)}
\]
Under this condition, the transfer function of each integrator at the oscillating frequency can be determined from Eq. (13.25). That is,
\[
A_1(j\omega) = A_2(j\omega) = A_3(j\omega) = -\frac{2}{1 + j\sqrt{3}} = 1 \angle 120^\circ \quad \text{(13.30)}
\]
If we select voltage \(v_a(t)\) as the reference so that
\[
v_a(t) = V_m \sin \omega t \quad \text{(13.31)}
\]
then \(v_b(t)\) and \(v_c(t)\) will be phase shifted by 120° and 240°, respectively. That is,
\[
v_b(t) = V_m \sin (\omega t + 120^\circ) \quad \text{(13.32)}
\]
and
\[
v_c(t) = V_m \sin (\omega t + 240^\circ) = V_m \sin (\omega t - 120^\circ) \quad \text{(13.33)}
\]
For \(f_o = 60\ \text{Hz}\) and assuming \(C = 0.1\ \mu\text{F}\), Eq. (13.28) gives
\[
R_F = \frac{\sqrt{3}}{2\pi \times 60 \times 0.1\ \mu\text{F}} = 45.94\ \text{k}\Omega \quad \text{use a 50-k}\Omega\ \text{potentiometer}
\]
and Eq. (13.29) gives \(R = R_F/2 = 22.97\ \text{k}\Omega\). This oscillator can be stabilized by connecting two zener diodes back to back across one of the integrating capacitors, as shown in Fig. 13.10 by the shaded area.

### 13.3.4 Wien-Bridge Oscillators

A Wien bridge, which is used for making measurements of unknown resistors or capacitors, is shown in Fig. 13.11(a). The bridge has a series RC network in one arm and a parallel RC network in the adjoining arm. \(R_1\) and \(R_F\) are connected in two other arms. While the bridge is making measurements, either \(R_1\) or \(R_F\) acts as a calibrated resistor; the resistance is varied until the null voltage \(V_d = 0\) is found. If all component values are known except one, the value of that one can be determined from the following relation:
\[
\frac{R_2}{R_3} + \frac{C_2}{C_1} = \frac{R_F}{R_1} \quad \text{(13.34)}
\]
If an op-amp is inserted into the basic bridge, as shown in Fig. 13.11(b), the bridge is known as a *Wien-bridge oscillator*, provided the elements are adjusted so that \(R_2 = R_3 = R\) and \(C_1 = C_2 = C\). The op-amp, along with \(R_1\) and \(R_F\), operates as a noninverting amplifier, as shown in Fig. 13.12(a). The Wien-bridge oscillator is one of the most commonly used AF oscillators.

The transfer function of the feedback network shown in Fig. 13.12(b) is given by
\[
\beta(s) = \frac{V_f(s)}{V_o(s)} = \frac{RCs}{R^2C^2s^2 + 3RCs + 1} \quad \text{(13.35)}
\]
(see Prob. 13.9).
The closed-loop voltage gain of the noninverting amplifier is given by

\[ A(s) = \frac{V_o(s)}{V_i(s)} = 1 + \frac{R_F}{R_1} \]  

(13.36)

For an oscillator, \( A\beta = 1 \). Using Eqs. (13.35) and (13.36), we get

\[ \left(1 + \frac{R_F}{R_1}\right) \frac{R C_s}{R^2 C^2 s^2 + 3 R C s + 1} = 1 \]  

(13.37)
Substituting \( s = j\omega \) into Eq. (13.37), we get

\[
\left(1 + \frac{R_F}{R_1}\right) jRC\omega = -R^2 C^2 \omega^2 + j3RC\omega + 1
\]

Equating real parts on the left side to those on the right side, we get

\[0 = -R^2 C^2 \omega^2 + 1\]

which gives the oscillation frequency as

\[f_0 = \frac{1}{2\pi RC} \text{ (in Hz)} \tag{13.38}\]

Equating imaginary parts on the left side to those on the right side, we get

\[
\left(1 + \frac{R_F}{R_1}\right) jRC\omega = j3RC\omega
\]

which gives the condition for oscillation as

\[1 + \frac{R_F}{R_1} = 3 \]

or

\[\frac{R_F}{R_1} = 2 \tag{13.39}\]

For stabilization, a power-sensitive resistor such as a lamp or a thermistor is usually used to adjust dynamically the loop gain of the oscillator. Figure 13.13(a) shows the use of a small incandescent lamp, whose resistance characteristic is shown in Fig. 13.13(b). When the filament of the lamp is cold, the resistance is small and the gain \( A \) is large. But when the lamp filament becomes hot, the resistance becomes larger, and the gain \( A \) becomes small. This automatic adjustment of the gain causes distortion of the amplifier to be low and stabilizes the oscillator. The nonlinear characteristic can be provided by two back-to-back zener diodes in series with a resistor \( R_B \), as in Fig. 13.6(a).

![Diagram of Wien-bridge oscillator with lamp and zener diodes](image-url)
EXAMPLE 13.4

Designing a Wien-bridge oscillator

(a) Design the Wien-bridge oscillator in Fig. 13.12(a) so that \( f_0 = 1 \text{ kHz} \).

(b) Use PSpice/SPICE to plot the transient response of the output voltage \( v_o(t) \) in part (a) from 0 to 2 ms. Assume \( V_{CC} = V_{EE} = 12 \text{ V} \).

SOLUTION

(a) The following steps can be used to complete the design.

Step 1. Choose a suitable value of \( C \): Let \( C = 0.01 \mu \text{F} \).

Step 2. Calculate the value of \( R \) from Eq. (13.38):

\[
R = \frac{1}{2\pi f_0 C} = \frac{1}{2\pi \times 1 \text{ kHz} \times 0.01 \mu \text{F}} = 15.915 \Omega
\]

Choose \( R = 16 \text{ k}\Omega \).

Step 3. Choose the value of \( R_F \) from Eq. (13.39). Letting \( R_1 = 10 \text{ k}\Omega \), we have

\[
R_F = 2R_1 = 2 \times 10 \text{ k}\Omega = 20 \text{ k}\Omega
\]

**NOTE:** The exact numbers of the resistances are used to demonstrate the validity of the calculated values with the PSpice simulation results. In a practical design, the commercially available standard values will be used (i.e., 16 k\Omega instead of 15.916 \Omega; see Appendix E).

(b) The Wien-bridge oscillator with the desired values is shown in Fig. 13.14. The PSpice plot of the output voltage \( v_o = V(U1:OUT) \) is shown in Fig. 13.15, which gives the peak-to-peak voltage \( V_{pp} = 5.94 + 5.97 = 11.91 \text{ V} \) at \( f_0 = 1 \text{ kHz} \) (expected value is 1 kHz).

**NOTE:** An initial voltage of 1 V has been assigned to the capacitor \( C_p \) in order to start the oscillator, and the UIC (use initial condition) is used in transient analysis. In practice, random noise or transients can cause the oscillations to begin, and they are sustained by the feedback of the appropriate signal.
13.3.5 Ring Oscillators

A ring oscillator consists of a number of gain stages in a loop in order to make the total phase shift of 180°. A three-stage ring oscillator is shown in Fig. 13.16. By neglecting the MOS capacitance, we can find the transfer function of each stage as given by

\[ A(s) = -g_m \left( R \frac{1}{sC} \right) = -\frac{g_m R}{1 + RCs} = -\frac{A_o}{1 + RCs} \]  

(13.40)

which in frequency domain becomes

\[ A(j\omega) = -\frac{A_o}{1 + (j\omega/\omega_n)} \]  

(13.41)

where \( A_o = g_m R \) is the small-signal low-frequency gain and \( \omega_n = 1/RC \) is the natural frequency. Since three stages are cascaded, the open-loop transfer function becomes

\[ H(j\omega) = -\frac{A_o^3}{1 + (j\omega/\omega_n)^3} \]  

(13.42)

The circuit oscillates only if the frequency-dependent phase shift equals \(-180^\circ\) so that the total phase of \( H(j\omega) = 360^\circ \) or \( 0^\circ \). Therefore, each stage must contribute \( \theta = 60^\circ \), which gives the condition for the oscillation frequency \( \omega_o \) as \( \tan^{-1}(\omega_o/\omega_n) = 60^\circ \). This gives

\[ \omega_o = \sqrt{3} \omega_n = \frac{\sqrt{3}}{RC} \]  

(13.43)

At the oscillation frequency of \( \omega = \omega_o \), the magnitude of the loop gain in Eq. (13.42) must equal 1; that is,

\[ |H(j\omega)| = \frac{A_o^3}{\left[1 + (\omega_o/\omega_n)^3\right]^3} = 1 \]  

(13.44)
which, for \( \omega_o = \sqrt{3} \omega_n \), gives the minimum value of \( A_o \) to sustain the oscillation as

\[
A_o = g_m R \geq 2
\]

(13.45)

**EXAMPLE 13.5**

**Designing a MOS ring oscillator**

(a) Design the ring oscillator in Fig. 13.16 so that \( f_o = 1 \) kHz. The MOSFET parameters are MOS constant \( K_p = 25 \mu A \) \( V^2 \), the threshold voltage \( V_t = 1 \) V, length \( L = 10 \) \( \mu m \), width \( W = 20 \) \( \mu m \), and modulation length \( \lambda = 0.01 \).

(b) Use PSpice [2] to plot the transient response of the output voltage \( v_o(t) \) in part (a) from 0 to 2 ms. Assume \( V_{DD} = 5 \) V.

**SOLUTION**

(a) Assuming that the MOS is operating in the saturation region, the transconductance of each MOS is \( g_m = 2K_p(v_{GS} - V_t) \), which can give the desired value of \( R \) to obtain the required gain \( A_o \) as given by

\[
R = \frac{A_o}{g_m} = \frac{A_o}{2K_p(v_{GS} - V_t)}
\]

(13.46)

Using KVL in the loop formed by \( V_{DD} \), \( R \), and the gate–source voltage of \( M_2 \), we can write

\[
V_{DD} = R i_{D1} + v_{GS1} = \frac{A_o}{2K_p(v_{GS1} - V_t)} \times K_p(v_{GS1} - V_t)^2 + v_{GS1}
\]

which, for identical MOSFETs of \( v_{GS} = v_{GS1} = v_{GS2} \), gives the gate–source voltage as

\[
v_{GS} = \frac{V_{DD} + (A_o/2)V_t}{1 + (A_o/2)}
\]

(13.47)

Letting \( A_o = 4 \) (higher than 2), Eq. (13.47) gives the DC biasing gate–source voltage

\[
v_{GS} = \frac{5 + (4/2) \times 1}{1 + (4/2)} = 2.333 \text{ V}
\]

\[
i_D = K_p(v_{GS1} - V_t)^2 = 25 \mu A \times (2.333 - 1)^2 = 44.44 \mu A
\]

\[
g_m = 2K_p(v_{GS} - V_t) = 2 \times 25 \mu A \times (2.333 - 1) = 66.67 \mu A/V
\]

\[
R = \frac{A_o}{g_m} = \frac{4}{66.67 \mu A/V} = 60 \text{ k}\Omega
\]

For \( f_o = 1 \) kHz, Eq. (13.43) gives

\[
C = \frac{\sqrt{3}}{2\pi f_o R} = \frac{\sqrt{3}}{2\pi \times 1 \times 60 \text{ k}} = 4.59 \text{ nF}
\]
(b) The PSpice schematic with desired values is shown in Fig. 13.17. The PSpice plot of the output voltage $v_o = V(C3:2)$ is shown in Fig. 13.18, which gives the peak-to-peak voltage of $V_{pp} = (3.7974 - 1.4154) = 2.38$ V at $f_o = 1 (2.6499 \text{ ms} - 1.7135 \text{ ms}) = 1046$ Hz (expected 1 kHz). Note that an initial voltage of 0 has been assigned to the capacitor $C_1$ in order to start the oscillator, and the UIC (use initial condition) is used in transient analysis. In practice, random noise or transients can cause the oscillations to begin, and they are sustained by the feedback of the appropriate signal.

![FIGURE 13.17 PSpice schematic for Example 13.5](image)

![FIGURE 13.18 Output voltage waveform for Example 13.5](image)

**KEY POINTS OF SECTION 13.3**

- A phase-shift oscillator uses an inverting amplifier and a phase-shifting network to satisfy the requirements of unity loop gain with $0^\circ$ or $360^\circ$ phase shift. The oscillation frequency $\omega_o$ is inversely proportional to the $RC$ product of the feedback network. A nonlinear device is often introduced, however, to stabilize the oscillator.
- A quadrature oscillator uses two op-amp inverting integrators and an $RC$ phase shifter. The output could be sine or cosine. The oscillation frequency $\omega_o$ is inversely proportional to the $RC$ product of the feedback network.
- A three-phase oscillator uses three inverting integrators with a unity feedback loop. The oscillation frequency $\omega_o$ is inversely proportional to the $RC$ product of the feedback network.
- A Wien-bridge oscillator uses a noninverting amplifier and an $RC$ phase-shifting network. The oscillation frequency $\omega_o$ is inversely proportional to the $RC$ product of the feedback network.
13.4 Radio Frequency Oscillators

A radio frequency (RF) oscillator uses $LC$ components for generating a waveform in the frequency range from 100 kHz to 100 MHz. There are different ways to make amplifiers by using op-amps, BJTs, and MOSFETs; choose the $LC$ components to create conditions for oscillations. We consider the following types of RF oscillators: Colpitts oscillator, Hartley oscillator, and two-stage MOS oscillator. The op-amp Colpitts oscillator is the most common and easy to implement. The MOS oscillators allow CMOS implementation.

13.4.1 Colpitts Oscillators

A Colpitts oscillator is a tuned $LC$-type oscillator, as shown in Fig. 13.19(a). $LC$ oscillators have the advantage of having relatively small reactive elements. They exhibit higher $Q$ than $RC$ oscillators, but they are difficult to tune over a wide range. For a positive feedback circuit to operate as an oscillator, the loop gain must be zero; that is,

$$1 - A\beta = 0$$

which is really the characteristic equation of the circuit. Therefore, the condition for oscillation can be found from the characteristic equation without deriving the transfer function. Nodal analysis can be applied to find the determinant, which is then set to zero.

The op-amp operates as an inverting amplifier of gain $A = \frac{R_F}{R_1}$. If the amplifier is replaced by its equivalent circuit, Fig. 13.19(a) can be simplified to Fig. 13.19(b). If the voltage source $AV_f$ is replaced by its equivalent current source $g_mV_f$, Fig. 13.19(b) can be reduced to Fig. 13.19(c). Using nodal analysis in Fig. 13.19(c) in Laplace’s domain of $s$, we can write

$$\left[ sC_2 + \frac{1}{R_L} + \frac{1}{sL} \right] V_o(s) + \left[ g_m - \frac{1}{sL} \right] V_f(s) = 0 \quad \text{(at node B)} \quad (13.48)$$

$$-\frac{1}{sL} V_o(s) + \left[ sC_1 + \frac{1}{sL} + \frac{1}{R_1} \right] V_f(s) = 0 \quad \text{(at node A)} \quad (13.49)$$

![Figure 13.19 Colpitts oscillator](image-url)
To find the condition for oscillation, we set the determinant to zero; that is,
\[ \left( sC_2 + \frac{1}{R_L} + \frac{1}{sL} \right) \left( sC_1 + \frac{1}{sL} + \frac{1}{R_1} \right) + \left( g_m - \frac{1}{sL} \right) \frac{1}{sL} = 0 \]
which, after simplification, yields
\[ s^3C_1C_2LR_1R_L + s^2L(C_1R_1 + C_2R_L) + s(C_1R_1R_L + C_2R_1R_L + L) + (R_1 + R_L + g_mR_1R_L) = 0 \]
(13.50)
where \( g_m = A R_L = R_F (R_1R_L) \). Substituting \( s = j\omega \) and equating the imaginary parts to zero, we get
\[ -j\omega^3(C_1C_2LR_1R_L) + j\omega(C_1R_1R_L + C_2R_1R_L + L) = 0 \]
which gives the frequency of oscillation \( \omega_o \) as
\[ \omega_o = \left[ \frac{C_1 + C_2}{C_1C_2L} + \frac{1}{C_1C_2R_1R_L} \right]^{1/2} \text{ (in rad/s)} \]
(13.51)
Assuming \( R_L \) is large, such that \( R_1R_L > 1 \) \( (C_1C_2) \), Eq. (13.51) can be approximated by
\[ \omega_o = \left[ \frac{C_1 + C_2}{C_1C_2L} \right]^{1/2} \text{ (in rad/s)} \]
(13.52)
Similarly, equating the real parts of Eq. (13.50) to zero yields
\[ -\omega^2L(C_1R_1 + C_2R_L) + (R_1 + R_L + g_mR_1R_L) = 0 \]
which gives
\[ \omega^2L(C_1R_1 + C_2R_L) = R_1 + R_L + g_mR_1R_L \]
Substituting the value of \( \omega = \omega_o \) from Eq. (13.51) gives
\[ L(C_1R_1 + C_2R_L) \left[ \frac{C_1 + C_2}{C_1C_2L} + \frac{1}{C_1C_2R_1R_L} \right] = R_1 + R_L + g_mR_1R_L \]
After simplification, the above equation becomes
\[ g_mR_1 \approx \frac{C_2}{C_1} + \frac{C_1}{C_2} \frac{R_1}{R_L} + \frac{L}{C_2R_L^2} + \frac{L}{C_1R_1R_L} \]
which, for a large value of \( R_L \), becomes
\[ g_mR_1 = \frac{C_2}{C_1} \]
(13.53)
or \[ \frac{AR_1}{R_L} = \frac{R_FR_1}{R_1R_L} = \frac{C_2}{C_1} \]
That is, \[ \frac{R_F}{R_L} = \frac{C_2}{C_1} \]
(13.54)
which is independent of \( R_1 \) and gives the relationship among \( R_L, R_F, C_1, \) and \( C_2 \). Equation (13.53) gives the minimum value of \( g_m \) (or \( R_F R_1 R_L \)) required to sustain the oscillation with a constant amplitude. If \( g_m \) is smaller than this value, the oscillation will die exponentially to zero. On the other hand, if \( g_m \) is larger than this value, the amplitude will grow exponentially until the nonlinearity of the op-amp limits the amplitude. Therefore, to ensure oscillation, the value of \( g_m \) must exceed the minimum value.

In the above analysis, we used a simple op-amp model and neglected the loss in the resistance of the inductor. As a result, we obtained relatively simple expressions for the frequency and the condition to sustain oscillation. If a complex op-amp model including the inductor loss were used, the oscillation frequency would depend (generally only slightly) on other circuit parameters. Usually, the inductor or one of the capacitors is made adjustable so that the frequency can be initially tuned to the desired value.

EXAMPLE 13.6

Designing a Colpitts oscillator Design the Colpitts oscillator of Fig. 13.19(a) so that the oscillating frequency is \( f_0 = 150 \text{ kHz} \).

SOLUTION

**Step 1.** Choose suitable values of \( C_1 \) and \( C_2 \): Let \( C_1 = 0.01 \mu\text{F} \) and \( C_2 = 0.1 \mu\text{F} \); that is, \( C_2 C_1 = 0.1 \times 0.01 = 10 \).

**Step 2.** Calculate the value of \( L \) from Eq. (13.52):

\[
L = \frac{C_1 + C_2}{4\pi^2 C_1 C_2 f_o^2} = \frac{0.01 \mu\text{F} + 0.1 \mu\text{F}}{4\pi^2 \times 0.01 \mu\text{F} \times 0.1 \mu\text{F} \times (150 \text{ kHz})^2} = 124 \mu\text{H}
\]

**Step 3.** Choose the values of \( R_F \) and \( R_L \) from Eq. (13.54):

\[
\frac{R_F}{R_L} = \frac{C_2}{C_1} = \frac{0.1}{0.01} = 10
\]

Let \( R_L = 100 \text{ k}\Omega \). Therefore, \( R_F = 10R_L = 1 \text{ M}\Omega \).

**Step 4.** Choose a value of \( A \): Let \( A = 10 = R_F R_1 \). Then

\[
R_1 = \frac{R_F}{A} = \frac{1 \text{ M}\Omega}{10} = 100 \text{ k}\Omega
\]

**Step 5.** Check the values of \( g_m \) and \( f_o \):

\[
g_m = \frac{A}{R_1} = \frac{10}{100 \text{ k}\Omega} = 0.1 \text{ mA/V}
\]

Equation (13.51) gives \( \omega_0 = 941.86 \text{ krad/s} \), and \( f_0 = \omega_0 / 2\pi = 149.9 \text{ kHz} \).
EXAMPLE 13.7

Finding the oscillation frequency of a Colpitts BJT-tuned oscillator A Colpitts BJT oscillator is shown in Fig. 13.20. The circuit parameters are \( r_e = 1.1 \, \text{k}\Omega, \, h_{fe} = 100, \, L = 1.5 \, \text{mH}, \, C_1 = 1 \, \text{nF}, \, C_2 = 99 \, \text{nF}, \) and \( R_L = 10 \, \text{k}\Omega. \) \( R_2, \, R_3, \) and \( R_E \) set the DC-biasing circuit. \( L, \, C_1, \) and \( C_2 \) form a parallel resonant circuit. The RF choke offers very high impedance at the frequency of oscillation and acts as a constant-current source feeding the LC resonant circuit.

(a) Calculate the frequency of oscillation \( f_o. \)

(b) Check to make sure the condition for oscillation is satisfied.

(c) Calculate the value of \( R_2. \)

**SOLUTION**

Since the RF choke offers a very high impedance, it can be considered as a short circuit for small-signal analysis. Thus, the AC equivalent circuit is as shown in Fig. 13.21(a). Replacing the transistor by its transconductance model (voltage-controlled current source) gives the small-signal AC equivalent circuit shown in Fig. 13.21(b), which is similar to the circuit shown in Fig. 13.21(c). Thus, the analysis of the Colpitts op-amp oscillator in Sec. 13.4.1 is applicable in this case:

\[
g_m = \frac{h_{fe}}{r_e} = \frac{100}{1.1 \, \text{k}\Omega} = 90.91 \, \text{mA/V}
\]

(a) From Eq. (13.52), the frequency of oscillation is

\[
f_o = \frac{1}{2\pi} \left[ \frac{C_1 + C_2}{C_1 C_2 L} \right]^{1/2} = \frac{1}{2\pi} \left[ \frac{1 \, \text{nF} + 99 \, \text{nF}}{1 \, \text{nF} \times 99 \, \text{nF} \times 1.5 \, \text{mH}} \right]^{1/2} = 130.6 \, \text{kHz}
\]

(b) For \( R_2 \gg r_e, \) \( R_1 \approx r_e \| R_2 \approx r_e = h_{fe} g_m. \) Therefore, Eq. (13.53) becomes

\[
g_m R_1 \approx g_m r_e = \frac{g_m h_{fe}}{g_m} = \frac{C_2}{C_1}
\]

**FIGURE 13.20** Colpitts BJT oscillator
which gives

\[ h_\text{fe} = \frac{C_2}{C_1} = \frac{99}{1} = 99 \]

This value is approximately equal to the value of transistor \( h_\text{fe} = 100 \). Thus, the condition for oscillation is satisfied.

(c) From Eq. (13.53),

\[ R_1 = \frac{C_2}{C_1 g_m} = \frac{99}{g_m} = \frac{99r_\pi}{h_\text{fe}} = 99 \times \frac{1.1\,\Omega}{100} = 1089\,\Omega \]

Since \( R_1 = r_\pi \parallel R_2 = 1089\,\Omega \),

\[ \frac{r_\pi R_2}{r_\pi + R_2} = 1089 \]

which gives \( R_2 = 108.9\,k\Omega \).

EXAMPLE 13.8

Finding the oscillation frequency of a Colpitts BJT oscillator  A Colpitts BJT oscillator is shown in Fig. 13.22. The circuit parameters are \( r_\pi = 2.7\,k\Omega \), \( h_\text{fe} = 150 \), \( L = 10\,\mu\text{H} \), \( C_1 = 1\,\text{nF} \), \( C_2 = 100\,\text{nF} \), and \( R_L = 10\,\text{k}\Omega \). The BJT acts like a normal amplifier, and the LC resonant signal is fed back to the input base in order to sustain the oscillation. Therefore, the configuration is different from the oscillator circuit in Fig. 13.20.

(a) Find the approximate value of the frequency of oscillation \( f_o \).

(b) Use PSpice/SPICE to plot the transient response of the output voltage \( v_o(t) \) from 20 ms to 2 ms.

SOLUTION

(a) From Eq. (13.52), the frequency of oscillation is

\[ f_o = \frac{1}{2\pi} \left[ \frac{C_1 + C_2}{C_1 C_2 L} \right]^{1/2} = \frac{1}{2\pi} \left[ \frac{1\,\text{nF} + 100\,\text{nF}}{1\,\text{nF} \times 100\,\text{nF} \times 10\,\mu\text{H}} \right]^{1/2} = 1.6\,\text{MHz} \]
(b) The PSpice plot of the output voltage $v_o = V(RL:2)$ is shown in Fig. 13.23, which gives $f_o = 1.5 \text{ MHz}$ (expected value is 1.6 MHz).

**NOTE:** An initial voltage of 1 V has been assigned to the capacitor $C_2$ in order to start the oscillator, and the UIC (use initial condition) is used in transient analysis.

---

**EXAMPLE 13.9**

**Finding the oscillation frequency of an LC-tuned MOSFET oscillator**  
An LC-tuned MOSFET oscillator is shown in Fig. 13.24. Find the values of $L$, $C$, and $n$ for an oscillation frequency of $f_o = 150 \text{ kHz}$. The parameters of the MOSFET are $g_m = 5 \text{ mA/V}$, $r_d = 25 \text{ k} \Omega$, and $R_G = 10 \text{ k} \Omega$. 

---

**FIGURE 13.22** BJT Colpitts oscillator for PSpice simulation

**FIGURE 13.23** PSpice plot of output voltage for Example 13.8

**FIGURE 13.24** Colpitts MOSFET oscillator
This oscillator uses the transformer to feed back the oscillating signal to the gate input, to sustain the oscillation. The small-signal AC equivalent circuit is shown in Fig. 13.25(a). Replacing the MOSFET by its transconductance model gives the small-signal equivalent circuit shown in Fig. 13.25(b), which can be simplified to Fig. 13.25(c). The transfer characteristic of $v_o$ versus $v_f$ in Laplace’s domain can be written as

$$\frac{v_o(s)}{v_i(s)} = Z(s) = \frac{1}{1/R_1 + sC + 1/sL}$$

which gives the voltage gain $A$ as

$$A(s) = \frac{v_o(s)}{v_i(s)} = \frac{g_m}{(1/R_1) + sC + (1/sL)}$$

where

$$R_1 = r_d \| \frac{R_G}{n^2}$$

Substituting $s = j\omega$ into Eq. (13.56), we get

$$A(j\omega) = \frac{g_m}{(1/R_1) + j(\omega C - 1/\omega L)}$$

For oscillation, $A = |A(j\omega)| = 1 \angle 0^\circ$. Therefore, the imaginary part of the denominator must equal zero; that is,

$$j(\omega C - \frac{1}{\omega L}) = 0$$

FIGURE 13.25 Equivalent circuits for Example 13.9
13.4.2 Hartley Oscillators

If the inductor and the capacitors of a Colpitts op-amp oscillator are interchanged, it becomes a Hartley op-amp oscillator, as shown in Fig. 13.26(a). Since inductors are more expensive than capacitors, this oscillator is less desirable than a Colpitts oscillator. Replacing the amplifier with its equivalent current source $g_m V_f$ reduces Fig. 13.26(a) to Fig. 13.26(b).

which gives the oscillation frequency $f_o$ as

$$f_o = \frac{\omega_o}{2\pi} = \frac{1}{2\pi \sqrt{LC}}$$

(13.59)

At this frequency, the gain must be unity; that is,

$$|A(j\omega)| = g_m R_1 = 1$$

(13.60)

Step 1. Choose a suitable value of $C$: Let $C = 0.01 \text{ \mu F}$.

Step 2. Calculate the value of $L$ from Eq. (13.59):

$$L = \frac{1}{4\pi^2 C f_o^2} = \frac{1}{4\pi^2 \times 0.01 \text{ \mu F} \times (150 \text{ kHz})^2} = 112.6 \text{ \mu H}$$

Step 3. Find the value of $R_1$. From Eq. (13.60),

$$R_1 = \frac{1}{g_m} = \frac{1}{\frac{5 \text{ mA/V}}{5 \text{ A/V}}} = \frac{1000}{200} = 200 \Omega$$

Step 4. Using Eq. (13.57), calculate the value of the turns ratio $n$: Since

$$200 = 25 \text{ k}\Omega \left( \frac{10 \text{ k}\Omega}{n^2} \right)$$

$n^2 = 49.6$ and $n = 7.04$. Thus, $n = 7$.

**FIGURE 13.26** Hartley oscillator
Using nodal analysis in Fig. 13.26(b), we can write

\[
\left[ sC + \frac{1}{R_L} + \frac{1}{sL_2} \right] V_0(s) + (g_m - sC) V_I(s) = 0 \quad \text{(at node B)} \tag{13.61}
\]

\[-sCV_0(s) + \left[ sC + \frac{1}{sL_1} + \frac{1}{R_1} \right] V_I(s) = 0 \quad \text{(at node A)} \tag{13.62}\]

To find the condition for oscillation, we set the determinant to zero; that is,

\[
\left( sC + \frac{1}{R_L} + \frac{1}{sL_2} \right) \left( sC + \frac{1}{sL_1} + \frac{1}{R_1} \right) + (g_m - sC)sC = 0
\]

which, after simplification, yields

\[
s^3CL_1L_2(R_1 + R_L + g_mR_1R_L) + s^2[CR_1R_L(L_1 + L_2) + L_1L_2] + s(L_1R_L + L_2R_1) + R_1R_L = 0 \tag{13.63}\]

where \( g_m = A \quad R_L = R_F \quad (R_1R_L) \). Substituting \( s = j\omega \) and equating the real parts of Eq. (13.63) to zero, we get

\[-\omega^2[CR_1R_L(L_1 + L_2) + L_1L_2] + R_1R_L = 0\]

which gives the frequency of oscillation \( \omega_o \) as

\[
\omega_o = \frac{1}{[C(L_1 + L_2) + L_1L_2/R_1R_L]^{1/2}} \quad \text{(in rad/s)} \tag{13.64}
\]

For \( C(L_1 + L_2) >> L_1L_2 \quad R_1R_L \), Eq. (13.64) can be approximated by

\[
f_0 = \frac{1}{2\pi} \left[ \frac{1}{C(L_1 + L_2)} \right]^{1/2} \quad \text{(in Hz)} \tag{13.65}
\]

Equating the imaginary parts of Eq. (13.63) to zero, we get

\[-j\omega^3CL_1L_2(R_1 + R_L + g_mR_1R_L) + j\omega(L_1R_L + L_2R_1) = 0\]

Substituting \( \omega = 2\pi f_0 \) into Eq. (13.65), we get

\[
\frac{1}{C(L_1 + L_2)} \times CL_1L_2(R_1 + R_L + g_mR_1R_L) = L_1R_L + L_2R_1
\]

which, solved for \( g_mR_1 \), gives

\[
g_mR_1 = \frac{L_1}{L_2} + \frac{R_1L_2}{R_1L_1} \tag{13.66}\]

For a large value of \( R_L \), Eq. (13.66) gives the approximate value of \( g_m \):

\[
g_m \approx \frac{L_2}{R_1L_1} \tag{13.67}\]
Equation (13.67) gives the minimum value of \( g_m \) required to sustain the oscillation with a constant amplitude. To ensure oscillation, the value of \( g_m \) must exceed the minimum value. The capacitor or one of the inductors is usually made adjustable so that the frequency can be initially trimmed to the desired value.

**EXAMPLE 13.10**

**Designing a Hartley oscillator**

(a) Design the Hartley oscillator shown in Fig. 13.27 so that \( f_0 = 5 \) MHz. Use a depletion NMOS whose parameters are \( K_p = 250 \) μA/V² and \( V_p = -2.83 \) V. The load resistance is \( R = 150 \) Ω, and the power supply voltage is \( V_{DD} = 15 \) V. This oscillator uses the capacitor \( C \) to feed back the oscillating signal to the gate input, to sustain the oscillation.

(b) Use PSpice to plot the transient response of the output voltage \( v_o(t) \) in part (a) from 10 μs to 10.5 μs. Assume \( V_{DD} = 15 \) V.

**SOLUTION**

The small-signal AC equivalent circuit is shown in Fig. 13.28, which is similar to Fig. 13.26(b):

\[
g_m = 2K_p |V_p| = 2 \times 250 \text{ μA/V}^2 \times |-2.83| = 1.415 \text{ mA/V}
\]

**Step 1.** Choose suitable values of \( L_1 \) and \( L_2 \): Let \( L_1 = L_2 = 10 \) μH.

**Step 2.** Calculate the value of \( C \) from Eq. (13.65):

\[
C = \frac{1}{4\pi^2 f_0^2 (L_1 + L_2)} = \frac{1}{4\pi^2 \times (5 \text{ MHz})^2 (10 \text{ μH} + 10 \text{ μH})} = 50.66 \text{ pF}
\]

**Step 3.** Find the value of effective load resistance \( R_L \). The Q-point of the circuit is \( V_{DSQ} = V_{DD} = 15 \) V, and \( V_{GSQ} = 0 \). The value of \( g_m \) varies from 3 mA V to 6.5 mA V. To ensure oscillation, we choose \( g_m = 3 \) mA V. Since \( R_1 \) is infinity in Fig. 13.28, Eq. (13.66) can be reduced to

\[
g_m = \frac{L_1}{R_1 L_2} + \frac{L_2}{R_2 L_1}
\]

**FIGURE 13.27** Hartley JFET oscillator

**FIGURE 13.28** Simplified equivalent oscillator for Example 13.10
13.4.3 Two-Stage MOS Oscillators

In the analyses of the Colpitts and Hartley oscillators, we assumed that inductors are lossless, with no resistances. In practice, inductors suffer from a resistive component that can be modeled as a parallel resistance with an ideal inductor. The parallel equivalent makes the analysis simpler. The series impedance can

which gives the value of effective load resistance $R_L$ as

$$R_L = \frac{L_2}{g_{mL_1}} = \frac{10 \ \mu\text{H}}{1.415 \ \text{mA/V} \times 10 \ \mu\text{H}} = 707 \ \Omega$$

This is the lowest value of $R_L$ at which sustained oscillation can occur for $g_m = 1.415 \ \text{mA/V}$. Therefore, we select a higher value—say, $R_L = 1.5 \ \text{k}\Omega$.

**Step 4.** Calculate the value of turns ratio $n$, which is related to the load resistance $R$ and the effective load resistance $R_L$, by

$$n = \frac{\sqrt{R}}{R_L} = \frac{\sqrt{100 \ \Omega}}{1.5 \ \text{k}\Omega} = 0.316$$

**Step 5.** Calculate the inductance of the transformer secondary $L_3$, which is related to $L_2$, by

$$L_3 = n^2 L_2 = 0.316^2 \times 10 \ \mu\text{H} = 1 \ \mu\text{H}$$

(b) The PSpice schematic with the desired values is shown in Fig. 13.29. The PSpice plot of the output voltage $v_o = V(R:2)$ is shown in Fig. 13.30, which gives the peak-to-peak voltage of $V_{pp} = +3.75 \ \text{V}$ to $-4.0 \ \text{V}$ at $f_0 = 1 \ (10.326 \ \mu\text{s} - 10.124 \ \mu\text{s}) = 4.95 \ \text{MHz}$ (expected 5 MHz). Note that an initial voltage of 2 V has been assigned to the capacitor C in order to start the oscillator, and UIC (use initial condition) is used in transient analysis. In practice, random noise or transients can cause the oscillations to begin, and they are sustained by the feedback of the appropriate signal.

**NOTE:** Increasing the value of $R$ will increase the output voltage and the readers are encouraged to simulate the circuit with different values of $R$. * symbols in Figs. 13.27 and 13.29 are the dot symbols of the transformer and refer to the direction of the positive induced voltages in the windings.
be transformed to an equivalent parallel topology as shown in Fig. 13.31(a). A two-stage MOS oscillator [3] is shown in Fig. 13.31(b). The drain of one transistor is connected to the gate of the other transistor, and it is also called the cross-coupled oscillator. The frequency-dependent phase around the loop is zero (0°) because each stage contributes to a zero frequency-dependent phase shift at resonance. The equivalent \( R_p \) and \( L_p \) can be determined by equating their impedances.

\[
j \omega L_s + R_s = \frac{R_p(j \omega L_p)}{R_p + j \omega L_p} \tag{13.68}
\]

or

\[
(L_s R_p + L_p R_s) j \omega L_s + R_s R_p - \omega^2 L_s L_p = j \omega L_s R_p \tag{13.69}
\]

Equating the real and imaginary parts on both sides, we get

\[
R_s R_p - \omega^2 L_s L_p = 0 \quad \text{and} \quad L_s R_p + L_p R_s = L_s R_p \tag{13.70}
\]

Solving for \( R_p \) and \( L_p \), we get

\[
L_p = L_s \left(1 + \frac{R_p^2}{L_s \omega^2}\right) = L_s \left(1 + \frac{1}{Q^2}\right) \tag{13.71}
\]

\[
R_p = \frac{\omega^2 L_s L_p}{R_s} \tag{13.72}
\]

where \( Q = \omega L_s / R_s \) is the quality factor of an inductor. For \( Q > 3 \), \( L_p \approx L_s \) and \( R_p \approx \omega^2 L_s^2 / R_s \). Assuming identical transistors, their transconductances and the output resistances will be equal, \( g_m = g_{m1} = g_{m2} \) and \( r_o = r_{o1} = r_{o2} \). The open-loop voltage gain is

\[
A_o = -g_{m1}(R_p \| r_{o1}) \times -g_{m1}(R_p \| r_{o1}) = g_m^2(R_p \| r_o)^2 \tag{13.73}
\]

To satisfy the oscillation condition, it requires a gain of 1 only—that is, \( A_o \approx 1 \)—which is an advantage of this oscillator. The oscillation frequency \( f_o \) is given by

\[
f_o = \frac{1}{2 \pi \sqrt{C_p L_p}} \tag{13.74}
\]

The MOS oscillator in Fig. 13.31(b) can be made to operate as a Colpitts oscillator by applying a voltage feedback from the drain to the source terminal of the MOSFET, as shown in Fig. 13.32(a). Capacitors \( C_1 \)
and $C_2$ form a potential divide. The equivalent capacitance $C_F$ offered by $C_1$ and $C_2$ is in parallel with $C_p$. To simplify the analysis, we will omit $C_p$ to derive the oscillation conditions, and the small-signal AC equivalent circuit is shown in Fig. 13.32(b). Using the equivalent circuit in Fig. 13.19(c), we can express the equivalent impedance of $M_1$, $C_1$, and $C_2$ as

$$Z_M = \frac{1}{sC_1} + \frac{1}{sC_2} + \frac{1}{sC_1} \frac{1}{sC_2} g_m = \frac{s(C_1 + C_2) + g_m}{s^2C_1C_2} \quad (13.75)$$

Therefore, the equivalent impedance of $Z_M$, $L_p$, and $R_p$ is given by

$$\frac{1}{Z_T} = \frac{1}{Z_M} + \frac{1}{sL_p} + \frac{1}{R_p} = \frac{s^2C_1C_2}{s(C_1 + C_2) + g_m} + \frac{1}{sL_p} + \frac{1}{R_p}$$

$$= \frac{s^3C_1C_2L_pR_p + s^2(C_1 + C_2)L_p + s(C_1 + C_2)R_p + sg_mL_p + g_mR_p}{[s(C_1 + C_2) + g_m]sL_pR_p} \quad (13.76)$$

For resonant oscillation, $Z_T(j\omega) = 0$, which gives the following expression:

$$(j\omega)^3C_1C_2L_pR_p + (j\omega)^2(C_1 + C_2)L_p + j\omega(C_1 + C_2)R_p + j\omega g_mL_p + g_mR_p = 0$$

Equating the imaginary part gives the resonant frequency as

$$\omega = \sqrt{\frac{g_mL_p + (C_1 + C_2)R_p}{C_1C_2L_pR_p}} = \sqrt{\frac{(C_1 + C_2)}{C_1C_2L_p}}$$

Equating the real part gives the gain as

$$g_mR_p = \omega^2(C_1 + C_2)L_p$$

which, for $\omega^2 = C_1 + C_2/C_1C_2L_p$, becomes

$$g_mR_p = \frac{(C_1 + C_2)^2}{C_1C_2} \quad (13.77)$$
EXAMPLE 13.11

Designing a MOS ring oscillator

(a) Design the MOS oscillator shown in Fig. 13.31(b) so that $f_o = 250$ kHz. The MOS parameters are MOS constant $K_p = 25$ $\mu$A/V$^2$, threshold voltage $V_t = 1$ V, length $L = 10$ $\mu$m, width $W = 10$ $\mu$m, and modulation $\lambda = 0.01$.

(b) Use PSpice to plot the transient response of the output voltage $v_o(t)$ in part (a) from $4000$ $\mu$s to $4020$ $\mu$s. Assume $V_{CC} = 12$ V.

SOLUTION

(a) The DC-biasing analysis is similar to the MOS ring oscillator in Fig. 13.16. Choosing a voltage gain of $A_o = 2$, Eq. (13.47) gives the DC biasing gate–source voltage

$$v_{GS} = \frac{5 + (2/2) \times 1}{1 + (2/2)} = 6.5 \text{ V}$$

$$i_D = K_p(v_{GS1} - V_t)^2 = 25 \mu\text{A/V}^2 \times (6.5 - 1)^2 = 765.2 \mu\text{A}$$

$$g_m = 2K_p(V_{GS} - V_t) = 2 \times 25 \mu\text{A/V}^2 \times (6.5 - 1) = 275 \mu\text{A/V}$$

$$R_p = \frac{\sqrt{A_o}}{g_m} = \frac{\sqrt{2}}{275 \mu\text{A/V}} = 5.1 \Omega$$

Let $C_p = 0.01$ $\mu$F. For $f_o = 250$ kHz, Eq. (13.74) gives

$$L_p = \frac{1}{(2\pi f_o)^2 C_p} = \frac{1}{(2\pi \times 250 \text{ kHz})^2 \times 0.01 \mu\text{F}} = 40 \mu\text{H}$$

(b) The PSpice schematic with the desired values is shown in Fig. 13.33. The PSpice plot of the output voltage $v_o = V(M2:d)$ is shown in Fig. 13.34, which gives the peak-to-peak voltage of $V_{pp} = 23,635$ V at $f_o = 1.0049$ ms $- 4.0009$ ms $= 250$ kHz (expected 250 kHz). Note that an initial voltage of 1.5 V has been assigned to the capacitor $C_{p1}$ in order to start the oscillator, and UIC (use initial condition) is used in transient analysis. In practice, random noise or transients can cause the oscillations to begin, and they are sustained by the feedback of the appropriate signal.
13.5 Crystal Oscillators

Because of their excellent frequency stability, quartz crystals are commonly used to control the frequency of oscillation. If the inductor L of the Colpitts oscillator in Fig. 13.19(a) is changed to a crystal, the oscillator is called a crystal oscillator. Crystal oscillators are commonly used in digital signal processing. The symbol for a vibrating piezoelectric crystal is shown in Fig. 13.35(a); its circuit model is shown in Fig. 13.35(b), which can be simplified to Fig. 13.35(c). The quality factor $Q$ of a crystal can be as high as several hundred thousands. $C_p$ represents the electrostatic capacitance between the two parallel plates of the crystal. $L$ has a large value (as high as hundreds of henries) and is determined from

$$L = \frac{1}{2\pi f_0 C_s}$$

where $f_0$ is the resonant frequency of the crystal. $R_s$ can be as high as a few hundred thousand ohms and is determined from

$$R_s = \frac{1}{2\pi f_0 L C_p}$$

$Q$ is very high in the typical quartz crystal, we may neglect $R_s$. The crystal impedance is given by

$$Z(s) = \frac{1}{sC_p + sL + 1/sC_s} = \frac{1}{sC_p} \left[ \frac{s^2 + 1/LC_s}{s^2 + (C_p + C_s)/(LC_sC_p)} \right]$$

$$= \frac{1}{sC_p} \left[ \frac{s^2 + \omega_0^2}{s^2 + \omega_p^2} \right]$$

(13.78)

<table>
<thead>
<tr>
<th>Table 13.1</th>
<th>Common cuts of quartz oscillator crystals (RCA Corp.)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Frequency</strong></td>
<td>32 kHz</td>
</tr>
<tr>
<td><strong>Cut</strong></td>
<td>XY bar</td>
</tr>
<tr>
<td><strong>$R_s$</strong></td>
<td>40 kΩ</td>
</tr>
<tr>
<td><strong>$L$</strong></td>
<td>4800 H</td>
</tr>
<tr>
<td><strong>$C_s$, in pF</strong></td>
<td>0.00491</td>
</tr>
<tr>
<td><strong>$C_p$, in pF</strong></td>
<td>2.85</td>
</tr>
<tr>
<td><strong>$C_p/C_s$</strong></td>
<td>580</td>
</tr>
<tr>
<td><strong>$Q$</strong></td>
<td>25,000</td>
</tr>
</tbody>
</table>
If we substitute $s = j\omega$, the impedance in Eq. (13.78) becomes

$$Z(j\omega) = -\left(\frac{j}{\omega C_p}\right)\left(\frac{\omega^2 - \omega_s^2}{\omega^2 - \omega_p^2}\right)$$

(13.79)

Therefore, the crystal exhibits two resonant frequencies: series resonance at

$$\omega_s = \frac{1}{\sqrt{LC_s}}$$

(13.80)

and parallel resonance at

$$\omega_p = \left[\frac{C_s + C_p}{C_s C_p L}\right]^{1/2}$$

(13.81)

Note that $\omega_p > \omega_s$. However, since $C_p >> C_s$, the two resonance frequencies are very close. The plot of crystal reactance against frequency in Fig. 13.35(d) illustrates that the crystal exhibits the characteristic of an inductor over the narrow frequency range between $\omega_s$ and $\omega_p$.

It is possible to have a variety of crystal oscillators. A Colpitts-derived op-amp crystal oscillator is shown in Fig. 13.36(a); its equivalent circuit is shown in Fig. 13.36(b). This circuit should oscillate at the resonance frequency of the crystal inductance $L$ with the series equivalent of $C_s$ and $C_p + C_1 C_2$ ($C_1 + C_2$). Since $C_s$ is much smaller than $C_p$, $C_1$, or $C_2$, it will be dominant and the oscillating frequency can be approximately found from

$$\omega_o \approx \frac{1}{\sqrt{LC_s}}$$

(13.82)

Using nodal analysis in Fig. 13.36(b), we can write

$$\left[\frac{s C_2 + \frac{1}{R_c} + s L_c}{R_1} \right] V_o(s) + \left[ g_m - \frac{1}{R_c + s L_c} \right] V_i(s) = 0$$

(13.83)

$$-\frac{1}{R_c + s L_c} V_o(s) + \left[ \frac{s C_1 + \frac{1}{R_c + s L_c} + \frac{1}{R_1}}{R_1} \right] V_i(s) = 0$$

(13.84)
Assuming $R_L$ is large, tending to infinity, we set the determinant to zero to find the condition for oscillation; that is,

$$\left(sC_2 + \frac{1}{R_c + sL_c}\right)\left(sC_1 + \frac{1}{R_c + sL_c} + \frac{1}{R_1}\right) + \left(g_m - \frac{1}{R_c + sL_c}\right)\frac{1}{R_c + sL_c} = 0$$

which, after simplification, yields

$$s^3C_1C_2L_cR_1 + s^2(C_1C_2R_cR_1 + C_2L_c) + s(C_1R_1 + C_2R_1 + C_2R_c) + 1 + g_mR_1 = 0 \quad (13.85)$$

where $g_m = A R_L = R_F (R_1R_L)$. Substituting $s = j\omega$ and equating the imaginary parts of Eq. (13.85) to zero, we get

$$-j\omega^3(C_1C_2L_cR_1) + j\omega(C_1R_1 + C_2R_1 + C_2R_c) = 0$$

which gives the frequency of oscillation $f_o$ as

$$f_o = \frac{1}{2\pi} \left[ \frac{C_1R_1 + C_2R_1 + C_2R_c}{C_1C_2L_cR_1} \right]^{1/2} \text{ (in Hz)} \quad (13.86)$$

Similarly, equating the real parts of Eq. (13.85) to zero, we get

$$-\omega^2(C_1C_2R_cR_1 + C_2L_c) + 1 + g_mR_1 = 0$$

which gives

$$1 + g_mR_1 = \omega^2(C_1C_2R_cR_1 + C_2L_c)$$

After substitution of the value of $\omega = \omega_o = 2\pi f_o$ from Eq. (13.86), the above equation becomes

$$1 + g_mR_1 = \frac{(C_1R_1 + C_2R_1 + C_2R_c)(C_1C_2R_cR_1 + C_2L_c)}{C_1C_2L_cR_1} \quad (13.87)$$
Since $Q$ is very high, $R_c = 0$, and Eqs. (13.86) and (13.87) can be reduced to

$$f_o = \frac{1}{2\pi} \left[ \frac{C_1 + C_2}{C_1 C_2 L_c} \right]^{1/2} \quad \text{(in Hz)}$$

(13.88)

and

$$1 + g_m R_1 = \frac{C_1 + C_2}{C_1} = 1 + \frac{C_2}{C_1}$$

or

$$g_m R_1 = \frac{C_2}{C_1}$$

(13.89)

which is the same condition as expressed in Eq. (13.53) for the Colpitts oscillator in Fig. 13.19(a).

**EXAMPLE 13.12**

**Finding the oscillation frequency of a crystal oscillator** The op-amp oscillator in Fig. 13.36(a) uses a 2-MHz crystal and has $C_1 = 0.01 \mu F$, $C_2 = 0.1 \mu F$, $R_L = 100 \, k\Omega$, $R_1 = 100 \, k\Omega$, and $R_F = 1 \, M\Omega$.

(a) Find the frequency of oscillation $f_o$.

(b) Use PSpice/SPICE to verify the frequency in part (a). Assume $V_{CC} = V_{EE} = 15 \, V$.

**SOLUTION**

(a) For a 2-MHz crystal, $C_s = 0.0122 \, pF$, $C_p = 4.27 \, pF$, $R_s = 82 \, \Omega$, and $L = 0.52 \, H$. Let

$$C_{eq} = \frac{C_p + C_1 C_2}{C_1 + C_2}$$

$$= \frac{4.27 \, pF + 0.01 \, \mu F \times 0.1 \, \mu F}{0.01 \, \mu F + 0.1 \, \mu F} = 9095 \, pF$$

The effective capacitance $C_{eq}$ is given by $C_{eq}$ in series with $C_{eq} = C_p + C_1 C_2$ ($C_1 + C_2$):

$$C_{eq} = \frac{C_s \times C_{eq}}{C_s + C_{eq}}$$

$$= \frac{0.0122 \, pF \times 9095 \, pF}{0.0122 \, pF + 9095 \, pF} \approx 0.0122 \, pF$$

Thus, the frequency of oscillation $f_o$ becomes

$$f_o = \frac{1}{2\pi} \left[ \frac{1}{C_{eq} L} \right]^{1/2} = \frac{10^6}{2\pi \sqrt{0.0122 \times 0.52}} = 1.998 \, MHz$$

(b) The circuit for PSpice simulation is shown in Fig. 13.37. The PSpice plot of the output voltage across $C_s$ [i.e., $v_o = V(C_s \cap)$] is shown in Fig. 13.38, which gives $f_o = 1 \, (763.146 \, n - 243.137 \, n) = 1.923 \, MHz$, close to the calculated frequency of 1.998 MHz. The amplitude of the output voltage is stable, not falling.
NOTE: An initial voltage of 5 V has been assigned to the capacitor C₂ in order to start the oscillator, and the UIC (use initial condition) is used in transient analysis. In practice, random noise or transients can cause the oscillations to begin, and they are sustained by the feedback.

### KEY POINT OF SECTION 13.5

- A crystal oscillator has the same circuit topology as a Colpitts oscillator except that it uses a crystal instead of an inductor. A crystal has strong frequency stability.

### 13.6 Active-Filter Tuned Oscillators

An active band-pass filter with a high $Q$-value can be operated as an oscillator provided that a positive feedback is applied. This type of oscillator, which consists of a narrow-band filter and a limiter, is illustrated in Fig. 13.39(a). To understand the operation of the circuit, let us assume that the oscillation has already started. The output of the filter $v_o$ is a sine wave whose frequency is the center frequency of the filter $f_o$. This sine wave is fed to a limiter, which produces a square-wave output $v_l$ of frequency $f_o$. The peak amplitude of the square wave is determined by the type of limiting devices. The square wave is in turn fed back to the band-pass filter, which filters the harmonics and produces a sinusoidal output $v_o$ at the fundamental frequency $f_o$. The quality of the sine wave is a direct function of the selectivity ($Q$-factor) of the band-pass filter. The design of this type of oscillator is very simple, and the oscillator has independent frequency control.
A typical practical implementation of an active-filter tuned oscillator using a narrow-band filter is shown in Fig. 13.39(b). An inverter is added to the output of the filter in order to provide positive feedback. A simple diode limiter along with resistance \( R_L \) is used to generate a square wave, which is fed back to the input of the filter.

**EXAMPLE 13.13**

**PSpice simulation of an active-filter tuned oscillator**  Modify the narrow-band filter that was designed in Example 12.10 so that it operates as an oscillator, and use PSpice/SPICE to plot the output voltage.

**SOLUTION**

The circuit for PSpice simulation is shown in Fig. 13.40. Instead of an inverter, a voltage-controlled voltage source with \(-1\) gain is used to provide the positive feedback required for oscillation.
The PSpice plots of the output voltage $v_o$ at the output of the op-amp and the voltage across the diodes $v_{D1} = V(RL:1)$ are shown in Fig. 13.41. We get the oscillation frequency $f_o = 1 \ (5.913 \text{ m} - 4.8548 \text{ m}) = 962 \text{ Hz}$ (expected value is 1 kHz), the peak amplitude of the output voltage is ±5.17 V, and the peak amplitude of the diode limiter is ±557 mV.

**KEY POINT OF SECTION 13.6**

An active-filter tuned oscillator uses a narrow-band filter with positive feedback. A limiter is used in the feedback path to provide square-wave input signals to the filter. This type of oscillator has the advantages of independent frequency and amplitude control.
13.7 Design of Oscillators

The design of a sinusoidal oscillator involves the following steps:

**Step 1.** Identify the specifications of the output stage—for example, oscillation frequency \( f_0 \), load resistance \( R_L \), and the DC supply voltages \( V_{CC} \) and \( V_{EE} \) (or \( V_{DD} \) and \( V_{SS} \)).

**Step 2.** Select the type of oscillator and the circuit topology, depending on the oscillation frequency and the types of devices available, such as BJTs, MOSFETs, or op-amps.

**Step 3.** Analyze the circuit, and find the component values such that the condition \( \beta \) satisfies \( \angle 0^\circ \) or \( \angle 360^\circ \) is satisfied.

**Step 4.** Limit the output voltage by introducing nonlinearity to stabilize the oscillator, if necessary.

**Step 5.** Use PSpice/SPICE to simulate and verify your design. Use the standard values of components with their tolerances.

**EXAMPLE 13.14**

**Worst-case analysis of the phase-shift oscillator in Example 13.3** Use PSpice/SPICE to find worst-case output voltage and frequency ranges of the phase-shift oscillator in Example 13.3. Use standard component values: \( C = 0.1 \mu F \pm 10\% \), \( R = 1.6 \, k\Omega \pm 5\% \), \( R_1 = 17 \, k\Omega \pm 5\% \), and \( R_F = 490 \, k\Omega \pm 5\% \).

**SOLUTION**

To assign tolerances to resistors and capacitors, we will use model RMOD for resistors and CMOD for capacitors. Also, we will add a statement for the worst-case analysis (.WCASE) [2].

The PSpice plots of the worst-case maximum and nominal output voltages \( v_o = V(3) \) are shown in Fig. 13.42(a), which give the worst-case peak values of 8.7 V and \(-9.1 \) V and an oscillation frequency of 1 2.322 ms = 430 Hz (nominal value is 422 Hz). The plots of the worst-case minimum and nominal output voltages \( v_o = V(3) \) are shown in Fig. 13.42(b), which give the worst-case peak values of 8.05 V and \(-9.45 \) V and an oscillation frequency of 1 2.246 ms = 445 Hz (nominal value is 422 Hz). Thus, the output frequency can vary from 430 Hz to 445 Hz based on component tolerances.

**FIGURE 13.42** Worst-case output of the phase-shift oscillator for Example 13.14
Summary

Oscillators use positive feedback and are commonly employed in electronics circuits. There are many types of oscillators. However, RC, LC, and crystal oscillators are the most commonly used. For a feedback circuit to operate as an oscillator, the magnitude and the phase shift of the loop gain must be unity and 0° (or 360°), respectively. Frequency stability is an important criterion in defining the quality of an oscillator. Crystal oscillators have the highest frequency stability. The frequency of oscillation and the conditions for oscillation can be determined from the transfer function or the determinant of a circuit.

References


Review Questions

1. What is an oscillator?
2. What are the two conditions for oscillation?
3. What are the major types of oscillators?
4. What is an RC oscillator?
5. What is an LC oscillator?
6. What is a crystal oscillator?
7. What is the frequency stability of oscillators?
8. What is the figure of merit of an oscillator?
9. What is a phase-shift oscillator?
10. What is a Wien-bridge oscillator?
11. What is a quadrature oscillator?
12. What is a Colpitts oscillator?
13. What is a Hartley oscillator?
14. What is a crystal oscillator?
15. What is an active-filter tuned oscillator?

Problems

The symbol indicates that a problem is a design problem. The symbol indicates that you can check the solution to a problem using PSpice/SPICE or Electronics Workbench.

13.2 Principles of Oscillators

13.1 For the circuit in Fig. P13.1, determine the values of A and θ that will produce a steady-state sinusoidal oscillation.
13.2 The amplifier in Fig. 13.3 has a voltage gain of $A = 200$, input resistance $R_i = 50 \, \text{k}\Omega$, and output resistance $R_o = 500 \, \text{Ω}$. Find the values of $R$, $R_3$, $C$, and $L$ so that the oscillation frequency is $f_0 = 5 \, \text{kHz}$.

13.3 *Audio-Frequency Oscillators*

13.3 Derive the transfer function $\beta(s)$ of the feedback network shown in Fig. 13.5(b).

13.4 Design a phase-shift oscillator as shown in Fig. 13.5(a) so that $f_0 = 1 \, \text{kHz}$.

13.5 Find the values of $R$ and $C$ for the phase-shift oscillator in Fig. P13.5 so that the oscillation frequency is $f_0 = 5 \, \text{kHz}$.

13.6 Find the values of $R$, $R_3$, $C$, and $L$ for the phase-shift oscillator in Fig. P13.6 so that the oscillation frequency is $f_0 = 5 \, \text{kHz}$. 
13.7 Design a quadrature oscillator as shown in Fig. 13.9(a) so that \( f_o = 500 \) Hz.

13.8 Design a three-phase oscillator as shown in Fig. 13.10 so that \( f_o = 50 \) Hz.

13.9 Derive the transfer function \( \beta(s) \) of the feedback network in Fig. 13.12(b).

13.10 Design a Wien-bridge oscillator as shown in Fig. 13.12(a) so that \( f_o = 5 \) kHz.

13.11 Design the MOS ring oscillator in Fig. 13.16 so that \( f_o = 2 \) kHz. The MOSFET parameters are MOS constant \( K_p = 25 \) \( \mu \)A/V\(^2\), threshold voltage \( V_t = 1 \) V, length \( L = 10 \) \( \mu \)m, width \( W = 20 \) \( \mu \)m, and modulation length \( \lambda = 0.01 \). Use PSpice to plot the transient response of the output voltage \( v_o(t) \) in part (a) from 0 to 1 ms. Assume \( V_{DD} = 5 \) V.

13.12 Determine the oscillation frequency \( f_o \) for the ring oscillator in Fig. 13.16 if \( R = 40 \) k\( \Omega \) and \( C = 6 \) nF. The MOSFET parameters are MOS constant \( K_p = 25 \) \( \mu \)A/V\(^2\), threshold voltage \( V_t = 1 \) V, length \( L = 10 \) \( \mu \)m, width \( W = 20 \) \( \mu \)m, and modulation length \( \lambda = 0.01 \). Use PSpice to plot the transient response of the output voltage \( v_o(t) \) to verify the result. Assume \( V_{DD} = 5 \) V.

13.4 Radio Frequency Oscillators

13.13 Design a Colpitts oscillator as shown in Fig. 13.19(a) so that the oscillation frequency is \( f_o = 500 \) kHz.

13.14 A Colpitts BJT oscillator is shown in Fig. 13.20. The circuit parameters are \( r_n = h_{ie} = 500 \) \( \Omega \), \( h_{fe} = 200 \), \( L = 1.5 \) mH, \( C_1 = 10 \) nF, \( C_2 = 10 \) nF, and \( R_L = 5 \) k\( \Omega \). Calculate the frequency of oscillation \( f_o \) and the value of \( R_1 \) required to sustain the oscillation.

13.15 Design a Colpitts BJT oscillator as shown in Fig. 13.20 so that \( f_o = 250 \) kHz. The circuit parameters are \( r_n = h_{ie} = 500 \) \( \Omega \), \( h_{fe} = 200 \), and \( R_L = 5 \) k\( \Omega \).
13.16 An LC-tuned MOSFET oscillator is shown in Fig. 13.24. Find the values of $L$, $C$, and $n$ for an oscillation frequency of $f_o = 100$ kHz. The parameters of the MOSFET are $g_m = 7.5$ mA/V, $r_d = 50$ kΩ, and $R_G = 20$ kΩ.

13.17 A Colpitts BJT oscillator is shown in Fig. P13.17. Calculate the frequency of oscillation $f_o$ and the value of $R_{E1}$ required to sustain the oscillation.

**FIGURE P13.17**

13.18 Design a Colpitts oscillator as shown in Fig. 13.19(a) so that the oscillation frequency is $f_o = 5$ kHz. Assume $V_{CC} = V_{EE} = 12$ V.

13.19 The Colpitts oscillator of Fig. 13.19(a) has $C_1 = 400$ pF, $C_2 = 200$ pF, and $L = 1$ mH. Determine the frequency of oscillation $f_o$ and the minimum value of gain $A = R_E/R_1$ needed to sustain the oscillation.

13.20 Determine the frequency of oscillation for the Colpitts MOSFET oscillator in Fig. P13.20(a). The MOSFET can be replaced by its transconductance model, shown in Fig. P13.20(b). The parameters are $r_d = 25$ kΩ, $g_m = 5$ mA/V, $R_G = 1$ MΩ, $L = 1.5$ mH, $C_1 = 10$ nF, and $C_2 = 10$ nF. Calculate the frequency of oscillation and check to make sure the condition for oscillation is satisfied.

**FIGURE P13.20**

13.21 Design a Hartley oscillator as shown in Fig. 13.27 so that $f_o = 500$ kHz. Use a 2N3821 n-channel JFET whose parameters are $I_{DSS} = 0.5$ mA to 2.5 mA, $V_p = -4$ V, and $g_m = 1.5$ mA/V to 4.5 mA/V. The load resistance is $R = 50$ Ω, and the power supply voltage is $V_{DD} = 15$ V.
13.22 Design the MOS oscillator in Fig. 13.32(b) so that \( f_o = 200 \text{ kHz} \). The MOS parameters are MOS constant 
\( K_p = 25 \text{ \mu A/V}^2 \), threshold voltage \( V_t = 1 \text{ V} \), length \( L = 10 \text{ \mu m} \), width \( W = 10 \text{ \mu m} \), and modulation \( \lambda = 0.01 \). Use PSpice to plot the transient response of the output voltage \( v_o(t) \) in part (a) from 4000 \( \mu \text{s} \) to 4020 \( \mu \text{s} \). Assume \( V_{CC} = 12 \text{ V} \).

13.23 Determine the oscillation frequency \( f_o \) for the ring oscillator in Fig. 13.31(a) if \( R_p = 3.5 \text{ k\Omega} \) and \( L_p = 50 \text{ \mu H} \). The MOSFET parameters are MOS constant \( K_p = 25 \text{ \mu A/V}^2 \), threshold voltage \( V_t = 1 \text{ V} \), length \( L = 10 \text{ \mu m} \), width \( W = 20 \text{ \mu m} \), and modulation length \( \lambda = 0.01 \). Use PSpice to plot the transient response of the output voltage \( v_o(t) \) to verify the result. Assume \( V_{DD} = 12 \text{ V} \).

13.5 Crystal Oscillators

13.24 Design a crystal oscillator as shown in Fig. 13.36(a) so that \( f_o = 14 \text{ MHz} \). The crystal parameters are \( f_o = 10 \text{ MHz}, Q = 150 \times 10^3, C_s = 0.0145 \text{ pF}, C_p = 4.35 \text{ pF}, \) and \( L = 12 \text{ mH} \). Assume a transconductance gain of \( |g_m| = 1 \text{ mA/V} \) for the amplifier.

13.6 Active-Filter Tuned Oscillators

13.25 The equivalent circuit of a tuned oscillator is shown in Fig. P13.25. Derive the expressions for the oscillation condition and the frequency of oscillation.

**FIGURE P13.25**

![Equivalent circuit of a tuned oscillator](image)

13.26 Design an active-filter tuned oscillator as shown in Fig. 13.39(b) so that \( f_o = 10 \text{ kHz} \). Assume \( R_L = 10 \text{ k\Omega} \) and \( V_{CC} = V_{EE} = 12 \text{ V} \).
CHAPTER 14

OPERATIONAL AMPLIFIERS

Learning Outcomes
After completing this chapter, students should be able to do the following:

• Describe the internal structure and types of op-amps (CMOS, bipolar, and BiCMOS) and the effects of amplifier configurations (difference and single-ended output) on the op-amp performance.

• Identify the circuit parameters of the internal design that affect the op-amp performance (offset voltage, offset current, and unity-gain frequency).

• List the typical values of performance parameters for different types of op-amps.

• Analyze op-amp circuits to determine the DC-biasing conditions and the performance parameters.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>( i_{B1}, i_{B2} )</td>
<td>Instantaneous base currents of transistors 1 and 2</td>
</tr>
<tr>
<td>( i_{Bn}, i_{Cn}, i_{En} )</td>
<td>Base, collector, and emitter currents of an ( n )th BJT ( Q_n )</td>
</tr>
<tr>
<td>( G_m )</td>
<td>Transconductance of an amplifier</td>
</tr>
<tr>
<td>( I_{Sn} )</td>
<td>Saturation current of an ( n )th BJT ( Q_n )</td>
</tr>
<tr>
<td>( K_{pn}, V_{tn} )</td>
<td>MOS constant and threshold voltage of an ( n )th MOSFET ( M_n )</td>
</tr>
<tr>
<td>( r_{on} )</td>
<td>Output resistance of an ( n )th BJT ( Q_n ) or MOSFET ( M_n )</td>
</tr>
</tbody>
</table>
14.1 Introduction

So far we have discussed separately the analysis and design of transistor amplifiers, differential amplifiers, and output stages. An operational amplifier normally consists of these stages. In this chapter, we examine the internal circuitry of 10 commercially available op-amps. Much of the circuitry will be closely related to that of other ICs. We analyze in detail one of the oldest but most popular amplifiers, the LM741.

14.2 Internal Structure of Op-Amps

The general configuration of an op-amp is shown in Fig. 14.1 [1]. All stages are direct coupled—that is, there are no coupling or bypass capacitors. Since capacitors and resistors of over 50 kΩ occupy large areas on IC chips and exhibit parasitic effects, they are usually avoided in ICs. Therefore, op-amp circuits are designed using transistors with matching characteristics. Mismatches do exist, however, and cause offset voltages.

![General configuration of an op-amp](image-url)
The voltage gain of each of the differential and gain stages is normally in the range from 300 to 1000. The output stage is an emitter follower, and its gain is unity. Thus, the overall open-loop voltage gain of an op-amp is on the order of $10^5$ to $10^6$. The output stage provides a low-output impedance so that it can drive a load with relatively low values of load resistance. It is normally operated in class AB mode to reduce crossover distortion. In general, BJT op-amps have a larger voltage gain, whereas MOSFET op-amps have higher input resistances.

For stability of the op-amp, the phase shift of each stage is kept to a minimum. Since each stage contributes to the phase shift, the total number of stages is generally limited to three. A compensation capacitor $C_x$ is normally connected across the second stage.

**NOTE** Unless specified, we will assume that the base-emitter voltage is $V_{BE} = 0.6$ V.

---

**KEY POINT OF SECTION 14.2**

- In general, an op-amp circuit consists of a differential input stage, a gain stage, an output stage, and protection circuitry. Each stage uses active biasing and an active load.

---

### 14.3 Parameters and Characteristics of Practical Op-Amps

Most op-amps contain a differential-coupled pair as an input stage. Practical op-amps exhibit characteristics that deviate significantly from the ideal characteristics [2, 3]. These deviations are discussed in detail in Chapter 6. Here we consider some parameters that depend on the internal design of the op-amp. The effects of deviations from the ideal can be incorporated in the equivalent circuit of the op-amp, as shown in Fig. 14.2 where $R_{cm} = R_{ic}$ is the common-mode input resistance.

**FIGURE 14.2** Equivalent circuit of an op-amp
14.3.1 Input Biasing Current

The input biasing current $I_B$ is defined as the average of the two DC input currents to the bases of $Q_1$ and $Q_2$; that is,

$$I_B = \frac{I_{B1} + I_{B2}}{2} \quad (14.1)$$

$$\approx \frac{I_C}{\beta_F} = \frac{I_Q}{2\beta_F} \quad \text{for BJT input stage only} \quad (14.2)$$

The polarity of $I_B$ is shown for an $nnp$ transistor input stage in Fig. 14.2. For a $pnp$ input stage, $I_B$ would flow out of the amplifier terminals. Transistors in an ideal op-amp are assumed to have a large value of $\beta_F$, tending to infinity, and to draw zero DC input current. In a practical op-amp, however, $\beta_F$ has a finite value. Thus, $I_B$ has a small but finite value. The typical magnitudes of the biasing currents are 10 pA to 100 nA for BJT input devices and 1 nA to 10 pA for MOSFET input devices. This input biasing current will cause a small DC-output voltage when the external input voltage is zero.

The effect of biasing current $I_B$ can be determined for the inverting or noninverting amplifier, as shown Fig. 14.3. Assuming $I_{B1} = I_{B2} = I_B$, then $V_d = V_1 (I_{B1} - I_{B2}) = 0$. Since $V_d = 0$ due to the feedback of $V_{ob}$, there will be no current flowing through $R_1$; that is, $i_S = 0$, and the biasing current $I_B$ will flow through $R_F$. Thus,

$$V_{ob} = R_F I_B$$

which gives the output voltage due to input biasing current $I_B$ as

$$V_{ob} = R_F I_B \quad (14.3)$$

Thus, the output offset voltage due to the input biasing current $I_B$ depends directly on the feedback resistance $R_F$. To minimize the effect of $I_B$, the value of $R_F$ should be small. However, the ratio $R_F / R_1$ determines the voltage gain $v_{oV} / v_S$. The output voltage due to $I_B$ does not depend on whether the input voltage source $v_S$ is connected. Thus, Eq. (14.3) is applicable for both inverting and noninverting amplifiers.

![Figure 14.3 Inverting or noninverting amplifier with input biasing current](image-url)
The effect of $I_B$ on the output voltage can be eliminated or minimized by making Thevenin’s equivalent resistance at the ($/H_11002$) terminal equal to that at the ($/H_11001$) terminal. This arrangement can be implemented by connecting a resistance $R_x$ to the noninverting terminal, as shown in Fig. 14.4, such that

$$V_+ = V_- \quad \text{or} \quad R_{Th+} = R_{Th-}$$

which gives the offset-minimizing resistance $R_x$ as

$$R_x = \frac{R_1 R_F}{R_1 + R_F} = R_1 \parallel R_F$$

(14.4)

For example, if $R_F = 100 \, \text{k}\Omega$ and $I_B = 500 \, \text{nA}$, then $V_{ob} = 100 \, \text{k}\Omega \times 500 \times 10^{-9} = 50 \, \text{mV}$ without $R_x$, and $V_{ob} = 0$ if $R_x = (10 \, \text{k}\Omega \parallel 100 \, \text{k}\Omega) = 9.091 \, \text{k}\Omega$. Therefore, by connecting resistance $R_x$, which is equal to the parallel combination of $R_1$ and $R_F$, we can minimize the output voltage due to the input biasing current. Since $I_{B1}$ and $I_{B2}$ are not exactly equal, $V_{ob}$ will be minimized but not completely eliminated. The connections of the input voltage $v_S$ and the input/output relations are shown in Fig. 14.5 for inverting and noninverting amplifiers.
$R_x$ was included in the op-amp circuits in Chapter 3, although it is not necessary for an ideal op-amp. However, while an op-amp circuit is being built, $R_x$ should be connected. For op-amps with an FET input stage (e.g., the LF411 op-amp), the input biasing current is very low (50 pA), and the absence of offset-minimizing resistance $R_x$ should not introduce any significant error.

### 14.3.2 Input Offset Current

The DC input base currents will be equal only if the two transistors have equal current gains (betas). However, even two theoretically identical transistors right next to each other on an IC chip will not be exactly identical. Geometrically identical devices on the same IC die typically display a mismatch that is normally distributed with a standard deviation of 5% to 10% of the mean value. This mismatch in the two biasing currents is random from circuit to circuit and is described by the input offset current $I_{OS}$, which is defined as the difference between the two base currents of the transistors:

$$I_{OS} = |I_{B1} - I_{B2}|$$  \hspace{1cm} (14.5)

$I_{OS}$ arises from mismatches in the area and $\beta_F$ of the transistors. Beta and collector current values typically deviate by 10% and 1%, respectively. Thus, $I_{OS}$ can be found approximately from

$$I_{OS} = \frac{(0.1 + 0.01)I_C}{\beta_F} = \frac{0.11I_Q}{2\beta_F} = \frac{0.055I_Q}{\beta_F}$$  \hspace{1cm} (14.6)

The value of the bias-minimizing resistance $R_x$ in Eq. (14.4) was derived by assuming equal input-biasing currents: $I_{B1} = I_{B2} = I_B$. In practice, these currents are not equal because of internal imbalances within the op-amp circuit. The input offset current $I_{io}$ is a measure of the degree of mismatching, and it is defined by

$$I_{io} = I_{B1} - I_{B2}$$  \hspace{1cm} (14.7)

For the μA741C op-amp, $I_{io}$ is quoted as having a maximum absolute value of 200 nA and a typical value of 30 nA; it can be either positive or negative. An amplifier with offset-minimizing resistance $R_x$ is shown in Fig. 14.6. The effective input voltage at the noninverting terminal is given by

$$V_+ = -R_x I_{B1}$$  \hspace{1cm} (14.8)

![FIGURE 14.6 Inverting or noninverting amplifier with input offset current](image-url)
Since the differential voltage between the input terminals is \( v_d = 0 \), \( V_- = V_+ \). Assuming the op-amp draws negligible current and applying KCL at the inverting terminal, we get
\[
\frac{V_+}{R_1} + I_{B2} + \frac{V_+ - V_{oi}}{R_F} = 0
\]
which gives
\[
\frac{V_{oi}}{R_F} = \frac{V_+}{R_1 || R_F} + I_{B2}
\]  
(14.9)
Substituting \( V_+ = -R_x I_{B1} \), we get the output offset voltage
\[
V_{oi} = R_F \left( \frac{-R_x}{R_1 || R_F} I_{B1} + I_{B2} \right)
\]  
(14.10)
Substituting \( R_x (= R_1 || R_F) \) from Eq. (14.4) into Eq. (14.10) gives the output voltage as
\[
V_{oi} = R_F (I_{B2} - I_{B1}) = -R_F I_{io}
\]  
(14.11)
For example, if \( R_F = 100 \text{ k}\Omega \) and \( I_{io} = \pm 200 \text{ nA} \), then
\[
V_{oi} = \pm 100 \times 10^3 \times 200 \times 10^{-9} = \pm 20 \text{ mV} \quad \text{(DC)}
\]
To minimize the effect of \( I_{io} \), the value of \( R_F \) should be small.

**NOTE** If \( R_x \) is zero or is absent, Eq. (14.8) is to be used. If \( R_x = R_1 || R_F \), Eq. (14.11) is to be used.

**EXAMPLE 14.1**

Finding the effects of offsets on the output of an op-amp integrator

The inverting integrator in Fig. 14.7 has \( R_1 = 1 \text{ k}\Omega \), \( R_x = 1 \text{ k}\Omega \), \( C_F = 0.1 \mu\text{F} \), \( V_{CC} = 15 \text{ V} \), \( V_{EE} = -15 \text{ V} \), and maximum saturation voltage \( = \pm 14 \text{ V} \). The op-amp parameters are \( V_{io} = 6 \text{ mV} \), \( I_B = 500 \text{ nA} \), and \( I_{io} = 200 \text{ nA} \) at \( 25^\circ \text{C} \).

(a) Determine the total output offset voltage \( v_{of} \).

(b) Repeat part (a) if \( R_x = 0 \)

![Figure 14.7](image-url)
SOLUTION

$R_1 = 1 \text{k}\Omega$, $R_x = 1 \text{k}\Omega$, $V_{io} = 6 \text{ mV}$, $I_B = 500 \text{nA}$, and $I_{io} = 200 \text{nA}$. The equivalent circuit of the integrator with input offset voltage and input biasing currents is also shown in Fig. 14.7.

(a) $R_x = R_1$. The output voltage will be due to $V_{io}$ and $I_{io}$. With $V_{io} = 0$, we get

$$v_+ = -R_x I_{B1} = -R_1 I_{B1}$$
$$v_- = v_+ = -R_1 I_{B1}$$
$$i_S = \frac{v_-}{R_1}$$
$$i_f = i_S - I_{B2} = I_{B1} - I_{B2}$$

With $I_{B1} = I_{B2} = 0$, we get

$$i_f = \frac{-V_{io}}{R_1}$$

Applying the superposition theorem, we get for the current flowing through the capacitor $C_F$

$$i_f = i_S + I_{B1} - I_{B2} = \frac{-V_{io}}{R_1} + I_{io} \tag{14.12}$$

where $I_{io} = (I_{B1} - I_{B2})$ is the input offset current. The total output offset voltage due to the capacitor current $i_f$ can be found from

$$-v_{of} = \frac{1}{C_F} \int i_f \, dt + (-V_{io} + R_x I_{B1}) + v_C(t = 0) \tag{14.13}$$

where $v_C(t = 0)$ is the initial capacitor voltage. Substituting $i_f$ from Eq. (14.12) into Eq. (14.13) gives the total output offset voltage as

$$-v_{of} = \frac{1}{C_F} \int \left( \frac{-V_{io}}{R_1} + I_{io} \right) \, dt + (-V_{io} + R_x I_{B1}) + v_C(t = 0)$$
$$v_{of} = \frac{V_{io}}{C_F R_1} t - \frac{I_{io}}{C_F} t + V_{io} - R_x I_{B1} - v_C(t = 0) \tag{14.14}$$

This equation indicates that the output offset voltage will rise linearly until the output reaches the saturation voltage of the amplifier, which is $\pm 14 \text{ V}$ in this example. If the power supplies are turned on and enough time is allowed, the output will build up to the saturation voltage, even without any external input signal to the integrator. For this reason, this is not a practical circuit (as discussed in Sec. 3.5); it needs a DC feedback resistor $R_F$, as shown in Fig. 3.16. After the power supply is switched on, the time required for the total output offset voltage to reach the saturation level $v_{of} = 14 \text{ V}$ can be found from Eq. (14.14) with $v_C(t = 0) = 0$:

$$14 = \left( \frac{6 \times 10^{-3}}{0.1 \times 10^{-6} \times 1 \times 10^3} - \frac{200 \times 10^{-9}}{0.1 \times 10^{-6}} \right) t + 6 \times 10^{-3} - 1 \text{k}\Omega \times 600 \times 10^{-9} - 0$$

which gives $t = 241.3 \text{ ms}$. 
If the input terminals of an op-amp are tied together and connected to the ground, as shown in Fig. 14.8(a), a certain DC voltage exits at the output. This voltage is called the output offset voltage $V_{oo}$. The input offset voltage is the differential input voltage that exists between two terminals without any external inputs applied. In other words, it may be regarded as the input voltage that should be applied between the input terminals to force the output voltage to zero, as shown in Fig. 14.8(b). If $V_{oo}$ is divided by the voltage gain $A_o$ of the op-amp, the result is the input offset voltage $V_{io}$. Assuming that the output voltage is not saturated, $V_{fo}$ can be determined from

$$V_{io} = \frac{V_{oo}}{A_o} \text{ for } |V_{oo}| \leq |V_{sat}|$$  \hspace{1cm} (14.16)

Input offset voltage is quoted as an absolute value, and it may be positive or negative. The maximum value of $V_{io}$ for the μA741C is ±6 mV. If the output voltage is saturated to $V_{sat}$, as is usually the case, Eq. (14.16) is valid only if $|V_{oo}| \leq |V_{sat}|$. The polarity of $V_{io}$ is unpredictable, and so is the output offset voltage $V_{oo}$.

(b) If $R_k = 0$, the total output offset voltage will be due to the input offset voltage, the input biasing current, $-I_{B2} = -I_B + (I_{io}/2)$, and the input offset current, and it can be found from Eq. (14.14) by replacing $I_{io}$ by $I_{B2}$. That is,

$$V_{of} = \left( \frac{V_{io}}{C_F R_1} + \frac{I_B - I_{io}/2}{C_F} \right) t + V_{io} - V_C(t = 0)$$

(14.15)

The time required for the output voltage to reach the saturation level $v_{of} = 14$ V can be found from Eq. (14.15) with $V_C(t = 0) = 0$:

$$14 = \left( \frac{6 \times 10^{-3}}{0.1 \times 10^{-6} \times 1 \times 10^3} + \frac{400 \times 10^{-9}}{0.1 \times 10^{-6}} \right) t + 6 \times 10^{-3} + 0$$

which gives $t = 218.7$ ms.
The output offset voltage is caused by internal mismatching in the input stage. A simple differential pair, as shown in Fig. 9.2(b), consists of two transistors Q1 and Q2. Any differential signal between the input terminals is amplified and gives the output voltage $V_{oo}$. In practice, the characteristics of the two transistors will not be exactly the same; therefore, the collector-biasing currents $I_{C1}$ and $I_{C2}$ will differ. As a result, even without any input voltages, there could be a differential output voltage, which is amplified in subsequent stages and possibly aggravated by more mismatching.

The effect of the input offset voltage can be determined for the inverting and noninverting amplifiers in Fig. 14.9. For both configurations, $V_{io}$ may be considered as the input to the noninverting terminal, since there is no other input signal. Applying Eq. (3.18) gives the output offset voltage:

$$V_{oo} = (1 + \frac{R_F}{R_1})V_{io}$$  \hspace{1cm} (14.17)

For example, if $R_1 = 10 \text{k}\Omega$, $R_F = 100 \text{k}\Omega$, and $V_{io} = \pm 6 \text{ mV}$, then

$$V_{oo} = \pm \left(1 + \frac{100 \text{k}\Omega}{10 \text{k}\Omega}\right) \times 6 \text{ mV} = \pm 66 \text{ mV}$$

That is, the output voltage $V_{oo}$ can be $\pm 66 \text{ mV}$ (DC) without any external input signal $v_S$ applied.

### 14.3.4 Power Supply Rejection Ratio

So far we have assumed that the DC supply voltages $V_{CC}(V_{DD})$ and $V_{EE}(V_{SS})$ have no effect on the output voltage. In practice, the power supply voltages change, causing the DC-biasing currents of the internal transistors to change. As a result, the input offset voltage will also change. The power supply rejection ratio (PSRR) is defined as the change in input offset voltage per unit change in the DC supply voltage. If $\Delta V_{io}$ is the change in input offset voltage due to a change in the DC supply voltage $\Delta V_{DC}$, PSRR is expressed as

$$\text{PSRR} = \frac{\Delta V_{io}}{\Delta V_{DC}}$$  \hspace{1cm} (14.18)

$$= 20 \log \left| \frac{\Delta V_{io}}{\Delta V_{DC}} \right|$$  \hspace{1cm} (14.19)
This ratio is also known as the supply voltage rejection ratio (SVRR) or the power supply sensitivity (PSS). The maximum value of PSRR for the μA741C op-amp is 150 μV V. For example, if the DC supply voltages change from \( V_{\text{DC}} = \pm 15 \text{ V} \) to \( \pm 12 \text{ V} \) and PSRR = 150 μV V, then

\[
\Delta V_{\text{DC}} = 2 \times 15 - 2 \times 12 = 6 \text{ V}
\]

and

\[
\Delta V_{\text{io}} = \text{PSRR} \Delta V_{\text{DC}} = 150 \mu \text{V} \times 6 = 900 \mu \text{V}
\]

### 14.3.5 Thermal Voltage Drift

In previous sections, we assumed that the input offset voltage \( V_{\text{io}} \), input biasing current \( I_{\text{B}} \), and input offset current \( I_{\text{io}} \) remain constant. A practical op-amp consists of devices such as diodes and transistors whose parameters change with temperature. **Thermal drift** is a measure of the change in an offset parameter due to a unit change in temperature. **Thermal voltage drift** is defined as the rate of change of input offset voltage \( V_{\text{io}} \) per unit change in temperature, and it is expressed as

\[
D_v = \frac{\Delta V_{\text{io}}}{\Delta T} \quad (\text{V/}^{\circ}\text{C}) \quad (14.20)
\]

**Thermal-biasing current drift** is defined as the rate of change of input biasing current \( I_{\text{B}} \) per unit change in temperature, and it is expressed as

\[
D_b = \frac{\Delta I_{\text{B}}}{\Delta T} \quad (\text{A/}^{\circ}\text{C}) \quad (14.21)
\]

**Thermal input offset current drift** is defined as the rate of change of input offset current \( I_{\text{io}} \) per unit change in temperature, and it is expressed as

\[
D_i = \frac{\Delta I_{\text{io}}}{\Delta T} \quad (\text{A/}^{\circ}\text{C}) \quad (14.22)
\]

Thus, the output voltage due to drifts can be found from

\[
V_{\text{od}} = \left( 1 + \frac{R_F}{R_1} \right) \Delta V_{\text{io}} + R_F \Delta I_{\text{io}} = \left( 1 + \frac{R_F}{R_1} \right) D_v \Delta T + R_F D_i \Delta T
\]

\[
= \left( 1 + \frac{R_F}{R_1} \right) D_v \Delta T + R_F D_b \Delta T \quad \text{for } R_x = 0 \quad (14.23)
\]

\[
= \left( 1 + \frac{R_F}{R_1} \right) D_v \Delta T + R_F D_b \Delta T \quad \text{for } R_x = 0 \quad (14.24)
\]

### EXAMPLE 14.2

**Finding the effects of thermal drift on the output of an inverting op-amp circuit** The inverting amplifier in Fig. 14.5(a) has \( R_1 = 10 \text{ k} \Omega, \ R_F = 100 \text{ k} \Omega \), and \( R_x = R_F \parallel R_1 = 9.091 \text{ k} \Omega \). The op-amp parameters are \( V_{\text{io}} = \pm 6 \text{ mV}, \ I_{\text{B}} = 500 \text{ nA, } \ I_{\text{io}} = \pm 200 \text{ nA, and PSRR = 150 } \mu \text{V V}\). The thermal drifts are \( D_v = 15 \mu \text{V }^{\circ}\text{C}, \ D_i = 0.5 \text{ nA }^{\circ}\text{C}, \) and \( D_b = 0.5 \text{ nA }^{\circ}\text{C} \) at 25°C. The temperature is 55°C. The DC supply voltages change from \( V_{\text{CC}} = 15 \text{ V} \) to 12 V and \( -V_{\text{EE}} = -15 \text{ V} \) to -12 V. The input voltage is \( v_S = 100 \text{ mV (DC)} \). Determine the output voltage \( v_O \) if (a) \( R_x = R_F \parallel R_1 = 9.091 \text{ k} \Omega \) and (b) \( R_x = 0 \).
SOLUTION

\( R_1 = 10 \, \text{k}\Omega, \, R_F = 100 \, \text{k}\Omega, \, V_{io} = \pm 6 \, \text{mV}, \, I_B = 500 \, \text{nA}, \, I_{io} = \pm 200 \, \text{nA}, \, D_v = 15 \, \mu\text{V/°C}, \, D_I = 0.5 \, \text{nA/°C}, \, D_B = 0.5 \, \text{nA/°C}, \) and \( V_S = 100 \, \text{mV}. \) Then

\[ \Delta T = 55 - 25 = 30 \, ^\circ\text{C} \]

\[ \Delta V_{DC} = (15 + 15) - (12 + 12) = 6 \, \text{V} \]

\[ \Delta V_{io} = D_v \Delta T + \text{PSRR} \Delta V_{DC} = 15 \times 10^{-6} \times 30 + 150 \times 10^{-6} \times 6 = 1.35 \, \text{mV} \]

\[ \Delta I_{io} = D_I \Delta T = 0.5 \times 10^{-9} \times 30 = 15 \, \text{nA} \]

\[ \Delta I_B = D_B \Delta T = 0.5 \times 10^{-9} \times 30 = 15 \, \text{nA} \]

(a) With offset-minimizing resistance \( R_x, \) the total output voltage of the inverting amplifier is given by

\[ v_O = -\frac{R_F}{R_1} v_S \pm \left( 1 + \frac{R_F}{R_1} \right) (V_{io} + \Delta V_{io}) \pm R_F (I_{io} + \Delta I_{io}) \]  \hspace{1cm} (14.25)

\[ = -\left( \frac{100 \, \text{k}\Omega}{10 \, \text{k}\Omega} \right) \times 100 \times 10^{-3} \pm \left( 1 + \frac{100 \, \text{k}\Omega}{10 \, \text{k}\Omega} \right) \times (6 + 1.35) \times 10^{-3} \]

\[ \pm 100 \times 10^3 \times (200 + 15) \times 10^{-9} \]

\[ = -1000 \, \text{mV} \pm 80.85 \, \text{mV} \pm 21.5 \, \text{mV} \]

\[ = -1102.35 \, \text{mV} \, \text{(min)} \, \text{or} \, -897.65 \, \text{mV} \, \text{(max)} \]

(b) With \( R_x = 0, \) the total output voltage of the inverting amplifier is given by

\[ v_O = -\frac{R_F}{R_1} v_S \pm \left( 1 + \frac{R_F}{R_1} \right) (V_{io} + \Delta V_{io}) + R_F (I_B + \Delta I_B) \]  \hspace{1cm} (14.26)

\[ = -\left( \frac{100 \, \text{k}\Omega}{10 \, \text{k}\Omega} \right) \times 100 \times 10^{-3} \pm \left( 1 + \frac{100 \, \text{k}\Omega}{10 \, \text{k}\Omega} \right) \times (6 + 1.35) \times 10^{-3} \]

\[ + 100 \times 10^3 \times (500 + 15) \times 10^{-9} \]

\[ = -1000 \, \text{mV} \pm 80.85 \, \text{mV} + 51.5 \, \text{mV} \]

\[ = -1029.4 \, \text{mV} \, \text{(min)} \, \text{or} \, -867.65 \, \text{mV} \, \text{(max)} \]

14.3.6 Determining the Thermal Voltage Drift

Because of mismatches, an output voltage will exist even when the external input is zero. The differential voltage that must be applied to the input terminals of an amplifier to drive the output to zero is called the input offset voltage \( V_{io}. \) This input offset voltage is a function of temperature. The rate of change of the input offset voltage \( V_{io} \) per unit change in temperature is known as the thermal voltage drift, and it is expressed as

\[ D_v = \frac{\Delta V_{io}}{\Delta T} \, \text{ (in V/°C)} \]  \hspace{1cm} (14.27)
**BJT Amplifiers**

The difference in the B-E voltages will cause an offset voltage. Using Eqs. (9.129) and (9.130), we can express this offset voltage as

\[
v_{BE2} - v_{BE1} = V_T \left[ \ln \frac{i_{C2}}{I_{S2}} - \ln \frac{i_{C1}}{I_{S1}} \right] = V_T \ln \left( \frac{i_{C2}}{i_{C1}} \times \frac{I_{S1}}{I_{S2}} \right)
\]  
\[ (14.28) \]

Since the mismatch in the collector current is generally small (i.e., \(i_{C1} \approx i_{C2}\)), we can find the input offset voltage \(V_{io}\) from

\[
V_{io} = v_{BE2} - v_{BE1} = V_T \ln \left( \frac{I_{S1}}{I_{S2}} \right)
\]  
\[ (14.29) \]

Thus, the saturation current \(I_S\) is the prime contributing factor to \(V_{io}\) in a BJT amplifier and is proportional to the transistor base width \(W_B\). Its value can vary from one transistor to another. Assuming that \(W_{B1} = W_B\) and \(W_{B2} = W_B + \Delta W_B\) are the base widths of transistors Q1 and Q2, respectively, Eq. (14.29) becomes

\[
V_{io} = V_T \ln \left( \frac{W_B}{W_B + \Delta W_B} \right) = V_T \ln \left( \frac{1}{1 + \Delta W_B/W_B} \right)
\]  
\[ (14.30) \]

For \(W_B >> \Delta W_B\), which is usually the case, Eq. (14.30) can be approximated by

\[
V_{io} = V_T \left( \frac{\Delta W_B}{W_B} \right)
\]  
\[ (14.31) \]

The beta of a transistor is inversely proportional to the base width \(W_B\). Therefore, any change in \(W_B\) will cause an almost equal change in beta; that is, \(\Delta \beta_F / \beta_F = \Delta W_B / W_B\). Since the change in the base width is generally within 10%, the offset voltage can be found approximately from

\[
V_{io} = 0.1 V_T
\]  
\[ (14.32) \]

For BJT input devices, this value is typically 2 mV to 5 mV (for compound devices) and can often be nulled with an external potentiometer.

The ratio \(\Delta W_B / W_B\) is relatively independent of temperature. Since \(V_T = kT q\), \(dV_T / dT = k q = V_T / T\). Thus, we can find the thermal drift from Eq. (14.31) as follows:

\[
D_v = \frac{dV_{io}}{dT} = \frac{V_T}{T} \left( \frac{\Delta W_B}{W_B} \right) = \frac{V_{io}}{T}
\]  
\[ (14.33) \]

For example, if \(V_{io} = 2.6\) mV and \(T = 25^\circ C = 273 + 25 = 298\) K,

\[
D_v = \frac{2.6\text{ mV}}{298} = 8.72 \mu V/K
\]

**CMOS Amplifiers**

From Eq. (9.54), the transconductance of a MOSFET is given by

\[
g_m = 2K_p(V_{GS} - V_i) = \frac{2I_D}{V_{GS} - V_i}
\]  
\[ (14.34) \]
If $\Delta I_D$ is the difference between the drain currents of MOSFETs, an input offset voltage $\Delta V_{GS} = V_{OS}$ must be applied between the two gates to cancel the difference. Any increase in $\Delta V_{GS}$ will cause an increase in $\Delta I_D$. However, any differential increase $\Delta V_t$ in the threshold voltages will cause a decrease in drain current. That is, Eq. (14.19) gives

$$\Delta I_D = \frac{g_m}{2} (\Delta V_{GS} - \Delta V_t) = \frac{g_m}{2} (V_{io} - \Delta V_t)$$

which gives the input offset voltage $V_{io}$ as

$$V_{io} = \Delta V_t + \frac{\Delta I_D}{I_D} (V_{GS} - V_t)$$  \hspace{1cm} (14.35)

The ratio $\Delta I_D/I_D$ will depend on any change in the $W/L$ ratio of the MOSFET. Thus, Eq. (14.35) can be written in terms of the $W/L$ ratio:

$$V_{io} = \Delta V_t + \frac{\Delta (W/L)}{W/L} (V_{GS} - V_t)$$  \hspace{1cm} (14.36)

In CMOS amplifiers, the value of $\Delta V_t$ alone, which is absent in BJTs, can be as high as 2 mV—a magnitude as large as $V_{io}$ for BJTs. The value of $(V_{GS} - V_t)$ is usually much greater than $V_T$. Thus, the offset voltage of CMOS amplifiers will generally be considerably larger than that of BJT amplifiers. To have a low value of $V_{io}$, MOSFETs should be operated with a low value of $(V_{GS} - V_t)$.

For MOSFET amplifiers, thermal drift cannot be correlated with the offset voltage. The thermal drift of $\Delta V_t$ $dT$ can be quite significant.

### 14.3.7 Offset Voltage Adjustment

We saw in the previous section that an op-amp can have an output voltage $V_{oo}$ without any external input signal. Op-amps are generally compensated internally; they have built-in offset adjustment terminals, as shown in Fig. 14.10. The output voltage can be adjusted to zero by an offset null potentiometer.

**FIGURE 14.10** Op-amp with compensating terminals
The recommended value of the potentiometer is normally quoted in the data sheet; it is 10 kΩ for the μA741 series. By varying the potentiometer, the output offset voltage can be adjusted to zero within a certain input offset voltage adjustment range (±15 mV for the μA741 op-amp).

It is possible to compensate for the offset voltage by injecting a small voltage into the (+) terminal or the (−) terminal of an op-amp. An offset-compensating network is shown in Fig. 14.11(a). The potentiometer \( R \) is varied to produce an input offset voltage \( V_{io} \), which should be just adequate to negate the output offset voltage. Thevenin’s equivalent circuit of the network is shown in Fig. 14.11(b); the equivalent resistance of the compensating network is \( R_{nt} = 0 \), shown in Fig. 14.11(c), and the equivalent voltage is \( V_{nt} = V_{CC} = V_{EE} \), shown in Fig. 14.11(d). From the voltage division rule, the voltage \( V_x \), which should be equal to \( V_{io} \), is given by assuming \( R_b \gg R_{nt} \):

\[
V_x \approx V_{io} = \frac{R_c V_{nt}}{R_{nt} + R_b + R_c}
\]  

The values for the network should be such that they do not alter the normal operation of the amplifier. That is, \( R_b > R_{nt} \) so that \( R_b \) does not affect \( V_{nt} \) significantly; \( R_b > > R_c \) so that the op-amp biasing current flows mostly through \( R_c \) (usually \( \leq 100 \, \Omega \)). The following relations are recommended:

\[
R_b \approx 10R_{nt} \quad \text{[where } R_{nt(max)} = (R/2) \parallel (R/2) = R/4]\]

\[
R_b \approx 1000R_c
\]

For \( R_b > R_{nt} > R_c \), Eq. (14.37) can be approximated by

\[
V_{io} \approx \frac{R_c V_{nt}}{R_b} = \frac{R_c V_{CC}(=V_{EE})}{R_b}
\]

The compensating network can be used in inverting, noninverting, and differential amplifiers, as shown in Fig. 14.12. For a voltage follower, Fig. 14.12(a) can be modified by making \( R_1 = 0 \) and \( R_F = \infty \) and then connecting one side of \( R_c \) to the output side instead of connecting to the ground.
EXAMPLE 14.3

Designing an offset-compensating network  The noninverting amplifier in Fig. 14.12(a) has $R_1 = 10 \, \Omega$, $R_F = 100 \, \Omega$, and $R_x = R_F \parallel R_1 = 9.091 \, \Omega$. Design the offset-compensating network. The op-amp parameters are $V_{io} = 6 \, \text{mV}$, $I_B = 500 \, \text{nA}$, $I_{io} = 200 \, \text{nA}$, and PSRR = 150 $\, \mu$V. The DC supply voltages are $V_{CC} = 15 \, \text{V}$ and $-V_{EE} = -15 \, \text{V}$.

SOLUTION

Since the offset due to the biasing current $I_B$ is minimized by the resistance $R_x$, the output offset voltage will be contributed mostly by $V_{io}$. For $V_{io} = 6 \, \text{mV}$ and $V_{int} = V_{CC} = 15 \, \text{V}$, Eq. (14.38) gives $6 \, \text{mV} = 15R_c \, R_b$.

Letting $R_c = 10 \, \Omega$, we get

$$R_b = \frac{15R_c}{6 \, \text{mV}} = 25 \, \text{k}\Omega$$

Letting $R_b = 10R_{int(max)} = 10(R_{4})$, we get

$$R = \frac{4R_b}{10} = 4 \times \frac{25 \, \text{k}\Omega}{10} = 10 \, \text{k}\Omega \quad \text{(potentiometer)}$$
The CMRR, which is defined as the ratio of the differential voltage gain to the common-mode voltage gain, may also be defined as the change in input offset voltage per unit change in common-mode voltage. Let $v_{ic} = 0$ and apply $v_{id}$ to drive the output voltage to zero. $v_{id}$ should thus be equal to the input offset voltage $V_{OS}$. If we keep $v_{id}$ constant and increase $v_{ic}$ by an amount $\Delta v_{ic}$, the output voltage will change by an amount

$$\Delta v_O = A_c \Delta v_{ic}$$

(14.39)

To drive the output voltage to zero, we have to change $v_{id}$ by an amount $\Delta v_{id}$, where

$$\Delta v_{id} = \frac{\Delta v_O}{A_d} = A_c \frac{\Delta v_{ic}}{A_d} = \Delta V_{OS}$$

(14.40)

Equation (14.40) indicates that any change in $v_{ic}$ causes a corresponding change in $V_{OS}$. From Eq. (14.40), we get

$$CMRR = \frac{A_d}{A_c} = \frac{\Delta v_{ic}}{\Delta v_{id}} \bigg|_{v_o=0} = \frac{\Delta v_{ic}}{\Delta V_{OS}} \bigg|_{v_o=0} = \frac{\delta v_{ic}}{\delta V_{OS}} \bigg|_{v_o=0}$$

(14.41)

which shows that the input offset voltage is dependent on the CMRR and the common-mode signal. For CMRR $= 10^5$ (or 100 dB) and $\Delta v_{ic} = 15 \text{ V}$, the change in the input offset voltage will be $\Delta V_{OS} = 150 \mu \text{V}$.

### 14.3.9 Input Resistance

The input resistance for a MOSFET input stage is very high, in the range of $10^9 \Omega$ to $10^{12} \Omega$. For a BJT input stage, however, the input resistance is typically in the range of 100 k$\Omega$ to 1 M$\Omega$. Usually, the voltage gain is large enough that this input resistance has little effect on the circuit performance. In some differential amplifiers, a compound transistor configuration known as a Darlington pair is used to give a much higher input resistance and a much lower input biasing current than a single transistor would provide. A Darlington pair is shown in Fig. 14.13. The effective B-E voltage of the equivalent transistor $Q_T$ in Fig. 14.13(b) is

$$V_{BE} = V_{BE1} + V_{BE2} = V_T \ln \left( \frac{I_{C1}}{I_{S1}} \right) + V_T \ln \left( \frac{I_{C2}}{I_{S2}} \right)$$

$$= V_T \ln \left( \frac{I_{C1}I_{C2}}{I_{S1}I_{S2}} \right)$$

(14.42)
Since $I_C = I_{C2} = I_{C1}\beta_F\beta_F/(1 + \beta_F) \approx \beta_F I_{C1}$, Eq. (14.42) becomes

$$V_{BE} = V_T \ln \left( \frac{I_C^2}{\beta_F I_{S1} I_{S2}} \right) = 2V_T \ln \left( \frac{I_C}{\beta_F I_{S1} I_{S2}} \right)$$  \hspace{1cm} (14.43)

Solving for $I_C$, we get

$$I_C = \sqrt{\beta_F I_{S1} I_{S2}} \left( \frac{V_{BE}}{2V_T} \right)$$

$$= I_S \exp \left( \frac{V_{BE}}{V_T} \right)$$  \hspace{1cm} (14.44)

where $I_S = \sqrt{\beta_F I_{S1} I_{S2}} =$ effective saturation current and $V_T = 2V_T =$ effective thermal voltage.

The collector current $I_C$ of equivalent transistor $Q_T$ can be related to $I_{B1}$ by

$$I_C = I_{C2} = \beta_F I_{B2} = (1 + \beta_F) I_{C1} = \beta_F (1 + \beta_F) I_{B1} \approx \beta_F^2 I_{B1}$$  \hspace{1cm} (14.45)

Thus, the effective input resistance of the compound pair is given by

$$r''_\pi = \frac{V'_T}{I_{B1}} \approx \beta_F^2 \frac{2V_T}{I_C}$$  \hspace{1cm} (14.46)

which will be $2\beta_F$ times greater than that for a single device. For a single equivalent transistor, $r''_\pi = 2\beta_F r_\pi$.

Thus, if $I_C = 200$ $\mu$A, $\beta_F = 100$, and $V_T = 26$ mV,

$$r_\pi = \frac{\beta_F V_T}{I_C} = 100 \times \frac{26 \text{ mV}}{200 \text{ $\mu$A}} = 13 \text{ k}\Omega$$

for a single transistor and

$$r''_\pi = 100^2 \times 2 \times \frac{26 \text{ mV}}{200 \text{ $\mu$A}} = 2.6 \text{ M}\Omega$$

for a Darlington pair. The input offset voltage $V_{OS}$, however, will increase (generally $\sqrt{2}$ times) as a result of the increase in the effective thermal voltage.
### 14.3.10 Output Resistance

The output stage is usually an emitter follower in class AB operation, and hence it gives a low output resistance, on the order of 40 Ohms to 100 Ohms. This resistance is low enough that it does not strongly affect performance. If \( I_{\text{C(out)}} \) is the collector-biasing current of the output stage, then \( r_e \) of the output transistor becomes \( r_e = \beta_F V_T I_{\text{C(out)}} \), which, if converted to the emitter terminal by a dividing factor of \((1 + \beta_F)\), will give approximately the output resistance of the op-amp. That is,

\[
R_{\text{out}} = \frac{\beta_F V_T}{I_{\text{C(out)}}(1 + \beta_F)} \approx \frac{V_T}{I_{\text{C(out)}}}
\]

The collector-biasing current of the output stage (which is in milliamperes) is generally much greater than that of the differential stage (which is in microamperes). For example, if \( \beta_F = 100 \), \( V_T = 26 \text{ mV} \), and \( I_{\text{C(out)}} = 1 \text{ mA} \), \( R_{\text{out}} = 26 \text{ mV} \ 1 \text{ mA} = 26 \text{ }\Omega\).

### 14.3.11 Frequency Response

Because of the parasitic capacitances and the minority carrier charge storage in the devices within the op-amps, the voltage gain decreases at high frequency. The frequency at which the open-loop voltage gain falls to unity is defined as the *unity-gain bandwidth*, and it is in the range of 1 MHz to 20 MHz. The differential stage in Fig. 14.1 can be represented by a voltage-controlled current source, as shown in Fig. 14.14(a). \( C_1 \) is the effective capacitance due to the output capacitance of the differential stage and the input capacitance of the second stage. \( R_1 \) is the effective resistance due to the output resistance of the differential stage and the input resistance of the second stage. \( G_{\text{m1}} \) is the transconductance of the differential stage.

![Small-signal equivalent circuit of op-amp](a)

![Small-signal high-frequency equivalent circuit](b)

**FIGURE 14.14** High-frequency equivalent circuit of an op-amp
The second stage generally has a common-emitter or common-source configuration with an active load for a large voltage gain. It can also be represented by another voltage-controlled current source, as shown in Fig. 14.14(b). $G_{m2}$ is the transconductance of the second stage. $C_2$ and $R_2$ are the effective capacitance and the output resistance of the second stage, respectively. $A_{v2}$ is the voltage gain of the second stage and is negative—that is, $A_{v2} = -G_{m2}R_2$.

The gain of the output stage is generally unity: $A_{v3} = 1$. Since the gain of the second stage is negative, the capacitance $C_x$ will exhibit Miller’s effect and split the poles. Also, $C_x$ will influence both the frequency response and the slew rate considerably. From Eqs. (10.131) and (10.132), we can find the new poles:

$$\omega_{p1} \approx \frac{1}{G_{m2}C_xR_1R_2} \quad (14.48)$$

$$\omega_{p2} \approx \frac{G_{m2}C_x}{C_1C_2 + C_x(1 + C_2)} \quad (14.49)$$

If $C_x >> C_1$ and $C_2$, Eq. (14.49) can be approximated by

$$\omega_{p2} \approx \frac{G_{m2}C_x}{C_x(1 + C_2)} = \frac{G_{m2}}{C_1 + C_2} \quad (14.50)$$

The first pole is due to the Miller capacitance

$$C_M = C_x(1 + G_{m2}R_2) \approx C_xG_{m2}R_2 \quad (for \ G_{m2}R_2 >> 1)$$

which is much larger than $C_1$. To make $\omega_{p1}$ act as the unity-gain frequency $\omega_u$, we can select the appropriate value of $C_x$. Since the values of $C_1$ and $C_2$ are small, the second pole $\omega_{p2}$ will become very large and move to the right, provided the value of $G_{m2}$ is large enough.

**Effects of $C_x$ on Unity-Gain Bandwidth**

To determine the effect of $C_x$ on the bandwidth, let us consider Fig. 14.15, which is a simplified version of Fig. 14.14(b) in which $R_1$ and $R_2$ are treated as open circuited over the high-frequency range of interest. Under these assumptions, $\omega_{p1}$ moves to the extreme left—that is, $\omega_{p1} \approx 0$—and does not give the unity-gain bandwidth. Using KCL at the input node of Fig. 14.15, we can write the node voltages as

$$(v_o - v_{o1})(\omega C_x) = G_{m1}v_{id} + v_{o1}(\omega C_1) \quad (14.51)$$

**FIGURE 14.15** Simplified high-frequency equivalent circuit of an op-amp
where $G_{m1} = I_Q \cdot 2V_T$ is the transconductance of the differential stage. $I_Q$ is the DC biasing current of the differential amplifier shown in Fig. 14.1.

Substituting $v_o = -A_{v2}v_{o1}$ (i.e., $v_{o1} = -v_o \cdot A_{v2}$) into Eq. (14.51) and simplifying, we get the overall voltage gain $A_o$:

\[
A_o(\omega) = \frac{v_o}{v_{id}} = \frac{G_{m1}}{(1 + 1/A_{v2})(\omega C_x) + (\omega C_1)/A_{v2}}
\]

where $A_{v2}$ is the voltage gain of the second stage. For $A_{v2} >> 1$, which is usually the case, Eq. (14.53) can be simplified to

\[
A_o(\omega) = \frac{G_{m1}}{\omega C_x}
\]

(14.54)

At the unity-gain frequency $\omega_u$, $|A(\omega)| = 1$. Thus, $\omega_u$ is given by

\[
\omega_u = \frac{G_{m1}}{C_x}
\]

(14.55)

which gives the corresponding frequency (i.e., the unity-gain bandwidth) $f_u$ as

\[
f_u = \frac{G_{m1}}{2\pi C_x}
\]

(14.56)

\[
= \frac{I_Q}{4\pi V_T C_x}
\]

(14.57)

Thus, $f_u$ is directly proportional to the biasing current $I_Q$ of the differential stage and inversely proportional to the compensating capacitance $C_x$. A specific value of $C_x$ is purposely added to the circuit, either on the chip for compensated op-amps or as an external capacitor, to set the desired value of $f_u$. For example, if $I_Q = 20 \mu A$, $V_T = 26 mV$, and $C_x = 50 pF$, we get $f_u = 1.22 MHz$.

### Effects of $C_x$ on Zeros

The capacitance $C_x$ also has a Miller’s effect on the output side of the second stage, given by

\[
C_N = C_x \left(1 + \frac{1}{G_{m2}R_2}\right) \approx C_x \quad \text{(for } G_{m2}R_2 >> 1)\]

and introduces a right-half-plane zero on the transfer function. The zero $\omega_z$ can be found from Fig. 14.16 by making the output voltage $v_o \approx 0$, as shown in Fig. 14.16(a). We get

\[
G_{m2}v_{o1} = (v_{o1} - v_o)(\omega C_x) = v_{o1}(\omega C_x)
\]

which gives the zero frequency $\omega_z$ as

\[
\omega_z = \frac{G_{m2}}{C_x}
\]

(14.58)
If \( G_{m2} \) is large, which is usually the case for BJT amplifiers, then the zero will be at a very high frequency. However, if \( G_{m2} \) of the second stage is of the same magnitude as \( G_{m1} \) of the first stage, which is generally the case for CMOS amplifiers, the zero frequency will be close to the unity-gain frequency \( \omega_u \).

Since a zero introduces a phase shift, the phase margin of the amplifier will be decreased, affecting the amplifier stability. However, it is possible to remedy the stability problem by adding a resistance \( R_x \) in series with \( C_x \), as shown in Fig. 14.16(b). The combination of \( R_x \) and \( C_x \) can move the zero and make \( \omega_z \) a very large frequency. The zero frequency \( \omega_z \) can be found by making the output voltage \( v_o = 0 \); that is,

\[
G_{m2}v_{o1} = \frac{(v_{o1} - v_o)}{R_x + 1/(j\omega C_x)}
\]

which gives the new zero frequency \( \omega_z \) as

\[
\omega_z = \frac{1}{C_x(1/G_{m2} - R_x)}
\]

Thus, as \( R_x \) approaches \( 1 \), \( G_{m2} \), the zero frequency \( \omega_z \) tends to approach infinity. It is important to note that, by making \( R_x > 1 \), \( G_{m2} \), we can locate the zero frequency at the negative real axis, which will increase the phase margin.

According to Eq. (14.50), the second pole \( \omega_{p2} \) may be close to the unity-gain frequency \( \omega_u \) for a low value of \( G_{m2} \). As a result, \( \omega_{p2} \) may introduce an appreciable phase shift and thus decrease the phase margin. This problem can be resolved by increasing the value of \( C_x \) in order to split the poles further.

### 14.3.12 Slew Rate

Op-amps are limited by the *slew rate* (SR), which specifies the maximum rate at which the output voltage can change without introducing any significant amount of distortion. That is, \( SR = (dv_o/dt)_{\text{max}} \). Like the unity-gain frequency, the SR depends on the capacitance \( C_x \). Figure 14.15 illustrates that the current \( i_c \) through the capacitor is related to the output voltage \( v_o \) by

\[
i_c = C_x \frac{d}{dt} (v_o - v_{o1})
\]

\[
i_c = C_x \frac{d}{dt} \left( v_o + \frac{v_o}{A_{v2}} \right)
\]
Let us assume that $A_v2 >> 1$—that is, $v_{o1}$ tending to zero, $v_{o1} \approx 0$—and that $i_c$ can be approximated by the output current of the differential amplifier (i.e., $i_c = i_{o1} = G_{m1}v_{id}$). Then Eq. (14.61) can be approximated by

$$\frac{dv_o}{dt} \approx \frac{i_c}{C_x} \frac{i_{o1}}{C_x}$$  \hspace{1cm} (14.62)

Consider the unity-gain follower shown in Fig. 14.17(a). If a step input $v_I = V_m$ (say, 10 V) is applied to it, in zero time the output will not change, as shown in Fig. 14.17(b). Thus, $V_m$ will appear as the differential voltage between the two input terminals, and the differential stage in Fig. 14.1 will be overdriven. Transistors Q1 and Q3 in Fig. 14.1 will carry the whole biasing current $I_Q$, and transistor Q2 will be cut off. Q4, however, is the mirror of Q3, and it will produce a current of $I_Q$, which will flow into the second stage.

The biasing current limits the maximum value of the output current $i_{o1}$ of the differential stage to the value $I_Q$ in one direction and the value $-I_Q$ in the other direction; that is, $i_{o1(max)} = \pm I_Q$. Thus, the SR corresponding to $i_{o1(max)}$ is given by

$$SR = \frac{dv_o}{dt}_{max} = \frac{i_{o1(max)}}{C_x} = \pm \frac{I_Q}{C_x}$$  \hspace{1cm} (14.63)

Like the unity-gain frequency $f_u$, the slew rate is directly proportional to $I_Q$ and inversely proportional to $C_x$. If the gain of the second stage is large, then the op-amp behaves as an integrator, as shown in Fig. 14.18.

**Relation between SR and $f_u$**

Substituting $I_Q = 4\pi V_T C_x f_u$ from Eq. (14.57) into Eq. (14.63), we can relate the positive SR (known simply as SR) to $f_u$; that is,

$$SR = \frac{4\pi V_T C_x f_u}{C_x} = 4\pi V_T f_u$$  \hspace{1cm} (14.64)
Thus, there is a direct relation between the SR and the unity-gain frequency of an op-amp. For example, if $I_Q = 20 \, \mu A$, $V_T = 26 \, mV$, and $C_x = 50 \, pF$, then Eq. (14.54) gives $f_u = 1.22 \, MHz$ and Eq. (14.63) gives $SR = \frac{I_Q}{C_x} = \frac{20 \, \mu A}{50 \, pF} = 0.4 \, V/\mu s$

\[ f_u = \frac{SR}{4\pi V_T} \]  \hspace{1cm} (14.65)

Thus, there is a direct relation between the SR and the unity-gain frequency of an op-amp. For example, if $I_Q = 20 \, \mu A$, $V_T = 26 \, mV$, and $C_x = 50 \, pF$, then Eq. (14.54) gives $f_u = 1.22 \, MHz$ and Eq. (14.63) gives $SR = \frac{I_Q}{C_x} = \frac{20 \, \mu A}{50 \, pF} = 0.4 \, V/\mu s$

**KEY POINTS OF SECTION 14.3**

- The CMRR of an op-amp is very large (typically $10^5$), and the output voltage due to a common-mode signal is negligible.
- The voltage gain of an op-amp decreases with frequency; however, the gain–bandwidth product remains constant. That is, if the gain decreases, the bandwidth increases.
- The slew rate SR of an op-amp limits the maximum input frequency at which the op-amp can amplify a signal without significant distortion. For minimum distortion, the SR of the input signal should be less than that of the op-amp.
- An imperfect op-amp produces an output offset voltage caused by parameters such as $V_{io}$, $I_{io}$, $I_B$, PSRR, and thermal drift. A resistor is usually connected at the (+) terminal to minimize offset due to DC-biasing currents.
- A compensating network (either internal or external) may be connected to negate the offset voltage.
- The input offset voltage of MOSFET amplifiers is considerably higher than that of BJT amplifiers. This difference is due to mismatches in base widths for BJTs, in $I_{DSS}$ and $V_p$ for MOSFETs, and in $V_t$ and $W/L$ for MOSFETs.
- A Darlington BJT pair is commonly used to provide high input resistance, low input base current, and high current gain.
- The unity-gain frequency $f_u$ depends directly on the biasing current $I_Q$ of the differential stage and inversely on the compensation capacitance $C_x$. The slew rate SR is directly proportional to $f_u$.
- If the transconductance of the gain stage is on the same order as that of the first stage, the zero frequency will be close to the unity-gain frequency and the phase margin will be reduced. A resistor $R_x$ is usually connected in series with $C_x$ to move the zero frequency to infinity.
14.4 CMOS Op-Amps

The internal structure of operational amplifiers has, in general, three stages: a differential stage, a gain stage, and an output stage. The gain and output stages are often combined into a single stage. This type of amplifier has a very high input resistance, and the voltage gain is generally less than that of other types of comparable amplifiers. In this section, we illustrate three CMOS amplifiers: one basic type and two commercial types.

14.4.1 Basic CMOS Op-Amp

A basic CMOS amplifier with a differential stage and a gain stage is shown in Fig. 14.19 [4, 5]. PMOS transistors M6, M7, and M8 are used to generate biasing current sources. NMOS transistors M1, M2, M3, and M4 form the differential stage. NMOS M1 operates as a common-source amplifier. is used for frequency compensation by introducing a dominant pole (see Sec. 10.12). The main task in the design of CMOS amplifiers is to determine the width-to-length (W/L) ratios of the NMOS and PMOS. Usually, W/L ratios of M3, M4, and M8 are half of those for other MOSFETs. The differential pair is the “heart” of op-amps and can be formed by cascoding MOSFETs to yield high gain and low CMRR (see Sec. 9.2.1).

Using Eq. (9.61), the voltage gain of the differential stage is given by

\[ A_1 = -(g_{m2} + g_{m4}) \frac{r_{o2}}{r_{o4}} \]

and the voltage gain of the gain (second) stage is given by

\[ A_2 = -g_{m5} \frac{r_{o5}}{r_{o6}} \]

Therefore, the overall low-frequency voltage gain of the amplifier is

\[ A_o = A_1A_2 = (g_{m2} + g_{m4}) \frac{r_{o2}}{r_{o4}} g_{m5} \frac{r_{o5}}{r_{o6}} \]  \hspace{1cm} (14.66)

\[ \text{FIGURE 14.19} \quad \text{Basic CMOS amplifier} \]
14.4.2 CMOS Op-Amp MC14573

The simplified schematic of op-amp MC14573 is shown in Fig. 14.20. Its internal structure can be divided into a differential CMOS stage, a gain stage, and a biasing circuit. This op-amp has a very high input resistance, but the gain is lower than that of other op-amps, as expected. Some parameters of the MC14573 op-amp are listed in Table 14.1.

### TABLE 14.1 Parameters of op-amp MC14573

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Minimum</th>
<th>Typical</th>
<th>Maximum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC supply voltages $V_{DD}$</td>
<td></td>
<td></td>
<td>±18 V</td>
<td>V</td>
</tr>
<tr>
<td>Input offset voltage $V_{OS}$</td>
<td></td>
<td>160</td>
<td>600</td>
<td>μV</td>
</tr>
<tr>
<td>Thermal drift $D_v$</td>
<td></td>
<td>1</td>
<td>1.0</td>
<td>μV °C</td>
</tr>
<tr>
<td>Input biasing current $I_B$</td>
<td></td>
<td></td>
<td>200 nA</td>
<td>nA</td>
</tr>
<tr>
<td>Differential input resistance $R_{id}$</td>
<td></td>
<td>$10^{12}$</td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Common-mode input resistance $R_{ic}$</td>
<td></td>
<td>$10^{12}$</td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Output resistance $R_o$</td>
<td></td>
<td>50</td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Input capacitance $C_i$</td>
<td></td>
<td>1.0</td>
<td></td>
<td>pF</td>
</tr>
<tr>
<td>Open-loop voltage gain $A_v$</td>
<td></td>
<td>90</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Common-mode rejection ratio (CMRR)</td>
<td></td>
<td>75</td>
<td>95</td>
<td>dB</td>
</tr>
<tr>
<td>Unity-gain bandwidth $f_u$</td>
<td></td>
<td>70</td>
<td></td>
<td>MHz</td>
</tr>
<tr>
<td>Slew rate (SR)</td>
<td></td>
<td>2.5</td>
<td></td>
<td>V μs</td>
</tr>
<tr>
<td>Power supply rejection ratio (PSRR)</td>
<td></td>
<td>75</td>
<td>97</td>
<td>dB</td>
</tr>
</tbody>
</table>

**FIGURE 14.20** Schematic for op-amp MC14573 (Copyright of Motorola. Used by permission.)
Differential Stage

The input stage is a PMOS differential amplifier consisting of transistors Q1 and Q2. Its load is a current mirror consisting of NMOS transistors Q3 and Q4.

Gain Stage

The gain stage is also the output stage. Q7 is operated as a common-source amplifier with Q8 as the active load. Cx is the compensation capacitor connected to Q7.

DC Biasing

The differential stage is biased by transistors Q5 and Q6 as a current source. An external resistance $R_{\text{ref}}$ sets the DC biasing current $I_Q$, which can be found approximately from

$$I_Q = I_{\text{ref}} = \frac{V_{DD} + V_{SS} - V_{SG}}{R_{\text{ref}}}$$

The value of $V_{GS5}$ can be solved for known values of $V_{DD}$, $V_{SS}$, $V_t$, and $K_p$. Transistor Q8 serves as the load to the output stage. Since the gate voltages of Q5, Q6, and Q8 are the same, the biasing drain current Q8 equals $I_Q = I_Q$. 

EXAMPLE 14.4

Analyzing the CMOS op-amp MC14573  

The CMOS amplifier in Fig. 14.20 is operated at a biasing current of $I_Q = 40 \mu A$. The parameters of the MOSFETs are $K_x = 10 \mu A \ V^2$, $|V_{M(NMOS)}| = |V_{M(PMOS)}| = 70 \ V$, $V_t = 0.5 \ V$, and $W/L = 160 \ \mu m \ 10 \ \mu m$, except for Q7, for which $W/L = 320 \ \mu m \ 10 \ \mu m$. Assume $V_{DD} = V_{SS} = 5 \ V$.

(a) Find $V_{GS}$, $g_m$, and $r_o$ for all MOSFETs.

(b) Find the low-frequency voltage gain of the amplifier $A_{vo}$.

(c) Find the value of the external resistance $R_{\text{ref}}$.

(d) Find the value of compensation capacitance $C_x$ that gives a unity-gain bandwidth of 1 MHz and the corresponding slew rate.

(e) Find the value of resistance $R_x$ to be connected in series with $C_x$ in order to move the zero frequency to infinity.

(f) Find the common-mode input voltage range.

(g) Find the output voltage range.

SOLUTION

(a) $K_p$, $V_{GS}$, $g_m$, and $r_o$ are calculated using the following equations:

$$K_p = \frac{K_x W}{L}$$

(14.68)
Their values are shown in Table 14.2.

(b) Using Eq. (7.34) the voltage gain of the first differential stage with the active load is given by

$$A_{v1} = -g_{m1}(ro7 || ro8)$$

$$= -113.1 \mu \times (3.5 \Omega \parallel 3.5 \Omega) = -198$$

The voltage gain of the second stage with the active load is given by

$$A_{v2} = -g_{m2}(ro7 || ro8)$$

$$= -226.3 \mu \times (1.75 \Omega \parallel 1.75 \Omega) = -198$$

Thus, the overall voltage gain is $A_{vo} = A_{v1}A_{v2} = 198 \times 198 = 39,204$ (or 91.87 dB).

(c) From Eq. (14.67), we get the value of the external resistance $R_{ref}$ as

$$R_{ref} = \frac{V_{DD} - V_{SS} - V_{GS}}{I_Q} = \frac{(5 + 5 - 1) V}{40 \mu A} = 225 \, \text{k}\Omega$$

(d) For $f_u = 1 \text{ MHz}$, Eq. (14.56) gives the value of compensation capacitance $C_x$ as

$$C_x = \frac{G_{m1}}{2\pi f_u} = \frac{G_{m2}}{2\pi f_u} = \frac{113.1 \mu A/V}{2\pi \times 1 \text{ MHz}} = 18 \text{ pF}$$

From Eq. (14.63), we get the slew rate as

$$SR = \frac{I_Q}{C_x} = \frac{40 \mu A}{18 \text{ pF}} = 2.22 \text{ V/\mu s}$$

(e) From Eq. (14.59), the value of resistance $R_x$ for $\omega_x = 0$ is

$$R_x = \frac{1}{G_{m2}} = \frac{1}{g_{m7}} = \frac{1}{226.3 \mu A/V} = 4.42 \, \text{k}\Omega$$
14.4.3 CMOS Op-Amp TLC1078

The simplified schematic of the TLC1078 op-amp is shown in Fig. 14.21. Its structure is similar to that of op-amp MC14573 in Fig. 14.20. The internal structure can be divided into a differential CMOS stage, a gain stage, an output stage, and a biasing circuit. Some parameters of the TLC1078 op-amp are listed in Table 14.3.

TABLE 14.3 Parameters of op-amp TLC1078

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Minimum</th>
<th>Typical</th>
<th>Maximum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC supply voltages ( V_{DD} )</td>
<td>18</td>
<td></td>
<td></td>
<td>V</td>
</tr>
<tr>
<td>Input offset voltage ( V_{OS} )</td>
<td>180</td>
<td>600</td>
<td></td>
<td>( \mu \text{V} )</td>
</tr>
<tr>
<td>Thermal drift ( D_v )</td>
<td>1</td>
<td></td>
<td></td>
<td>( \mu \text{V/}^{\circ}\text{C} )</td>
</tr>
<tr>
<td>Input biasing current ( I_B )</td>
<td>0.7</td>
<td></td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Input offset current ( I_{OS} )</td>
<td>0.1</td>
<td></td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Differential input resistance ( R_{id} )</td>
<td>1.5</td>
<td></td>
<td></td>
<td>T( \Omega )</td>
</tr>
<tr>
<td>Common-mode input resistance ( R_{ic} )</td>
<td>1.5</td>
<td></td>
<td></td>
<td>T( \Omega )</td>
</tr>
<tr>
<td>Output resistance ( R_o )</td>
<td>50</td>
<td></td>
<td></td>
<td>( \Omega )</td>
</tr>
<tr>
<td>Input capacitance ( C_i )</td>
<td>1.0</td>
<td></td>
<td></td>
<td>( \text{pF} )</td>
</tr>
<tr>
<td>Open-loop voltage gain ( A_o )</td>
<td>110</td>
<td>120</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Common-mode rejection ratio (CMRR)</td>
<td>75</td>
<td>97</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Unity-gain bandwidth ( f_u )</td>
<td>110</td>
<td></td>
<td></td>
<td>MHz</td>
</tr>
<tr>
<td>Slew rate (SR)</td>
<td>47</td>
<td></td>
<td></td>
<td>V ms</td>
</tr>
<tr>
<td>Power supply rejection ratio (PSRR)</td>
<td>75</td>
<td>97</td>
<td></td>
<td>dB</td>
</tr>
</tbody>
</table>
Differential Stage

The input stage is a PMOS differential amplifier consisting of transistors Q1 and Q5. Its load is a current mirror consisting of NMOS transistors Q2 and Q4, which have sources $R_2$ and $R_3$ to give high output resistances.

Gain Stage

The gain stage uses NMOS Q7 in a common-source configuration with PMOS Q6 as the current source active load. The combination of $C_x$ and $R_5$ is a pole-zero circuit, which can reduce the lowest pole or break frequency to a low value to ensure stability. Also, it can produce a zero to cancel out one of the second poles of the amplifier’s open-loop frequency response.

Output Stage

The output stage consists of a pair of NMOS transistors that operate in the class AB push-pull mode. Transistor Q8 is the source follower, sourcing current to the load during the interval when the signal at the output of Q7 goes up above the quiescent value. Transistor Q9 acts as the common-source amplifier, sinking current from the load during the interval when the signal at the output of the differential stage goes down below the quiescent value.

A decreasing signal at the output of the differential stage will be amplified by Q7 with a phase shift of $180^\circ$ and then will appear through the source follower of Q8 as an increasing signal to the load. However, an increasing signal will be amplified by the common-source amplifier of Q9 with a phase shift of $180^\circ$ and then will appear as a decreasing signal to the load. Thus, the voltages applied to the gates of Q8 and Q9 will be phase shifted by $180^\circ$. The voltage gain through the two paths will be the same, however.
Operational Amplifiers

DC Biasing

The differential amplifier is biased by transistors Q3 and Q6 as a current source whose gate voltage is set by the voltage divider circuit consisting of transistors Q10 through Q13. All transistors are identical, except for Q10 and Q13, whose channel widths differ from that of the others.

EXAMPLE 14.5

Analyzing the CMOS op-amp TLC1078  The CMOS amplifier in Fig. 14.21 is operated at a biasing current of $I_Q = 40 \mu A$. The parameters of the MOSFETs are $K_x = 10 \mu A/V^2$, $|V_{M(NMOS)}| = V_{M(PMOS)} = 70 V$, $V_t = 0.5 V$, and $W/L = 160 \mu m/10 \mu m$, except for Q10, for which $W/L = 40 \mu m/10 \mu m$. Find the value of the resistance $R_7$. Assume $V_{DD} = V_{SS} = 5 V$.

SOLUTION

We have

$$K_p = \frac{K_xW}{L} = 10 \mu A \times \frac{160}{10} = 160 \mu A/V^2 \quad \text{(for all MOSFETs expect Q10)}$$

$$K_{p10} = \frac{K_xW}{L} = 10 \mu A \times \frac{40}{10} = 40 \mu A/V^2 \quad \text{(for Q10)}$$

For $I_Q = 40 \mu A$ and $K_p = 160 \mu A/V^2$, Eq. (14.69) gives

$$V_{GS} = V_t + \sqrt{I_D/K_p} = 0.5 + \sqrt{40/160} = 1 V$$

Thus,

$$V_{GS3} = V_{GS6} = V_{GS11} = V_{GS12} = V_{GS13} = 1 V$$

Also,

$$I_{D3} = I_{D6} = I_{D11} = I_{D12} = I_{D10} = I_{D13} = I_Q$$

The drain current $I_{D13}$ of Q13 is given by

$$I_{D13} = K_{p13}(V_{GS13} - V_t)^2 \quad (14.74)$$

Since $V_{GS10} = V_{GS13} - I_QR_7$, the drain current $I_{D10}$ of Q10 is given by

$$I_{D10} = K_{p10}(V_{GS10} - V_t)^2 = K_{p10}(V_{GS13} - I_QR_7 - V_t)^2 \quad (14.75)$$

The $V_{GS}$ values of Q10 and Q13 are different, but their drain currents are equal; that is,

$$I_{D13} = I_{D10} \quad \text{and} \quad K_{p13}(V_{GS13} - V_t)^2 = K_{p10}(V_{GS13} - I_QR_7 - V_t)^2$$
Like MOSFET op-amps, BJT amplifiers have three stages: a differential stage, a gain stage, and an output stage. They have the disadvantages of a lower input resistance and a higher input biasing current. However, BJTs provide higher voltage gain [6–8]. In this section, we consider two popular op-amps.

14.5.1 BJT Op-Amp LM124

The LM124 op-amp is designed to operate from a power supply of ±16 V, but it can operate from a single power supply as low as 5 V. The simplified schematic is shown in Fig. 14.22. Some parameters of the LM124 op-amp are listed in Table 14.4.

Differential Stage

The differential input stage consists of bipolar transistors Q1 through Q4. They are connected in Darlington pairs to give high input resistances and low input biasing currents. These BJTs drive the current mirror load consisting of transistors Q8 and Q9 for a high differential gain.
Gain Stage

The gain stage is a common-emitter amplifier consisting of pnp transistor Q10 (with an active load), which is followed by a Darlington pair consisting of transistors Q11 and Q12. The beta of a pnp transistor is generally low. The Darlington pair offers a high load to transistor Q10. This arrangement ensures a high voltage gain (100 dB). The number of poles in the op-amp increases with the number of transistor stages. $C_x$ is used for feedback compensation.

### TABLE 14.4 Parameters of op-amp LM124

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Minimum</th>
<th>Typical</th>
<th>Maximum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC supply voltages $V_S$</td>
<td></td>
<td>±16</td>
<td></td>
<td>V</td>
</tr>
<tr>
<td>Input offset voltage $V_{OS}$</td>
<td>1.0</td>
<td>2.0</td>
<td></td>
<td>mV</td>
</tr>
<tr>
<td>Thermal drift $D_v$</td>
<td>100</td>
<td>200</td>
<td></td>
<td>μV °C</td>
</tr>
<tr>
<td>Input biasing current $I_B$</td>
<td>20</td>
<td>50</td>
<td></td>
<td>nA</td>
</tr>
<tr>
<td>Input offset current $I_{OS}$</td>
<td>2.0</td>
<td>10</td>
<td></td>
<td>nA</td>
</tr>
<tr>
<td>Differential input resistance $R_{id}$</td>
<td>2</td>
<td></td>
<td></td>
<td>MΩ</td>
</tr>
<tr>
<td>Common-mode input resistance $R_{ic}$</td>
<td>2</td>
<td></td>
<td></td>
<td>MΩ</td>
</tr>
<tr>
<td>Output resistance $R_o$</td>
<td>75</td>
<td></td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Input capacitance $C_i$</td>
<td>4.0</td>
<td></td>
<td></td>
<td>pF</td>
</tr>
<tr>
<td>Open-loop voltage gain $A_o$</td>
<td>50</td>
<td>100</td>
<td></td>
<td>V mV</td>
</tr>
<tr>
<td>Common-mode rejection ratio (CMRR)</td>
<td>70</td>
<td>85</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Unity-gain bandwidth $f_o$</td>
<td>1.0</td>
<td></td>
<td></td>
<td>MHz</td>
</tr>
<tr>
<td>Slew rate (SR)</td>
<td>3.0</td>
<td></td>
<td></td>
<td>V μs</td>
</tr>
<tr>
<td>Power supply rejection ratio (PSRR)</td>
<td>650</td>
<td>100</td>
<td></td>
<td>dB</td>
</tr>
</tbody>
</table>
Output Stage

The Darlington emitter follower consisting of transistors Q5 and Q6 offers a lower-than-average output resistance and a high resistance to the gain stage. If the voltage at the collector of transistor Q12 goes up in the positive direction, transistor Q6 will drive the load and source current of I_Q3 (50 μA). However, if the current of Q6 falls below the level of I_Q3 (50 μA), transistor Q6 will be off, and the voltage across the load will go down below the quiescent level. This will cause transistor Q13 to turn on and sink the load current. Transistor Q7, together with resistor R_SC, provides short-circuit protection by limiting the current through Q6. This is done by turning Q7 on if the voltage across R_SC exceeds the base-emitter voltage V_BE7.

14.5.2 BJT Op-Amp LM741

The LM741 op-amp is familiar to students because its characteristics are commonly used in illustrating the applications of op-amps. This op-amp was first introduced in 1966 by Fairchild Semiconductor, Inc. It is relatively simple. It has a large voltage gain and a large CMRR. The range of common-mode and differential input voltages is wide. The schematic is shown in Fig. 14.23. The op-amp circuit can be divided into five parts: a DC-biasing circuit, an input stage, an amplifier stage, an output stage, and overload protection. Some parameters of the LM741 op-amp are listed in Table 14.5.

Differential Stage

The differential input stage consists of bipolar transistors Q1 through Q4. They form common-emitter and common-base configurations (see Fig. 9.42) for higher bandwidth and gain. This stage is biased by the current source consisting of Q8, and it drives the current mirror active load consisting of Q5, Q6, and Q7. The biasing feedback loop formed by Q8 and Q9 stabilizes the biasing currents in each of the input transistors at approximately one-half of the collector current of Q10.

### TABLE 14.5 Parameters of op-amp LM741

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Minimum</th>
<th>Typical</th>
<th>Maximum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC supply voltages V_CC</td>
<td></td>
<td>±22</td>
<td></td>
<td>V</td>
</tr>
<tr>
<td>Input offset voltage V_OS</td>
<td></td>
<td>1.0</td>
<td>5.0</td>
<td>mV</td>
</tr>
<tr>
<td>Thermal drift D_v</td>
<td></td>
<td>15</td>
<td></td>
<td>μV °C</td>
</tr>
<tr>
<td>Input biasing current I_B</td>
<td></td>
<td>80</td>
<td>550</td>
<td>nA</td>
</tr>
<tr>
<td>Input offset current I_QS</td>
<td></td>
<td>20</td>
<td>200</td>
<td>nA</td>
</tr>
<tr>
<td>Differential input resistance R_id</td>
<td>0.3</td>
<td>2</td>
<td></td>
<td>MΩ</td>
</tr>
<tr>
<td>Common-mode input resistance R_ic</td>
<td></td>
<td>2</td>
<td></td>
<td>MΩ</td>
</tr>
<tr>
<td>Output resistance R_o</td>
<td></td>
<td>75</td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Input capacitance C_i</td>
<td></td>
<td>4.0</td>
<td></td>
<td>pF</td>
</tr>
<tr>
<td>Open-loop voltage gain A_o</td>
<td>50</td>
<td>200</td>
<td></td>
<td>V mV</td>
</tr>
<tr>
<td>Common-mode rejection ratio (CMRR)</td>
<td>70</td>
<td>90</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Unity-gain bandwidth f_u</td>
<td></td>
<td>1.0</td>
<td></td>
<td>MHz</td>
</tr>
<tr>
<td>Slew rate (SR)</td>
<td></td>
<td>0.5</td>
<td></td>
<td>V μs</td>
</tr>
<tr>
<td>Power supply rejection ratio (PSRR)</td>
<td>70</td>
<td>90</td>
<td></td>
<td>dB</td>
</tr>
</tbody>
</table>
Operational Amplifiers

FIGURE 14.23 Schematic for op-amp LM741 (Courtesy of National Semiconductor, Inc.)
Gain Stage

The gain stage consists of a common-emitter Darlington pair amplifier made up of transistors Q16 and Q17. Transistor Q17 drives an active load consisting of transistor Q13B for high gain, and it is followed by the pnp transistor Q23 in common-emitter configuration. The extra emitter on Q23 prevents Q17 from saturating by diverting the base drive current from Q16 when $V_{CB}$ of Q17 reaches zero volts. This arrangement eliminates the possibility of a high current condition—that is, a high base voltage of Q16 and a high collector voltage of Q7—that could damage Q16. Q23B acts as the DC feedback so that $V_{B16} - V_{C17} = V_{EB23B}$. $C_x$ is used for frequency compensation.

Output Stage

The output stage is a push-pull output stage and operates in class AB mode to reduce crossover distortion. It is biased by a $V_{BE}$ multiplier circuit (see Fig. 14.24) consisting of transistors Q18 and Q19.

Protection Circuitry

Resistors R6 and R7 provide short-circuit protection, turning on Q15 and Q21 to limit the currents through Q14 and Q20, respectively. Turning on Q21 also turns on transistors Q22 and Q24, thereby shorting the input signal to the base of Q16 in the gain stage.

KEY POINT OF SECTION 14.5

- BJT amplifiers usually have three stages: a differential stage, a gain stage, and an output stage. They have the disadvantages of a low input offset voltage (1 mV), a low input resistance (2 MΩ), and a high input biasing current (60 nA), but they provide large gain (100 dB) and wide bandwidth (1 MHz).

14.6 Analysis of the LM741 Op-Amp

As an example, let us carry out a complete analysis of the LM741 op-amp shown in Fig. 14.23, finding the DC biasing currents, the small-signal gain, the input and output resistances, and the unity-gain bandwidth. The analysis can be divided into three sections: DC analysis, AC analysis, and frequency-response analysis.

14.6.1 DC Analysis

The DC analysis to determine the quiescent operating currents and voltages of the transistors can be simplified by making the following assumptions:

1. The output resistances of the transistors are very high and do not affect the currents flowing in the circuit. Usually, this assumption results in a 10% to 20% error in the calculated currents.
2. The output voltage is maintained at a constant specified value by the internal feedback loop. Let us assume that the output voltage is zero. This assumption is necessary because of the very high gain,
typically $10^5$, which results in a very low input voltage. If the output voltage is calculated with two input terminals grounded, any change in the beta or output resistances could cause a large change in the output voltage, and the transistors could be operating in the saturation region rather than in the active region as expected.

3. The npn transistors have large betas; let us assume $\beta_{F_{\text{npn}}}=250$.
4. The betas of pnp transistors are much lower than those of npn transistors; let us assume $\beta_{F_{\text{pnp}}}=50$. Assume $V_T=26$ mV for all transistors.

**Biasing Circuit**

The currents in the biasing current sources of $Q_{10}$ and $Q_{13\text{AB}}$ can be calculated from Fig. 14.24. Let us assume that all transistors are operating in the forward-active region, the base currents are negligible, $V_T=26$ mV, and $V_{BE}=0.7$ V. The reference current can be calculated as follows:

$$I_{\text{ref}} = \frac{V_C - V_E - V_{BE11} - V_{EB12}}{R_5}$$

$$= \frac{(15 + 15 - 2 \times 0.7) \text{V}}{39 \text{ k}\Omega} = 0.73 \text{ mA}$$

The transistors $Q_{10}$ and $Q_{11}$ form a Widlar current source. Using Eq. (9.98), we can find the output current $I_1$ from

$$V_T \ln \left( \frac{I_{\text{ref}}}{I_1} \right) = R_4 I_1$$

$$V_T \ln \left( \frac{0.73 \text{ mA}}{I_1} \right) = 5 \text{ k}\Omega \times I_1$$

which, by trial-and-error solution, gives $I_1 = 19 \mu$A.
Transistor Q13 is a multicollector lateral pnp device. Thus, currents $I_2$ and $I_3$ are three-fourths and one-fourth of the reference current $I_{\text{ref}}$, respectively:

\[
I_2 = \left(\frac{3}{4}\right) \times 0.73 \text{ mA} = 0.55 \text{ mA}
\]
\[
I_3 = \left(\frac{1}{4}\right) \times 0.73 \text{ mA} = 0.18 \text{ mA}
\]

If we replace the biasing circuit in Fig. 14.24 by the equivalent current sources of $I_1$, $I_2$, and $I_3$, the circuit in Fig. 14.23 can be simplified to that in Fig. 14.25.

### Input Stage

The input stage provides a high input resistance for differential and common-mode input signals, the differential-to-single output, and some voltage gain. The input stage of the LM741 with biasing current sources is shown in Fig. 14.26. Since the npn transistors have large values of beta, the base currents are negligible compared to the collector currents. For identical transistors, $I_{C9} = I_{C8}$.

The current $I_T$, which is the sum of $I_{C8}$ plus the base currents of $Q_8$ and $Q_9$, can be found from

\[
I_T = I_{C8} + I_{B8} + I_{B9} = I_{C9} \left(1 + \frac{2}{\beta_{F(pnp)}}\right)
\]

(14.80)
where $\beta_{F(pnp)}$ is the current gain for $pnp$ transistors. Neglecting the base currents of $Q_1$ and $Q_2$, we find that the emitter currents of $Q_3$ and $Q_4$ are

$$I_{E3} = I_{E4} = \frac{I_T}{2} = \frac{I_{C9}}{2} \left(1 + \frac{2}{\beta_{F(pnp)}}\right)$$

(14.81)

The sum of the base currents of $Q_3$ and $Q_4$ and the collector current of $Q_9$ must be equal to the biasing current of $I_1 = 19 \mu A$. Thus,

$$I_1 = I_{C9} + \frac{I_{E3}}{1 + \beta_{F(pnp)}} + \frac{I_{E4}}{1 + \beta_{F(pnp)}} = I_{C9} \left[1 + \frac{1 + 2/\beta_{F(pnp)}}{1 + \beta_{F(pnp)}}\right]$$

(14.82)

which can be simplified to $I_1 = I_{C9}[(2 + \beta_{F(pnp)})/(1 + \beta_{F(pnp)})] \approx I_{C9}$. Substituting $I_{C9}$ from Eq. (14.82) into Eq. (14.80) yields

$$I_T \approx I_1 \left(1 + \frac{2}{\beta_{F(pnp)}}\right)$$

(14.83)

$$\approx (19 \mu A) \left(1 + \frac{2}{50}\right) = 19.38 \mu A$$
Then

\[ I_{C1} = I_{C2} = \frac{19.38 \, \mu A}{2} = 9.68 \, \mu A \]

\[ I_{C3} = I_{C4} = \frac{\beta_{F(pnp)} I_{E3}}{1 + \beta_{F(pnp)}} = \frac{50}{1 + 50} \times 9.69 \, \mu A = 9.5 \, \mu A \]

\[ I_{C5} = I_{C6} \approx I_{C3} = 9.5 \, \mu A \]

Because of the biasing feedback loop formed by Q8 and Q9, the biasing currents in each of the input transistors Q1 and Q2 are approximately one-half of the collector current of Q10, or \( I_{C1} \approx I_{C2} \).

The emitter current \( I_{E7} \) of Q7 is the sum of the base currents of Q5 and Q6 plus the current into resistor R3:

\[ I_{E7} = I_{B5} + I_{B6} + I_{R3} \approx I_{R3} \]

Assuming \( I_{E5} = I_{C5} \) and \( I_{E6} = I_{C6} \), the voltage across R3 is

\[ V_{R3} = V_{BE5} + R_1 I_{E5} = V_{BE6} + R_2 I_{E6} \]

\[ = V_T \ln \left( \frac{I_{C5}}{I_S} \right) + R_1 I_{E5} \]  

(14.84)

\[ = 26 \text{ mV} \times \ln \left( \frac{9.5 \, \mu A}{10^{-14} \text{ A}} \right) + 1 \text{ k}\Omega \times 9.5 \, \mu A = 537.5 \text{ mV} + 9.5 \text{ mV} = 547 \text{ mV} \]

The collector current of Q7 is

\[ I_{C7} \approx I_{E7} = \frac{V_{R3}}{R_3} = \frac{547 \text{ mV}}{50 \text{ k}\Omega} = 10.9 \, \mu A \]

**Gain Stage**

The gain stage provides a high voltage gain with a very high input and output resistance. The amplifier stage is shown in Fig. 14.27. Since it is assumed that the output voltage of the amplifier is zero, the base current of Q23 is zero and the collector current of Q17 is \( I_{C17} = I_2 = 550 \, \mu A \). The voltage at the base of Q17

![FIGURE 14.27 Gain stage](image)
is equal to the base–emitter voltage of Q17 plus the voltage drop across resistor R8. Assuming $I_{E17} = I_{C17}$ and $I_S = 10^{-14}$ A, the voltage at the base of Q17 with respect to $-V_{EE}$ is

$$V_{B17} = V_{BE17} + R_8 I_{E17}$$

$$= V_T \ln \left( \frac{I_{C17}}{I_S} \right) + R_8 I_{E17}$$

$$= V_T \ln \left( \frac{550 \ \mu A}{10^{-14} \ \text{A}} \right) + 100 \ \Omega \times 550 \ \mu A = 643 \text{ mV} + 55 \text{ mV} = 698 \text{ mV}$$

The current through $R_9$ is

$$I_{R9} = \frac{V_{B17}}{R_9} = \frac{698 \text{ mV}}{50 \ \text{k} \Omega} = 13.96 \ \mu A$$

The base current of $Q_{B17}$ is

$$I_{B17} = \frac{I_{C17}}{\beta_{F(npn)}} = \frac{550 \ \mu A}{250} = 2.2 \ \mu A$$

The collector current of $Q_{16}$ is

$$I_{C16} \approx I_{E16} = I_{B17} + I_{R9} = 2.2 \ \mu A + 13.94 \ \mu A = 16.14 \ \mu A$$

**Output Stage**

The output stage supplies a high load current and offers a low output resistance. This stage is shown in Fig. 14.28. Assuming that the base currents are negligible, the collector current of $Q_{23}$ is $I_{C23} = I_3 = 180 \ \mu A$. Further assuming that the circuit is connected with feedback in such a way that the output

![FIGURE 14.28 Output stage](image_url)
voltage is driven to zero and the output current is also zero, \( I_{C14} \) and \( I_{C20} \) are approximately equal in magnitude. Thus,

\[
I_{C14} = I_{C20} \tag{14.86}
\]

Let us estimate that \( V_{BE18} = 0.6 \text{ V} \) and the collector current of \( Q_{19} \) is given approximately by

\[
I_{C19} \approx \frac{V_{BE18}}{R_{10}} = \frac{0.6}{40 \text{ k}\Omega} = 15 \text{ \( \mu \)A}
\]

Then

\[
I_{C18} = I_{C23} - I_{C19} = 180 \text{ \( \mu \)A} - 15 \text{ \( \mu \)A} = 165 \text{ \( \mu \)A}
\]

Now we can calculate a more accurate value for \( V_{BE18} \):

\[
V_{BE18} = V_T \ln \left( \frac{I_{C18}}{I_S} \right) = (26 \text{ mV}) \ln \left( \frac{165 \text{ \( \mu \)A}}{10^{-14} \text{ A}} \right) = 611.7 \text{ mV}
\]

The collector current of \( Q_{19} \) is

\[
I_{C19} = \frac{I_{C18}}{\beta_{F(npn)}} + \frac{V_{BE18}}{R_{10}} = \frac{165 \text{ \( \mu \)A}}{250} + \frac{611.7 \text{ mV}}{40 \text{ k}\Omega} = 16 \text{ \( \mu \)A}
\]

and

\[
I_{C18} = I_3 - I_{C19} = 180 \text{ \( \mu \)V} - 16 \text{ \( \mu \)V} = 164 \text{ \( \mu \)A}
\]

which is very close to the original estimate of 165 \( \mu \)A. If there had been a significant difference between this value and the original estimate, we would have used this value to find new values of \( V_{BE18} \), \( I_{C19} \), and \( I_{C18} \), continuing the iterations until the desired value was found.

Using KVL around the loop formed by \( Q_{14}, Q_{20}, Q_{18}, \) and \( Q_{19} \) in Fig. 14.28, we get

\[
V_{BE18} + V_{BE19} = V_{BE20} + |V_{BE20}|
\]

which can be written as

\[
V_T \ln \left( \frac{I_{C18}}{I_{S18}} \right) + V_T \ln \left( \frac{I_{C19}}{I_{S19}} \right) = V_T \ln \left( \frac{I_{C14}}{I_{S14}} \right) + V_T \ln \left( \frac{I_{C20}}{I_{S20}} \right) \tag{14.87}
\]

For an output voltage of \( v_O = 0 \) and \( \beta_{F(npn)} >> 1 \),

\[
|I_{C14}| = |I_{C20}| \tag{14.88}
\]

and Eq. (14.87) can be simplified to

\[
\frac{I_{C18}I_{C19}}{I_{S18}I_{S19}} = \frac{I_{C14}^2}{I_{S14}I_{S20}} \tag{14.89}
\]

from which we get

\[
I_{C14} = I_{C20} = \sqrt{I_{C18}I_{C19}} \sqrt{\frac{I_{S14}I_{S20}}{I_{S18}I_{S19}}} \tag{14.90}
\]
Thus, the collector currents depend on the $I_S$ values, which depend on the physical geometry of the transistors. $Q_{14}$ and $Q_{20}$ are normally designed to carry much larger current than other transistors. The specific geometries used by different manufacturers may be different, but the $I_S$ value of $Q_{14}$ and $Q_{20}$ is typically three times that of $Q_{18}$ and $Q_{19}$. Thus,

$$I_{S14} = I_{S20} \approx 3I_{S18} = 3I_{S19} \quad (14.91)$$

Substituting $I_{S14}$ from Eq. (14.91) into Eq. (14.90) yields

$$I_{C14} = I_{C20} = 3\sqrt{I_{C18}I_{C19}}$$
$$= 3 \times \sqrt{(164 \, \mu A)(16 \, \mu A)} = 153.7 \, \mu A \quad (14.92)$$

---

**Overload Protection**

Transistor $Q_{15}$ (in Fig. 14.23) turns on only when the voltage across $R_6$ exceeds 550 mV at an output-sourcing current of 550 mV $R_6 = 550 \, \text{mV} \, 27 = 20 \, \text{mA}$. When $Q_{15}$ turns on, it limits the current to the base of $Q_{14}$ and the output current cannot increase further. Thus, $Q_{15}$ provides short-circuit protection, preventing damage to the op-amp due to excess current flow and power dissipation. These problems can occur if the output is shorted to a negative power supply. Similarly, transistors $Q_{21}$, $Q_{22}$, and $Q_{24}$ protect transistor $Q_{20}$ in the case of sinking current. When $Q_{21}$ turns on, it protects $Q_{20}$ by limiting the current to the base of $Q_{20}$, thereby turning on $Q_{22}$ and $Q_{24}$.

If the inverting terminal were overdriven such that its voltage became more positive than that of the noninverting terminal, $Q_1$ would turn off. As a result, $Q_6$ would also be off, and the current into the base of $Q_{16}$ would be $I_T = 19.4 \, \mu A$. This current would be amplified by the beta of $Q_{16}$, which could be as high as 1000, giving a collector current of $I_{C16} = 1000 \times 19.4 \, \mu A \approx 19.4 \, \text{mA}$ that would flow into the base of $Q_{17}$. $Q_{17}$ would thus become saturated. Saturation would result in a power dissipation in $Q_{16}$ of

$$I_{C16}(V_{CC} + V_{EE}) = 19.4 \, \text{mA} \times (15 + 15) \, \text{V} \approx 580 \, \text{mW}$$

The extra emitter on $Q_{23}$ prevents $Q_{17}$ from developing a high current condition that could damage $Q_{16}$.

---

**14.6.2 Small-Signal AC Analysis**

Small-signal analysis is performed to determine the input resistance, the output resistance, the transconductance, and the voltage gain. The op-amp circuit may be broken up into three stages: the input stage, the gain stage, and the output stage. Since the op-amp may be considered as three cascaded stages, we will represent the input and gain stages by their transconductance equivalents to simplify the analysis to determine the overall gain.

Notice from Eq. (14.41) that the CMRR is the change in the common-mode voltage per unit change in the input offset voltage $AV_{OS}$, which can be determined from Eq. (14.40) if $A_d$ is known. Thus, we need to determine only $A_d$. The small-signal parameters of the transistors are

$$r_{n16} = \frac{\beta_{F16} V_T}{I_{C16}} = \frac{250 \times 26 \, \text{mV}}{16.1 \, \mu A} = 403.7 \, \text{k}\Omega$$
The output resistance \( r_0 \) is given by

\[
r_0 = \frac{V_A}{I_C} = \frac{V_A}{V_T g_m} = \frac{1}{\eta_n g_m}
\]

(14.93)

where \( \eta_n = V_T / V_A \) is a transistor constant. Assuming \( \eta_n = 5 \times 10^{-4} \) for pnp transistors, Eq. (14.93) gives

\[
r_{o4} = \frac{1}{\eta_n g_m} = \frac{1}{5 \times 10^{-4} \times 365.4 \mu A / V} = 5.47 \text{ M}\Omega
\]

\[
r_{o13A} = r_{o23} = \frac{1}{\eta_n g_{m13A}} = \frac{1}{5 \times 10^{-4} \times 6.92 \text{ mA/V}} = 289 \text{ k}\Omega
\]

\[
r_{o13B} = \frac{1}{\eta_n g_{m13B}} = \frac{1}{5 \times 10^{-4} \times 21.15 \text{ mA/V}} = 94.56 \text{ k}\Omega
\]

Assuming \( \eta_n = 2 \times 10^{-4} \) for npn transistors, Eq. (14.93) gives

\[
r_{o6} = \frac{1}{\eta_n g_m} = \frac{1}{2 \times 10^{-4} \times 365.4 \mu A / V} = 13.68 \text{ M}\Omega
\]

\[
r_{o17} = \frac{1}{\eta_n g_{m17}} = \frac{1}{2 \times 10^{-4} \times 21.15 \text{ mA/V}} = 236.4 \text{ k}\Omega
\]

**Input Stage**

Let us assume that \( v_{ic} = 0 \) and only \( v_{id} \) is applied. The AC equivalent circuit for the differential-mode signal is shown in Fig. 14.29(a). The input voltage to transistor Q1 is \( +v_{id} \) and that to transistor Q2 is \( -v_{id} \). Assuming that the transistors are identical and the circuit is balanced, an increase in voltage at the base junction of Q3 and Q4 due to \( +v_{id} \) will be compensated by an equal decrease in voltage due to \( -v_{id} \). The voltage at the base terminals of pnp transistors Q3 and Q4 will not vary at all. As a result, the bases of Q3 and Q4 are effectively at ground potential.
The transconductance of the input stage can be found by shorting the output to the ground and calculating the resulting current. Since $i_{c5}$ and $i_{c6}$ is the current mirror of $i_{c5}$, the current in the active load circuit ($i_{c6}$) will be equal in magnitude to the collector current of Q3 ($i_{c3}$); that is,

$$i_{c6} = i_{c3}$$

Thus, the output current under short-circuit conditions is

$$i_o = i_{c4} - i_{c6} = i_{c4} - i_{c3}$$

Under these conditions, Fig. 14.29(a) can be reduced to Fig. 14.29(b), which has two identical sides. The half-circuit AC equivalent is shown in Fig. 14.30(a), and its small-signal equivalent circuit is shown in Fig. 14.30(b), which can be simplified to Fig. 14.30(c).

![AC equivalent circuit](image)

**FIGURE 14.29** AC equivalent of input stage with differential input

![Simplified circuit](image)

**FIGURE 14.30** Half-circuit AC equivalent for differential input
From Fig. 14.30(b),

$$\frac{v_{id}}{2} = v_1 + v_3 \quad (14.96)$$

Summing the emitter currents at the emitter junctions of Q1 and Q3, we get

$$g_{m1}v_1 + i_1 = g_{m3}v_3 + i_3 \quad (14.97)$$

Substituting for $i_1$ and $i_3$, we get

$$g_{m1}v_1 + \frac{g_{m1}v_1}{\beta_{F1}} = g_{m3}v_3 + \frac{g_{m3}v_3}{\beta_{F3}}$$

or

$$g_{m1}v_1 \left[ 1 + \frac{1}{\beta_{F1}} \right] = g_{m3}v_3 \left[ 1 + \frac{1}{\beta_{F3}} \right] \quad (14.98)$$

where $\beta_{F1}$ and $\beta_{F3}$ are the small-signal current gains of transistors Q1 and Q3. Since $|I_{C1}| = |I_{C3}|$, $g_{m1} = g_{m3}$. Assuming that $\beta_{F1} >> 1$ and $\beta_{F3} >> 1$, Eq. (14.98) is reduced to

$$v_1 = v_3 \quad (14.99)$$

and Eq. (14.96) becomes

$$\frac{v_{id}}{2} = v_1 + v_3 = v_3 + v_3 = 2v_3$$

or

$$v_3 = \frac{v_{id}}{4} \quad (14.100)$$

Using Eq. (14.100), we can find the collector current of Q3:

$$i_{c3} = g_{m3}v_3 = \frac{g_{m3}v_{id}}{4} \quad (14.101)$$

From the symmetry of the circuit in Fig. 14.29(a), we get

$$i_{c4} = -i_{c3} = \frac{g_{m3}v_{id}}{4} \quad (14.102)$$

Substituting $i_{c3}$ from Eq. (14.101) and $i_{c4}$ from Eq. (14.102) into Eq. (14.95), we get the output current as

$$i_o = \frac{g_{m3}v_{id}}{4} + \frac{g_{m3}v_{id}}{4} = \frac{g_{m3}v_{id}}{2} \quad (14.103)$$

which gives the transconductance of the input stage as

$$G_{m1} = \frac{i_o}{v_{id}} = \frac{g_{m3}}{2} \quad (14.104)$$

$$= \frac{I_{C3}}{2V_T} = \frac{9.5 \mu A}{2 \times 26 \text{ mV}} = \frac{1}{5.47 \text{ k}\Omega} = 182.7 \mu A/V$$
Q3 can be replaced by the equivalent resistance $R_{eq}$ seen looking from the emitter of Q3, as shown in Fig. 14.30(c). $R_{eq}$ can be found from

$$R_{eq} = \frac{v_3}{g_m v_3 + v_3/r_{\pi 3}} = \frac{1}{g_m + 1/r_{\pi 3}} = \frac{1}{g_m(1 + 1/\beta_{F3})}$$  \hspace{1cm} (14.105)

$$\approx \frac{1}{g_m} = \frac{r_{\pi 3}}{\beta_{F3}} = \frac{r_{\pi 1}}{\beta_{F1}}$$  \hspace{1cm} (14.106)

Using Eq. (8.85), we can relate $v_{id}$ to $i_1$ as follows:

$$\frac{v_{id}}{2i_1} = r_{\pi 1} + R_{eq}(1 + \beta_{F1})$$  \hspace{1cm} (14.107)

Substituting $R_{eq} = r_{\pi 1} \beta_{F1}$ from Eq. (14.106) into Eq. (14.107) gives the input resistance as

$$R_{id} = \frac{v_{id}}{i_1} = 2 \left[ r_{\pi 1} + \frac{r_{\pi 1}}{\beta_{F1}} (1 + \beta_{F1}) \right]$$  \hspace{1cm} (14.108)

$$\approx 4r_{\pi 1}$$

$$= \frac{4\beta_{F1}V_T}{I_{C1}} = 4 \times \frac{250 \times 26 \text{ mV}}{9.69 \text{ \mu A}} = 2.68 \text{ M\Omega}$$

Notice from Eq. (14.108) that the input resistance is four times the input resistance of the input transistors. Also note that when $v_{id}$ changes, the output voltage changes and produces feedback to the input through the output resistance of Q4. As a result, the input resistances seen from the two input terminals will not be exactly the same. This effect is neglected in the derivation of $R_{id}$.

The output resistance $R_{o1}$ of the differential stage can be determined by setting the input voltage to zero and applying a test voltage $v_x$, as shown in Fig. 14.31(a). The analysis can be simplified by making the following assumptions:

1. Thevenin’s equivalent resistance $R_{th6}$ at the base of Q6 is very small compared to $r_{\pi}$ of Q6, so the base of Q6 is grounded. In reality, the voltage at this point is very small, and this point may be considered a virtual ground without greatly affecting the results. That is, $v_B6 \approx 0$. Figure 14.31(a) can be viewed as shown in Fig. 14.31(b).

2. The output resistance of Q2 is large and does not affect the results.

Figure 14.31(b) can be further simplified to the half circuit shown in Fig. 14.31(c). If Q2 is replaced by $1/g_{m2}$, Fig. 14.31(c) can be represented by the equivalent circuit in Fig. 14.32(a). If $R_{Q4}$ and $R_{Q6}$ are the effective resistances seen from the collectors of Q4 and Q6, respectively, Fig. 14.32(a) can be represented by Fig. 14.32(b). Using Eq. (9.113), we get

$$R_{Q4} = r_{o4} \left[ \frac{1 + g_{m4}(1/g_{m2})}{1 + g_{m4}/\beta_{F4}g_{m2}} \right]$$  \hspace{1cm} (14.109)

$$\approx 2r_{o4} \quad \text{for} \quad g_{m4} = g_{m2} \quad \text{and} \quad \beta_{F4} >> 1$$

$$= 2 \times 5.47 \text{ M\Omega} = 10.94 \text{ M\Omega}$$
FIGURE 14.31 Test circuit for calculation of $R_{o1}$

$$R_{Q6} = r_{o6} \left[ \frac{1 + g_{mn}R_2}{1 + g_{m6}R_2/\beta_{F6}} \right]$$  \hspace{1cm} (14.110)

$$= r_{o6} \left[ \frac{1 + 365.4 \ \mu A/V \times 1 \ \text{k} \Omega}{1 + 365.4 \ \mu A/V \times 1 \ \text{k} \Omega/250} \right] = 1.36 r_{o6}$$

$$= 1.36 \times 13.68 \ \text{M} \Omega = 18.65 \ \text{M} \Omega$$

FIGURE 14.32 Two-port equivalent of the input stage
The output resistance of the input stage is

\[ R_{o1} = R_{Q4} \parallel R_{Q6} = 10.94 \ \text{M}\Omega \parallel 18.65 \ \text{M}\Omega = 6.9 \ \text{M}\Omega \]

The two-port equivalent circuit of the input stage is shown in Fig. 14.32(c).

**Gain Stage**

The AC equivalent for the gain stage, shown in Fig. 14.33(a), can be represented by the equivalent circuit shown in Fig. 14.33(b). If \( R_{eq1} \) is Thevenin’s equivalent resistance seen looking into the base of transistor Q17, Fig. 14.33(a) can be reduced to the form shown in Fig. 14.33(c). From Eq. (8.85), we can find the input resistance of a common-emitter amplifier with a resistance in the emitter as follows:

\[ R_{eq1} = r_{\pi 17} + (1 + \beta_{F17})R_8 \]

\[ = 11.82 \ \text{k}\Omega + (1 + 250) \times 100 \ \Omega = 36.9 \ \text{k}\Omega \]

Using Eq. (8.85), the input resistance of the gain stage is

\[ R_i2 = r_{\pi 16} + (1 + \beta_{F16})(R_{eq1} \parallel R_9) \]

\[ = 403.7 \ \text{k}\Omega + (1 + 250)(36.9 \ \text{k}\Omega \parallel 50 \ \text{k}\Omega) = 5.73 \ \text{M}\Omega \]

Assuming that the voltage gain of the emitter follower Q16 is unity, the transconductance \( G_{m2} \) of this stage is that of the common-emitter amplifier of Q17 with resistance in the emitter; that is, the transconductance of the gain stage is

\[ G_{m2} = \frac{g_{m17}}{1 + g_{m17}R_8} \]

\[ = \frac{21.15 \ \text{mA/V}}{1 + 21.15 \ \text{mA/V} \times 100 \ \Omega} = \frac{1}{14 \ \Omega} = 6.79 \ \text{mA/V} \]
The circuit for determining the output resistance is shown in Fig. 14.34(a). The small-signal AC equivalent is shown in Fig. 14.34(b). If $R_{Q13B}$ and $R_{Q17}$ are the effective resistances seen from the collectors of $Q_{13B}$ and $Q_{17}$, respectively, Fig. 14.34(b) can be represented by the equivalent circuit in Fig. 14.34(c) and

$$R_{Q13B} = r_{o13B} = \frac{1}{\eta h_{f4}} = 94.56 \, \text{k}\Omega$$

Using Eq. (9.113), we get

$$R_{Q17} \approx r_{o17} \left[ \frac{1 + g_{m17} R_8}{1 + g_{m17} R_8 / \beta_{f17}} \right]$$

$$\approx r_{o17} \left[ \frac{1 + 21.15 \, \text{mA/V} \times 100 \, \Omega}{1 + 21.15 \, \text{mA/V} \times 100 \, \Omega / 250} \right] = 3.09 r_{o17}$$

$$= 3.09 \times 236.4 \, \text{k}\Omega = 730.5 \, \text{k}\Omega$$

The output resistance of the gain stage is

$$R_o = R_{Q13B} \parallel R_{Q17} = 94.56 \, \text{k}\Omega \parallel 730.5 \, \text{k}\Omega = 83.72 \, \text{k}\Omega$$

**Output Stage**

Since $V_{CE}$ of transistor $Q_{18}$ is the sum of the $V_{BE}$ voltages of $Q_{18}$ and $Q_{19}$, transistors $Q_{18}$ and $Q_{19}$ can be replaced by two diodes, as shown in Fig. 14.35(a). The output could be either sourcing or sinking, depending on the output voltage and the load. As a result, the input and resistances of this stage greatly depend on the particular values of output voltage and current. We will make the following assumptions:

1. The output is sourcing—that is, the output current is flowing out of the output stage.
2. The load current is $i_L = 2 \, \text{mA}$ at $R_L = 5 \, \text{k}\Omega$.
3. Transistor $Q_{14}$ is in the active region.
4. Transistor $Q_{20}$ is conducting a very small amount of current and is considered to be off.
The small-signal AC equivalent of Fig. 14.35(a) is shown in Fig. 14.35(b). Notice that the circuit consists of two emitter followers in series and that the voltage gain is approximately unity. Thus, the voltage gain of the output stage is

\[ A_3 \approx 1 \quad (14.115) \]

Since Q14 carries the load current \( i_L \) and a no-load DC-biasing collector current of Q14 of 153.7 \( \mu \)A,

\[ I_{C14} \approx I_{E4} = 2 \text{ mA } + 153.7 \text{ } \mu \text{A} = 2.15 \text{ mA} \]

Thus,

\[ r_{\pi14} = \frac{\beta_{F14} V_T}{I_{C14}} = \frac{250 \times 26 \text{ mV}}{2.15 \text{ mA}} = 3.02 \text{ k}\Omega \]

Since Q23 and diodes D18 and D19 operate approximately at a current of \( I_3 \approx 180 \text{ } \mu \)A,

\[ r_{d18} = r_{d19} = \frac{V_T}{I_{d18}} = \frac{26 \text{ mV}}{180 \text{ } \mu \text{A}} = 144 \text{ } \Omega \]

If \( R_{eq2} \) is the resistance seen looking at the base of Q14, we can use Eq. (8.85) to find \( R_{eq2} \):

\[ R_{eq2} = r_{\pi14} + (1 + \beta_{F14}) R_L \]

\[ = 3.02 \text{ k}\Omega + (1 + 250) \times 5 \text{ k}\Omega = 1258 \text{ k}\Omega \]

Thevenin’s equivalent resistance seen looking from the emitter of Q23 can be found from

\[ R_{eq3} = r_{d18} + r_{d19} + (r_{o13A} \parallel R_{eq2}) \quad (14.117) \]

\[ = 144 \text{ } \Omega + 144 \text{ } \Omega + (289 \text{ k}\Omega \parallel 1258 \text{ k}\Omega) = 235.3 \text{ k}\Omega \]
Using Eq. (8.85), we can find the input resistance of the stage:

\[ R_{i3} = r_{\pi23} + (1 + \beta_{F23})R_{eq3} \]
\[ = 7.2 \text{ k}\Omega + (1 + 50) \times 235.3 \text{ k}\Omega = 12 \text{ M}\Omega \]

Notice that the input resistance of the output stage (12 M\(\Omega\)) is much larger than the output resistance of the preceding stage (83.72 k\(\Omega\)). As a result, the overall gain of the amplifier is contributed by the differential and gain stages, the overall gain is not affected by the variations in external load resistance.

The output resistance of the output stage can be determined from the AC equivalent circuit shown in Fig. 14.36(a), which includes the output resistance \(R_{o2}\) of the preceding stage. Converting \(R_{o2}\) at the base of Q23 to its emitter gives the equivalent resistance seen looking from the base of Q14 toward Q23:

\[ R_{eq4} = r_{d18} + r_{d19} + \frac{R_{o2} + r_{\pi23}}{1 + \beta_{F23}} \]
\[ = 144 + 144 + \frac{83.72 \text{ k}\Omega + 7.2 \text{ k}\Omega}{1 + 50} = 2.07 \text{ k}\Omega \]

The resistance seen looking from the base of Q14 to the left is

\[ R_{eq5} = r_{o13A} \parallel R_{eq4} \]
\[ = 289 \text{ k}\Omega \parallel 2.07 \text{ k}\Omega = 2.06 \text{ k}\Omega \]

The resistance seen looking into the output terminal is

\[ R_{out} = \frac{R_{eq5} + r_{\pi14}}{1 + \beta_{F14}} \]
\[ = \frac{2.06 \text{ k}\Omega + 3.02 \text{ k}\Omega}{1 + 250} = 20.2 \Omega \]
The current-limiting resistance $R_6$ must be added to $R_{\text{out}}$ to find the actual output resistance of the op-amp:

$$R_o = R_{\text{out}} + R_6 = 20.2 + 27 = 47.2 \ \Omega$$

The two-port equivalent circuit of the output stage is shown in Fig. 14.36(b).

### 14.6.3 Frequency-Response Analysis

For $G_{m1} = 182.7 \ \mu A / V$ and $C_x = 30 \ \text{pF}$, Eq. (14.56) gives the unity-gain frequency (or bandwidth) as

$$f_u = \frac{G_{m1}}{2\pi C_x} = \frac{182.7 \ \mu A / V}{2\pi \times 30 \ \text{pF}} = 969.3 \ \text{kHz}$$

From Eq. (14.64), the slew rate is

$$SR = 4\pi V_T f_u = 4\pi \times 26 \text{ mV} \times 969.3 \ \text{kHz} = 0.317 \ \text{V/\mu s}$$

The manufacturer-specified values are $f_u = 1 \ \text{MHz}$ and $SR = 0.5 \ \text{V/\mu s}$. The discrepancy between the calculated values and the manufacturer-specified values is caused by the fact that Eq. (14.64) gives an approximate value of $SR$ and does not take into account the input and output resistances of preceding and subsequent amplifier stages.

### 14.6.4 Small-Signal Equivalent Circuit

The small-signal equivalent of the complete circuit is shown in Fig. 14.37. The voltage gain is

$$A_v = G_{m1}(R_{o1} \parallel R_{i2}) \ G_{m2}(R_{o2} \parallel R_{i3}) = 182.7 \ \mu A / V \times (6.9 \ \text{M}\Omega \parallel 5.73 \ \text{M}\Omega) \times 6.79 \text{ mA/V} \times (83.72 \ \text{k}\Omega \parallel 12.0 \ \text{M}\Omega) = 571.9 \times 564.5 = 323,000$$

\[ \text{FIGURE 14.37 Small-signal equivalent circuit for the LM741 op-amp} \]
where 
$G_{m1} =$ the transconductance of the first (differential) stage 
$G_{m2} =$ the transconductance of the second (gain) stage 
$R_{o1} =$ the output resistance of the first (differential) stage 
$R_{o2} =$ the output resistance of the second (gain) stage 
$R_{i3} =$ the input resistance of the third (output) stage 

Input and output resistances are

$$R_{id} = 2.68 \ \text{M\Omega} \quad \text{and} \quad R_o = 47.2 \ \Omega$$

**NOTES**

1. The input stage and the gain stage contribute about the same gain: 571.9 and 564.5, respectively. The gain stage loads the input stage and reduces its gain by about half. This loading makes the op-amp beta dependent, causing the gain to vary with temperature and fabrication-process tolerances.
2. The output stage does not significantly load the gain stage, and the voltage gain is almost independent of the external load resistance.
3. Although the results of the analysis are approximate, they give insight into the operation and performance of the op-amp.
4. The output resistances of transistors were neglected in the DC analysis. If these were taken into consideration, the biasing currents would change, thereby changing the small-signal output resistance and the results.
5. The variation in the transistor beta was neglected. But the transistor current gain falls at low collector current levels because of recombination in the emitter–base space charge layer.

**KEY POINTS OF SECTION 14.6**

- This section illustrated a three-part analysis of a complete op-amp circuit, the LM741: DC analysis to find the DC biasing currents, small-signal analysis to find the voltage gain, and analysis of the frequency response to find the unity-gain bandwidth for all stages of the op-amp.
- The values obtained from hand calculations correspond well to the values specified in the manufacturer's data sheet. That is, $A_v = 323,000$ (compared to 200,000), $R_{id} = 2.68 \ \text{M\Omega}$ (compared to 2 \ \text{M\Omega}), $R_o = 47.2 \ \Omega$ (compared to 75 \ \Omega), $f_u = 969.3 \ \text{kHz}$ (compared to 1 MHz), and $SR = 0.317 \ \text{V\mu s}$ (compared to 0.5 V\mu s).

**14.7 BiCMOS Op-Amps**

BiCMOS op-amps contain both CMOS and BJT transistors on the same chip. This arrangement incorporates the advantages of both BJTs and MOSFETs to achieve desirable characteristics such as high input resistance, low offset, large gain, and wide bandwidth.

**14.7.1 BiCMOS Op-Amp CA3130**

The simplified schematic of the BiCMOS op-amp CA3130 is shown in Fig. 14.38. The internal structure can be divided into three stages: a differential MOS stage, a gain stage, and an output stage. Some parameters of the CA3130 op-amp are listed in Table 14.6.
**Differential Stage**

The input stage is a PMOS differential amplifier consisting of transistors Q₆ and Q₇. This stage is biased by a current source of $I_Q$, and it drives a current mirror load consisting of BJT transistors Q₉ and Q₁₀. Although a BJT active load offers high resistance, the voltage gain is only about 5 because of the relatively low transconductance of MOSFETs.

**TABLE 14.6  Parameters of op-amp CA3130**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Minimum</th>
<th>Typical</th>
<th>Maximum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC supply voltages $V_{DD}$</td>
<td></td>
<td>±18</td>
<td></td>
<td>V</td>
</tr>
<tr>
<td>Input offset voltage $V_{OS}$</td>
<td>8.0</td>
<td>15.0</td>
<td></td>
<td>mV</td>
</tr>
<tr>
<td>Thermal drift $D_v$</td>
<td>15</td>
<td>30</td>
<td></td>
<td>$\mu$V °C</td>
</tr>
<tr>
<td>Input biasing current $I_B$</td>
<td>5</td>
<td>50</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Input offset current $I_{OS}$</td>
<td>0.5</td>
<td>30</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Differential input resistance $R_{id}$</td>
<td>1.5</td>
<td></td>
<td></td>
<td>TΩ</td>
</tr>
<tr>
<td>Common-mode input resistance $R_{ic}$</td>
<td>1.5</td>
<td></td>
<td></td>
<td>TΩ</td>
</tr>
<tr>
<td>Output resistance $R_o$</td>
<td>75</td>
<td></td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Input capacitance $C_i$</td>
<td>4.3</td>
<td></td>
<td></td>
<td>pF</td>
</tr>
<tr>
<td>Open-loop voltage gain $A_o$</td>
<td>50</td>
<td>320</td>
<td></td>
<td>V mV</td>
</tr>
<tr>
<td>Common-mode rejection ratio (CMRR)</td>
<td>70</td>
<td>90</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Unity-gain bandwidth $f_u$</td>
<td>15</td>
<td></td>
<td></td>
<td>MHz</td>
</tr>
<tr>
<td>Slew rate (SR)</td>
<td>10</td>
<td></td>
<td></td>
<td>V $\mu$s</td>
</tr>
<tr>
<td>Power supply rejection ratio (PSRR)</td>
<td>70</td>
<td>90</td>
<td></td>
<td>dB</td>
</tr>
</tbody>
</table>
Resistors $R_5$ and $R_6$ allow offset voltage adjustment (in the range of $\pm R_5 I_Q/2 = \pm 1 \text{k}\Omega \times 100 \mu\text{A} = \pm 100 \text{mV}$) through an externally connected potentiometer (typically 10 k\OHM) across terminals 1 and 5. Also, resistor $R_6$ increases the output resistance of the active load and hence the voltage gain. Zener diodes $D_5$ and $D_8$ protect the thin gate-oxide of the MOSFETs from excessive voltage spikes and static discharge, which could cause breakdown of the oxide layer and hence damage the transistors.

**Gain Stage**

The gain stage is a common-emitter amplifier consisting of transistor $Q_{11}$ and has an active current load for a large voltage gain (about 6000). Note that the absence of a compensation capacitor gives a high bandwidth (15 MHz).

**Output Stage**

The output stage is a CMOS push-pull stage consisting of PMOS $Q_8$ and NMOS $Q_{12}$. If the voltage at the collector of $Q_{11}$ increases by a small amount above the quiescent level, then NMOS transistor $Q_{12}$ turns on and PMOS transistor $Q_8$ remains off. On the other hand, if the voltage goes down by a small amount, then PMOS transistor $Q_8$ turns on and NMOS transistor $Q_{12}$ remains off. The voltage gain of the output stage is about 30.

### 14.7.2 BiCMOS Op-Amp CA3140

The simplified schematic of the BiCMOS op-amp CA3140 is shown in Fig. 14.39. Its internal structure is similar to that of op-amp CA3130 in Fig. 14.38, except for the output stage and the addition of a compensation capacitor $C_x$ in the second stage. The voltage gain of the differential stage is about 10. The offset

![Simplified schematic for op-amp RCA-CA3140](image-url)
voltage adjustment is in the range of \( \pm R_{3}I_{Q} = \pm 500 \, \Omega \times 100 \, \mu A = \pm 50 \, mV \). \( C_{x} \) provides feedback stability, but it reduces the bandwidth. The Darlington npn emitter follower in the output stage increases the effective load resistance seen by transistor \( Q_{13} \) in the gain stage and hence increases the voltage gain to about 10. The voltage gain of the gain stage is about 10,000. Some parameters of the CA3140 op-amp are listed in Table 14.7.

### Output Stage

The Darlington emitter follower offers a lower-than-average output resistance and a high resistance to the gain stage. If the voltage at the collector of transistor \( Q_{13} \) goes up in the positive direction above the quiescent value, transistor \( Q_{18} \) will drive the load and source current of \( I_{Q15} \) (2 mA). However, if the current of \( Q_{18} \) falls below the level of \( I_{Q15} \) (2 mA), the voltage across the load will go down below the quiescent level. Transistor \( Q_{18} \) will always remain on. This will cause MOSFET \( Q_{21} \) to turn on. Since the collector current of \( Q_{16} \) will be the mirror of the drain current of MOSFET \( Q_{21} \), transistor \( Q_{16} \) will sink the load current.

### 14.7.3 BiCMOS Op-Amp LH0022

The simplified schematic of the LH0022 op-amp is shown in Fig. 14.40, which is a modification of the original circuit by replacing junction field-effect transistors \( Q_{1} \) and \( Q_{2} \) with depletion-type NMOS. The LH0022 is designed to operate from a \( \pm 15-V \) power supply. It is capable of a peak output voltage swing of about 12 V into a 1-k\( \Omega \) load over the entire frequency range of 1 MHz. The circuit can be divided into a differential stage, a gain stage, an output stage, overload protection, and a DC-biasing circuit. Some parameters of the LH0022 are listed in Table 14.8.

### Differential Stage

The differential input stage consists of \( Q_{1} \) through \( Q_{4} \). Transistors \( Q_{1} \) and \( Q_{2} \) are depletion NMOSs that operate in a common-drain (or source-follower) configuration. These NMOSs drive transistors \( Q_{3} \) and \( Q_{4} \),

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Minimum</th>
<th>Typical</th>
<th>Maximum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC supply voltages ( V_{DD} )</td>
<td>±18</td>
<td></td>
<td></td>
<td>V</td>
</tr>
<tr>
<td>Input offset voltage ( V_{OS} )</td>
<td>8.0</td>
<td>15.0</td>
<td></td>
<td>mV</td>
</tr>
<tr>
<td>Thermal drift ( D_{v} )</td>
<td>10</td>
<td>30</td>
<td></td>
<td>( \mu V , ^{\circ}C )</td>
</tr>
<tr>
<td>Input biasing current ( I_{B} )</td>
<td>10</td>
<td>50</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Input offset current ( I_{OS} )</td>
<td>0.5</td>
<td>30</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Differential input resistance ( R_{id} )</td>
<td>1.5</td>
<td></td>
<td></td>
<td>T( \Omega )</td>
</tr>
<tr>
<td>Common-mode input resistance ( R_{ic} )</td>
<td>1.5</td>
<td></td>
<td></td>
<td>T( \Omega )</td>
</tr>
<tr>
<td>Output resistance ( R_{o} )</td>
<td>50</td>
<td></td>
<td></td>
<td>( \Omega )</td>
</tr>
<tr>
<td>Input capacitance ( C_{i} )</td>
<td>1</td>
<td></td>
<td></td>
<td>pF</td>
</tr>
<tr>
<td>Open-loop voltage gain ( A_{o} )</td>
<td>20</td>
<td>100</td>
<td></td>
<td>V mV</td>
</tr>
<tr>
<td>Common-mode rejection ratio (CMRR)</td>
<td>70</td>
<td>90</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Unity-gain bandwidth ( f_{u} )</td>
<td>4.5</td>
<td></td>
<td></td>
<td>MHz</td>
</tr>
<tr>
<td>Slew rate (SR)</td>
<td>9</td>
<td></td>
<td></td>
<td>V ( \mu s )</td>
</tr>
<tr>
<td>Power supply rejection ratio (PSRR)</td>
<td>76</td>
<td>80</td>
<td></td>
<td>dB</td>
</tr>
</tbody>
</table>
which operate in a common-base configuration. The current mirror, consisting of transistors Q5, Q6, and Q7, acts as the active load.

The combination of common-drain and common-base configurations provides a very high input resistance and a very low input biasing current from the MOSFETs while giving a large voltage gain from the BJTs. Also, the common-drain configuration performs the function of shifting voltage levels

**TABLE 14.8** Parameters of op-amp LH0022

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Minimum</th>
<th>Typical</th>
<th>Maximum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC supply voltages (V_{CC}, V_{EE})</td>
<td>15</td>
<td>±15</td>
<td></td>
<td>V</td>
</tr>
<tr>
<td>Input offset voltage (V_{OS})</td>
<td>6.0</td>
<td>20</td>
<td></td>
<td>mV</td>
</tr>
<tr>
<td>Thermal drift (D_v)</td>
<td>10</td>
<td>100</td>
<td></td>
<td>μV °C</td>
</tr>
<tr>
<td>Input biasing current (I_B)</td>
<td>15</td>
<td>50</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Input offset current (I_{OS})</td>
<td>2</td>
<td>10</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Differential input resistance (R_{id})</td>
<td>10(^{12})</td>
<td>10(^{12})</td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Common-mode input resistance (R_{IC})</td>
<td>10(^{12})</td>
<td>10(^{12})</td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Output resistance (R_o)</td>
<td>75</td>
<td>100</td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Input capacitance (C_i)</td>
<td>4.0</td>
<td>20</td>
<td></td>
<td>pF</td>
</tr>
<tr>
<td>Open-loop voltage gain (A_o)</td>
<td>25</td>
<td>100</td>
<td></td>
<td>V mV</td>
</tr>
<tr>
<td>Common-mode rejection ratio (CMRR)</td>
<td>70</td>
<td>80</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Unity-gain bandwidth (f_o)</td>
<td>1.0</td>
<td>3.0</td>
<td></td>
<td>MHz</td>
</tr>
<tr>
<td>Slew rate (SR)</td>
<td>1.0</td>
<td>3.0</td>
<td></td>
<td>V μs</td>
</tr>
<tr>
<td>Power supply rejection ratio (PSRR)</td>
<td>70</td>
<td>80</td>
<td></td>
<td>dB</td>
</tr>
</tbody>
</table>
toward the negative supply voltage $-V_{EE}$ so that the output signal can be shifted upward in the positive direction in the following stages. Note that the DC level of the output voltage of this stage will be approximately

$$-V_{EE} + V_{BE16} + V_{BE17} = -V_{EE} + 1.2 \text{ V}$$

**Gain Stage**

The gain stage has the common Collector and common-emitter configurations forming a Darlington pair made up of Q$_{16}$ and Q$_{17}$. This arrangement offers a high load resistance to the differential stage and hence provides a large voltage gain. Capacitor $C_x$ is the pole-splitting compensating capacitor connected in shunt-shunt feedback, and it controls the unity-gain bandwidth and the slew rate.

**Output Stage**

The output stage is a complementary class AB push-pull circuit consisting of transistors Q$_{14}$ and Q$_{20}$. Diode-connected transistors Q$_{18}$ and Q$_{19}$ provide the biasing voltage for class AB operation in order to reduce crossover distortion.

**Protection Circuitry**

Transistors Q$_{15}$ and Q$_{21}$ act as the current booster. Resistors $R_{e1}$ and $R_{e2}$ provide short-circuit protection by turning on Q$_{15}$ and Q$_{21}$ so as to limit the currents through Q$_{14}$ and Q$_{20}$, respectively.

**Biasing Circuitry**

The complete schematic, including the biasing circuitry, is shown in Fig. 14.41, which has an extra protection circuit (consisting of transistors Q$_{23}$ and Q$_{24}$) to short the input terminal of Q$_{16}$ (in the gain stage) to ground through Q$_{23}$. Transistors Q$_{10}$ and Q$_{11}$ form a Widlar current mirror, which biases transistors Q$_{3}$ and Q$_{4}$ of the common-base configuration. The current source consisting of transistors Q$_{12}$ and Q$_{13}$ biases the gain and output stages. The reference current $I_{ref}$ can be found from

$$I_{ref} = \frac{V_{CC} + V_{EE} - V_{BE11} - V_{EB12}}{R_5} \tag{14.124}$$

Transistor Q$_{13}$ is a multicollector lateral pnp device. Its geometry is shown in Fig. 14.42(a), and its symbol is shown in Fig. 14.42(b). The collector ring has been split into two parts—one part faces on three-fourths of the emitter periphery and collects the holes injected from that periphery, and the second one faces on one-fourth of the emitter periphery and collects the holes from that periphery. The structure is analogous to two pnp transistors whose base-emitter junctions are connected in parallel, one with an $I_S$ that is one-fourth that of a standard pnp transistor and the other with an $I_S$ that is three-fourths that of a standard pnp transistor. This electrical equivalence is shown in Fig. 14.42(c). Thus, the currents $I_{Q2}$ and $I_{Q3}$ are three-fourths and one-fourth of the reference current $I_{ref}$, respectively. That is,

$$I_{Q2} = \frac{3I_{ref}}{4} \quad \text{and} \quad I_{Q3} = \frac{I_{ref}}{4}$$
FIGURE 14.41 Schematic for op-amp LH0022 (Courtesy of National Semiconductor, Inc.)
14.7.4 BiCMOS Op-Amp LF411

The simplified schematic of the LF411 op-amp is shown in Fig. 14.43, which is a modification of the original circuit by replacing junction field-effect transistors $J_1$ and $J_2$ with depletion-type PMOS. Its internal structure is similar to that of the LH0022 op-amp. The differential input stage consists of $p$-channel depletion transistors $M_1$ and $M_2$ with a current mirror load. This stage has low offset, low drift, and a high unity-gain bandwidth. The complete schematic appears in Fig. 14.44, which shows in detail the biasing circuitry. There is a transistor $Q_3$ in the drain of $M_2$, but none in that of $M_1$. Depletion NMOS $M_2$ operates in a common-source configuration, whereas MOSFET $M_1$ operates in a common-drain configuration. This type of arrangement gives a lower voltage gain but a wider bandwidth.

**FIGURE 14.42** Electrical equivalent for multicollector lateral $pnp$ transistor

**FIGURE 14.43** Simplified schematic for op-amp LF411 (Courtesy of National Semiconductor, Inc.)
A large value of DC biasing circuit $I_Q$ (in milliamperes) can be used for depletion NMOS differential amplifiers without significantly affecting the input biasing currents. On the other hand, for BJT differential amplifiers $I_Q$ is kept small (in microamperes) to have a low input biasing current or high input resistance. Some parameters of the LF411 op-amp are listed in Table 14.9.

**DC Biasing**

The zener diode $Z_1$ allows stable DC biasing with thermal compensation by diode $D_2$ and resistor $R_4$. With MOSFET $M_3$ acting as a current-regulator diode (see Sec. 9.6), the current through zener diode $Z_1$ is kept constant at $I_{DSS}$. If $V_Z$ is the zener voltage, then

$$I_{ref} = \frac{V_Z - V_{BE16} - V_{D2}}{R_4}$$  \hspace{1cm} (14.125)$$

which in turn sets the DC biasing currents $I_{Q1}$ and $I_{Q2}$.
Thermal Protection

Transistor Q19 is normally off. If the temperature rises, $V_{BE16}$ will fall because of the negative temperature coefficient of Q16, and the zener voltage $V_Z$ will rise because of the positive temperature coefficient of zener diode $Z_Z$. As a result, the voltage at the emitter of Q16 will rise; thus, the voltage at the anode of D2 will also rise. If the temperature rise is adequate, Q18 and Q19 will turn on and reduce the gain and the output voltage of the amplifier.

14.7.5 BiCMOS Op-Amp LH0062

The simplified schematic of the LH0062 op-amp is shown in Fig. 14.45, which is a modification of the original circuit by replacing junction field-effect transistors Q1 and Q2 with depletion-type NMOS. The internal structure is similar to that of op-amp LH0022.

Differential Stage

The differential input stage consists of n-channel transistors Q1 and Q2 with an active load. This is a normal source-coupled differential pair with a difference output, which gives a lower gain but a wider bandwidth. The input resistance is very high for the MOSFET input stage. The DC biasing current $I_Q$ is set by the current source consisting of transistor Q3 and zener diode D1. Thus,

$$I_Q = \frac{V_Z - V_{BE3}}{R_5}$$  \hspace{1cm} (14.126)

$$= \frac{1.2 \text{ V} - 0.6 \text{ V}}{600 \Omega} = 1 \text{ mA}$$

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Minimum</th>
<th>Typical</th>
<th>Maximum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC supply voltages $V_{CC}$, $V_{EE}$</td>
<td>±18 V</td>
<td></td>
<td></td>
<td>V</td>
</tr>
<tr>
<td>Input offset voltage $V_{OS}$</td>
<td>0.8 mV</td>
<td>2.0 mV</td>
<td></td>
<td>mV</td>
</tr>
<tr>
<td>Thermal drift $D_v$</td>
<td>7 μV °C</td>
<td>20 μV °C</td>
<td></td>
<td>μV °C</td>
</tr>
<tr>
<td>Input biasing current $I_B$</td>
<td>50 pA</td>
<td>200 pA</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Input offset current $I_{OS}$</td>
<td>25 pA</td>
<td>100 pA</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Differential input resistance $R_{id}$</td>
<td>$10^{12}$ Ω</td>
<td></td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Common-mode input resistance $R_{ic}$</td>
<td>$10^{12}$ Ω</td>
<td></td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Output resistance $R_o$</td>
<td>75 Ω</td>
<td></td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Input capacitance $C_i$</td>
<td>4.0 pF</td>
<td></td>
<td></td>
<td>pF</td>
</tr>
<tr>
<td>Open-loop voltage gain $A_o$</td>
<td>25 V mV</td>
<td>200 V mV</td>
<td></td>
<td>V mV</td>
</tr>
<tr>
<td>Common-mode rejection ratio (CMRR)</td>
<td>70 dB</td>
<td>100 dB</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Unity-gain bandwidth $f_u$</td>
<td>2.7 MHz</td>
<td>4.0 MHz</td>
<td></td>
<td>MHz</td>
</tr>
<tr>
<td>Slew rate (SR)</td>
<td>8.0 V μs</td>
<td>15 V μs</td>
<td></td>
<td>V μs</td>
</tr>
<tr>
<td>Power supply rejection ratio (PSRR)</td>
<td>70 dB</td>
<td>100 dB</td>
<td></td>
<td>dB</td>
</tr>
</tbody>
</table>
Gain Stage

The gain stage is a common-emitter coupled pair, which drives a current mirror load consisting of transistors \( Q_6 \) and \( Q_7 \). The single-ended output gives a larger voltage gain. The biasing current \( I_{Q1} \) can be found from

\[
I_{Q1} R_3 + V_{EB4} = \frac{R_2 I_Q}{2}
\]

That is,

\[
I_{Q1} = \frac{R_2 I_Q / 2 - V_{EB4}}{R_3}
\]  

\[
= \frac{20 \text{ k}\Omega \times 0.5 \text{ mA} - 0.6 \text{ V}}{6 \text{ k}\Omega} = 1.6 \text{ mA}
\]

The collector current of \( Q_7 \) is the mirror of that of \( Q_6 \); that is, \( I_{Q2} \) is the mirror of \( I_{Q1} \), and \( I_{Q2} = I_{Q1} = 1.6 \text{ mA} \), which is also the biasing current for the output stage. The high gain and wider bandwidth are obtained by cascading two differential stages. The slew rate is also high because of the higher DC biasing current. Some parameters of the LH0062 op-amp are listed in Table 14.10.

FIGURE 14.45  Simplified schematic for op-amp LH0062 (Courtesy of National Semiconductor, Inc.)
14.7.6 BiCMOS Op-Amp LH0032

The LH0032 is an ultrafast op-amp. Its schematic is shown in Fig. 14.46, which is a modification of the original circuit by replacing junction field-effect transistors \( Q_1 \) and \( Q_2 \) with depletion-type NMOS. The internal structure is similar to that of op-amp LH0062, except that the second differential stage uses common-base

### TABLE 14.10 Parameters of op-amp LH0062

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Minimum</th>
<th>Typical</th>
<th>Maximum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC supply voltages ( V_{CC}, V_{EE} )</td>
<td>( \pm 18 ) V</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Input offset voltage ( V_{OS} )</td>
<td>10</td>
<td>15</td>
<td></td>
<td>mV</td>
</tr>
<tr>
<td>Thermal drift ( D_\theta )</td>
<td>10</td>
<td>35</td>
<td></td>
<td>( \mu V \ °C )</td>
</tr>
<tr>
<td>Input biasing current ( I_B )</td>
<td>10</td>
<td>65</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Input offset current ( I_{OS} )</td>
<td>1</td>
<td>5</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Differential input resistance ( R_{id} )</td>
<td>( 10^{12} ) ( \Omega )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Common-mode input resistance ( R_{ic} )</td>
<td>( 10^{12} ) ( \Omega )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Output resistance ( R_o )</td>
<td>75</td>
<td></td>
<td></td>
<td>( \Omega )</td>
</tr>
<tr>
<td>Input capacitance ( C_i )</td>
<td>4.0</td>
<td></td>
<td></td>
<td>pF</td>
</tr>
<tr>
<td>Open-loop voltage gain ( A_o )</td>
<td>25</td>
<td>160</td>
<td></td>
<td>V mV</td>
</tr>
<tr>
<td>Common-mode rejection ratio (CMRR)</td>
<td>70</td>
<td>90</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Unity-gain bandwidth ( f_u )</td>
<td>70</td>
<td>15</td>
<td></td>
<td>MHz</td>
</tr>
<tr>
<td>Slew rate (SR)</td>
<td>50</td>
<td>75</td>
<td></td>
<td>V ( \mu s )</td>
</tr>
<tr>
<td>Power supply rejection ratio (PSRR)</td>
<td>70</td>
<td>90</td>
<td></td>
<td>dB</td>
</tr>
</tbody>
</table>

![Schematic for op-amp LH0032](image-url)
14.8 Design of Op-Amps

An operational amplifier is a complete IC that is expected to meet certain specifications with respect to input resistance, output resistance, gain, CMRR, and bandwidth. An op-amp is designed at the system level. The process involves designing the amplifying stages and protection circuitry and requires the following steps [9]:

**Step 1.** Identify the important specifications: the voltage gain $A_d$, the CMRR, the input resistance $R_{id}$, the output resistance $R_o$, the unity-gain bandwidth, and the DC supply voltages $V_{CC}$ and $V_{EE}$ (or $V_{DD}$ and $V_{SS}$).

### TABLE 14.11 Parameters of op-amp LH0032

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Minimum</th>
<th>Typical</th>
<th>Maximum</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC supply voltages $V_S$</td>
<td>±18 V</td>
<td></td>
<td></td>
<td>V</td>
</tr>
<tr>
<td>Input offset voltage $V_{OS}$</td>
<td>1.0 mV</td>
<td>2.0 mV</td>
<td></td>
<td>mV</td>
</tr>
<tr>
<td>Thermal drift $D_v$</td>
<td>15 μV °C</td>
<td>30 μV °C</td>
<td></td>
<td>μV °C</td>
</tr>
<tr>
<td>Input biasing current $I_B$</td>
<td>50 pA</td>
<td>150 pA</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Input offset current $I_{OS}$</td>
<td>10 pA</td>
<td>30 pA</td>
<td></td>
<td>pA</td>
</tr>
<tr>
<td>Differential input resistance $R_{id}$</td>
<td>$10^{12}$ Ω</td>
<td>$10^{12}$ Ω</td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Common-mode input resistance $R_{ic}$</td>
<td>$10^{12}$ Ω</td>
<td>$10^{12}$ Ω</td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Output resistance $R_o$</td>
<td>75 Ω</td>
<td></td>
<td></td>
<td>Ω</td>
</tr>
<tr>
<td>Input capacitance $C_i$</td>
<td>4.0 pF</td>
<td></td>
<td></td>
<td>pF</td>
</tr>
<tr>
<td>Open-loop voltage gain $A_o$</td>
<td>60 dB</td>
<td>70 dB</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Common-mode rejection ratio (CMRR)</td>
<td>50 dB</td>
<td>60 dB</td>
<td></td>
<td>dB</td>
</tr>
<tr>
<td>Unity-gain bandwidth $f_u$</td>
<td>70 MHz</td>
<td></td>
<td></td>
<td>MHz</td>
</tr>
<tr>
<td>Slew rate (SR)</td>
<td>350 V μs</td>
<td>500 V μs</td>
<td></td>
<td>V μs</td>
</tr>
<tr>
<td>Power supply rejection ratio (PSRR)</td>
<td>50 dB</td>
<td>60 dB</td>
<td></td>
<td>dB</td>
</tr>
</tbody>
</table>

cascode BJTs for a higher voltage gain. The zener diode $D_{Z1}$ gives a stable reference voltage for the biasing circuit consisting of transistors $Q_8$ and $Q_9$. In general, a differential stage with a difference output (i.e., the first differential stage) gives a lower voltage gain but a wider bandwidth. A differential pair with a single-ended output (i.e., the second differential stage) gives a higher voltage gain but a lower bandwidth. However, the common-base cascode connection gives a wider bandwidth.

The LH0032 op-amp has very high gain, a very large slew rate, and a very large bandwidth. Note that there is no compensation capacitor. As a result, the bandwidth is widened. Some parameters of the LH0032 op-amp are listed in Table 14.11.

**KEY POINTS OF SECTION 14.7**

- BiCMOS op-amps contain both CMOS and BJT transistors on the same chip. The advantages of both BJTs and MOSFETs are utilized to achieve desirable characteristics such as high input resistance (1.5 TΩ), low input biasing current (10 pA), large gain (100 dB), and wide bandwidth (4.5 MHz).
- The difference output (70 MHz for the LH0032 op-amp), which gives a lower gain but wider bandwidth, is normally used for ultrafast op-amps. Compensation capacitance is avoided (see Fig. 14.11).
Step 2. Select the type of op-amp (BJT, CMOS, or BiCMOS).

Step 3. Choose the circuit configurations and stages.

Step 4. Determine the biasing current requirement $I_Q$ for the desired specifications.

Step 5. Choose the type of current source (BJT or MOSFET) and determine its component ratings.

Step 6. Choose the active load with a current mirror (BJT or MOSFET) in order to obtain the desired voltage gain and determine its component ratings.

Step 7. Choose the type of gain stage (BJT or MOSFET) and determine its component ratings.

Step 8. Choose the type of output stage (BJT or MOSFET) and determine its component ratings.

Step 9. Choose the type of frequency compensation (pole or pole-zero circuit) and determine its component ratings.

Step 10. Determine the voltage, current, and power ratings of active and passive components.

Step 11. Analyze and evaluate the complete differential amplifier to see that it meets the desired specifications.

Step 12. Use PSpice/SPICE to simulate and verify your design. If it would be implemented with discrete devices, use standard values of components, with tolerances of, say, 5%.

Summary

The open-loop voltage gain of an op-amp does not remain constant, but rather decreases with frequency. The frequency response of an internally compensated op-amp has the characteristic of a single time-constant network. The slew rate of the input signal should be less than that of the op-amp in order to avoid distortion of the output voltage. The common-mode rejection ratio is a measure of the ability of an op-amp to reject common-mode signals; the ratio should be as high as possible.

The characteristics of practical op-amps differ from those of ideal ones. The output of an op-amp is affected by parameters such as input offset voltage, input offset current, input biasing current, thermal drift, the power supply rejection ratio, and the input frequency. The effect of input biasing currents can be minimized by adding an offset-minimizing resistor.

The internal structure of an op-amp usually consists of differential, gain, and output stages. There are many possible combinations of op-amp stages, depending on whether the amplifier is a bipolar, MOSFET, or CMOS op-amp. In general, a MOSFET op-amp gives a very high input resistance but less gain, whereas a bipolar op-amp gives a higher gain but lower input resistance. A Darlington BJT pair is commonly used for high current gain and input resistance. A difference output of a differential stage gives less gain but yields wider bandwidth.

Op-amps exhibit offset voltages and currents, which can be minimized by appropriate internal design. The input offset voltage is dependent on the CMRR and the common-mode signal. A pole or pole-zero compensation circuit is commonly used for frequency compensation, but the compensation comes at the expense of unity-gain bandwidth.

The small-signal voltage gain and the input and output resistances of the LM741 op-amp can be determined for various load conditions. Since the voltage gain is very high, the amplifier offers offset voltages and currents because of variations in the transistor parameters and circuit resistances. Although the analysis required to derive equations describing the characteristics of current sources and differential amplifiers can be simplified with some assumptions, computer-aided analysis is generally required to evaluate the actual
performance of an op-amp at the final design stage. The analysis of other op-amps would be similar to the analysis of the LM741 op-amp in this chapter.

References


Review Questions

1. What are the main stages of an op-amp?
2. What is the input biasing current of an op-amp?
3. What is the input offset current of an op-amp, and what factors influence its value?
4. What is the input offset voltage of an op-amp, and what factors influence its value?
5. What is the CMRR of an op-amp?
6. What factors influence the unity-gain bandwidth of an op-amp?
7. What factors influence the slew rate of an op-amp?
8. What is the relation between slew rate and unity-gain bandwidth?
9. What is the typical value of the input resistance of an op-amp?
10. What is the typical value of the output resistance of an op-amp?
11. Ideally, what should be the differential voltage gain of an op-amp?
12. Ideally, what should be the common-mode voltage gain of an op-amp?
13. What is the unity-gain bandwidth of an op-amp?
14. What is the effect of rise time on the frequency response of an op-amp?
15. What is a slew rate?
16. What is the slew rate of a step input voltage?
17. What is the slew rate of a sinusoidal input voltage?
18. What is the feedback factor?
19. What is the typical break frequency of an op-amp?
20. What is the effect of input offset voltage on the output of inverting and noninverting amplifiers?
21. What is the effect of input biasing currents on the output of inverting and noninverting amplifiers?
22. What is the effect of input offset current on the output of inverting and noninverting amplifiers?
23. What is the common method for minimizing the effect of input biasing currents?
24. What is the cause of thermal drift?
25. What is the effect of thermal drift?
26. What is the effect of output offset voltage on an integrator?
27. What is the PSRR?
28. What is pole-zero compensation of an op-amp?
29. What circuit configurations are used for ultrafast op-amps?
30. What are the advantages and disadvantages of MOSFET op-amps?
31. What are the advantages and disadvantages of bipolar op-amps?
32. What are the advantages and disadvantages of CMOS op-amps?
33. What are the advantages and disadvantages of BiCMOS op-amps?
34. What is the function of short-circuit protection in op-amps?
35. What are the typical values of input resistance, output resistance, and voltage gain for the LM741 op-amp?

Problems

The symbol 🔄 indicates that a problem is a design problem. The symbol 🔄 indicates that you can check the solution to a problem using PSpice/SPICE or Electronics Workbench.

14.3 Parameters and Characteristics of Practical Op-Amps

14.1 The input voltages of an op-amp are \( v_1 = 120 \, \mu V \) and \( v_2 = 80 \, \mu V \). The op-amp parameters are \( A_d = A_0 = 2 \times 10^5 \). Determine (a) the differential voltage \( v_{d} \), (b) the common-mode voltage \( v_{c} \), (c) the magnitude of the common-mode gain \( A_{cm} \), and (d) the output voltage \( v_{o} \).

14.2 The rise time of an op-amp is \( t_r = 0.3 \, \mu s \). What is the maximum frequency limit of the op-amp?

14.3 The inverting amplifier in Fig. 14.9(a) has \( R_1 = 15 \, k\Omega \) and \( R_F = 50 \, k\Omega \). The input offset voltage is \( v_{io} = \pm 6 \, mV \) at 25°C. Determine the output offset voltage \( V_{oo} \).

14.4 The input biasing current \( I_B \) for the amplifier in Fig. 14.3 is \( I_B = 500 \, nA \) (DC) at 25°C. If \( R_1 = 15 \, k\Omega \) and \( R_F = 50 \, k\Omega \), determine (a) the output offset voltage due to input biasing current \( I_B \) and (b) the offset-minimizing resistance \( R_x \).

14.5 The maximum input offset current of the amplifier in Fig. 14.4 is \( I_{io} = \pm 200 \, nA \) at 25°C. If \( R_1 = 15 \, k\Omega \) and \( R_F = 50 \, k\Omega \), determine the output offset voltage due to the input offset current.

14.6 The inverting amplifier in Fig. 14.5(a) has \( R_1 = 15 \, k\Omega \) and \( R_F = 50 \, k\Omega \). The op-amp has \( V_{io} = \pm 6 \, mV \), \( I_B = 500 \, nA \), and \( I_{io} = \pm 300 \, nA \) at 25°C. Determine the total output offset voltage \( V_{of} \) if (a) \( R_x = R_F = R_1 = 11.54 \, k\Omega \) and (b) \( R_x = 0 \). Assume \( v_S = 0 \).

14.7 The noninverting amplifier in Fig. 14.5(b) has \( R_1 = 15 \, k\Omega \) and \( R_F = 150 \, k\Omega \). The op-amp has \( V_{io} = \pm 6 \, mV \), \( I_B = 500 \, nA \), and \( I_{io} = \pm 300 \, nA \) at 25°C. Determine (a) the total output offset voltage if \( R_x = R_F = R_1 = 11.54 \, k\Omega \) and (b) the output offset voltage \( V_{of} \) if \( R_x = 0 \). Assume \( v_S = 0 \).

14.8 The integrator in Fig. 14.7 has \( R_1 = 10 \, k\Omega \), \( R_x = 10 \, k\Omega \), \( C_1 = 0.1 \, \mu F \), \( V_{CC} = 15 \, V \), \( V_{EE} = -15 \, V \), and maximum saturation voltage \( = \pm 11 \, V \). The op-amp has input offset voltage \( V_{io} = 6 \, mV \), input biasing current \( I_B = 500 \, nA \), and input offset current \( I_{io} = 300 \, nA \) at 25°C.

a. Determine the time required for the op-amp output offset voltage to reach the saturation limit of \( \pm 14 \, V \).

b. Repeat part (a) if \( R_x = 0 \).
14.9 The inverting amplifier in Fig. 14.5(a) has \( R_1 = 15 \, \text{k}\Omega \), \( R_F = 50 \, \text{k}\Omega \), and \( R_S = R_F \parallel R_1 = 11.54 \, \text{k}\Omega \). The op-amp has \( V_{io} = \pm 6 \, \text{mV} \), \( I_B = 500 \, \text{nA} \), and \( I_{io} = \pm 300 \, \text{nA} \). The thermal drifts are \( D_v = 15 \, \text{m}\mu\text{V} \degree \text{C} \), \( D_I = 0.5 \, \text{nA} \degree \text{C} \), and \( D_B = 0.5 \, \text{nA} \degree \text{C} \) at 25°C. The temperature is 55°C. Determine (a) the output offset voltage due to drifts \( V_{od} \) and (b) the total output voltage \( V_{of} \) if the input voltage is \( V_S = 150 \, \text{mV} \) (DC).

14.10 The noninverting amplifier in Fig. 14.5(b) has \( R_1 = 15 \, \text{k}\Omega \), \( R_F = 50 \, \text{k}\Omega \), and \( R_S = R_F \parallel R_1 = 11.54 \, \text{k}\Omega \). The op-amp has \( V_{io} = \pm 6 \, \text{mV} \), \( I_B = 500 \, \text{nA} \), and \( I_{io} = \pm 300 \, \text{nA} \). The thermal drifts are \( D_v = 15 \, \text{m}\mu\text{V} \degree \text{C} \), \( D_I = 0.5 \, \text{nA} \degree \text{C} \), and \( D_B = 0.5 \, \text{nA} \degree \text{C} \) at 25°C. The temperature is 55°C. Determine (a) the output offset voltage due to drifts \( V_{od} \) and (b) the total output voltage \( V_{of} \) if the input voltage is \( V_S = 150 \, \text{mV} \) (DC).

14.11 The inverting amplifier in Fig. 14.5(a) has \( R_1 = 15 \, \text{k}\Omega \) and \( R_F = 150 \, \text{k}\Omega \). The supply voltages change from \( \pm 12 \, \text{V} \) to \( \pm 10 \, \text{V} \), and PSRR = 150 \, \text{mV} \degree \text{V} \) at 25°C. Determine (a) the input offset voltage \( V_{io} \) due to changes in the supply voltages and (b) the corresponding output offset voltage \( V_{oo} \).

14.12 The noninverting amplifier in Fig. 14.5(b) has \( R_1 = 15 \, \text{k}\Omega \) and \( R_F = 50 \, \text{k}\Omega \). The supply voltages change from \( \pm 12 \, \text{V} \) to \( \pm 10 \, \text{V} \), and PSRR = 150 \, \text{mV} \degree \text{V} \) at 25°C. Determine (a) the input offset voltage \( V_{io} \) due to changes in the supply voltages and (b) the corresponding output offset voltage \( V_{oo} \).

14.13 The inverting amplifier in Fig. 14.5(a) has \( R_1 = 10 \, \text{k}\Omega \), \( R_F = 100 \, \text{k}\Omega \), and \( R_S = R_F \parallel R_1 = 9.091 \, \text{k}\Omega \). The op-amp parameters are \( V_{io} = 0.8 \, \text{mV} \), \( I_B = 200 \, \text{nA} \), \( I_{io} = 100 \, \text{nA} \), and PSRR = 150 \, \text{mV} \degree \text{V} \). The drifts are \( D_v = 15 \, \text{m}\mu\text{V} \degree \text{C} \), \( D_I = 0.5 \, \text{nA} \degree \text{C} \), and \( D_B = 0.5 \, \text{nA} \degree \text{C} \) at 25°C. The temperature is 55°C. The DC supply voltages change from \( V_{CC} = 12 \, \text{V} \) to 10 V and \( -V_{EE} = -12 \, \text{V} \) to \(-10 \, \text{V} \). The input voltage is \( V_S = 100 \, \text{mV} \) (DC). Determine the output voltage \( V_O \) if (a) \( R_X = R_F \parallel R_1 = 9.091 \, \text{k}\Omega \) and (b) \( R_X = 0 \).

14.14 Design the offset compensating network. The op-amp parameters are \( V_{io} = 0.8 \, \text{mV} \), \( I_B = 200 \, \text{nA} \), \( I_{io} = 100 \, \text{nA} \), and PSRR = 150 \, \text{mV} \degree \text{V} \). The DC supply voltages are \( V_{CC} = 12 \, \text{V} \) and \( -V_{EE} = -12 \, \text{V} \).

14.15 Design the offset compensating network. The op-amp parameters are \( V_{io} = 0.8 \, \text{mV} \), \( I_B = 200 \, \text{nA} \), \( I_{io} = 100 \, \text{nA} \), and PSRR = 150 \, \text{mV} \degree \text{V} \). The DC supply voltages are \( V_{CC} = 12 \, \text{V} \) and \( -V_{EE} = -12 \, \text{V} \).

14.16 Design the offset compensating network. The op-amp parameters are \( V_{io} = 6 \, \text{mV} \), \( I_B = 500 \, \text{nA} \), \( I_{io} = 200 \, \text{nA} \), and PSRR = 150 \, \text{mV} \degree \text{V} \). The DC supply voltages are \( V_{CC} = 12 \, \text{V} \) and \( -V_{EE} = -12 \, \text{V} \).

14.17 The Darlington pair shown in Fig. 14.13 is biased in such a way that the collector-biasing current \( I_{C2} \) of \( Q_2 \) is 400 \, \mu \text{A} \). The current gains of the two transistors are the same, \( \beta_{FE} = \beta_{FB} = 80 \), and the Early voltage is \( V_A = 50 \, \text{V} \). Calculate (a) the effective input resistance \( r_m \), (b) the effective transconductance \( g_m \), (c) the effective current gain \( \beta_{FE} \), and (d) the effective output resistance \( r_o \).

14.4 CMOS Op-Amps

14.18 The CMOS of the op-amp in Fig. 14.20 has \( K_X = 10 \, \mu \text{A} \) \degree \text{V}^2, W/L = 160 \, \mu \text{m} \times 10 \, \mu \text{m}, \) and \( V_I = 0.5 \, \text{V} \). The biasing current is \( I_Q = 40 \, \mu \text{A} \). If \( V_I \) changes by 2\% and \( W/L \) by 1\%, find the input offset voltage \( V_{OS} \).

14.19 The CMOS amplifier in Fig. 14.20 is operated at a biasing current of \( I_Q = 50 \, \mu \text{A} \). The parameters of the MOSFETs are \( K_X = 10 \, \mu \text{A} \) \degree \text{V}^2, \) \( |V_{MN(MOS)}| = V_{MM(PMOS)} = 60 \, \text{V}, \) \( V_I = 1 \, \text{V} \), and \( W/L = 80 \, \mu \text{m} \times 10 \, \mu \text{m}, \) except for \( Q_7, \) for which \( W/L = 160 \, \mu \text{m} \times 10 \, \mu \text{m}. \) Assume \( V_{DD} = V_{SS} = 5 \, \text{V} \).

a. Find \( V_{GS}, g_m, \) and \( r_o \) for all MOSFETs.

b. Find the low-frequency voltage gain of the amplifier \( A_{vo} \).

c. Find the value of the external resistance \( R_{ref} \).

d. Find the value of compensation capacitance \( C_x \) that gives a unity-gain bandwidth of 1 MHz and the corresponding slew rate.

e. Find the value of resistance \( R_X \) to be connected in series with \( C_x \) in order to move the zero frequency to infinity.
f. Find the common-mode input voltage range.
g. Find the output voltage range.

14.5 BJT Op-Amps

14.20 The BJTs for the op-amp shown in Fig. P14.20 have $\beta_{F(npn)} = 100$, $\beta_{F(pnp)} = 50$, $V_{A(npn)} = V_{A(pnp)} = 80$ V, $V_{BE} = 0.6$ V, and $I_S = 10^{-14}$ A. Find (a) the input biasing current $I_B$, (b) the input resistance $R_i$, (c) the unity-gain bandwidth $f_u$ and the slew rate, and (d) the overall low-frequency voltage gain $A_{vo}$. Assume $V_{CC} = -V_{EE} = 15$ V.

**FIGURE P14.20**

14.21 The BJTs for the op-amp shown in Fig. P14.21 have $\beta_{F(npn)} = 100$, $\beta_{F(pnp)} = 50$, $V_{A(npn)} = V_{A(pnp)} = 80$ V, $V_{BE} = 0.6$ V, and $I_S = 10^{-14}$ A. Find (a) the input biasing current $I_B$, (b) the input resistance $R_i$, and (c) the overall low-frequency voltage gain $A_{vo}$. Assume $V_{CC} = V_{EE} = 15$ V.

**FIGURE P14.21**
14.22 The BJTs for the LM124 op-amp shown in Fig. 14.22 have $\beta_{F(npn)} = 100$, $\beta_{F(pnp)} = 50$, $V_{A(npn)} = V_{A(pnp)} = 80 \text{ V}$, $V_{BE} = 0.6 \text{ V}$, and $I_S = 10^{-14} \text{ A}$. Assume $V_{CC} = 12 \text{ V}$. Find (a) the input biasing current $I_B$, (b) the input resistance $R_I$, (c) the unity-gain bandwidth $f_u$ and the slew rate, (d) the overall low-frequency voltage gain $A_{vo}$, and (e) the value of $R_{SC}$ for a short-circuit current limit of 25 mA.

14.6 Analysis of the LM741 Op-Amp

14.23 Calculate the gain of the LM741 op-amp in Fig. 14.23 if $R_S$ is reduced from 100 $\Omega$ to 0.

14.24 Calculate the gain of the LM741 op-amp in Fig. 14.23 if the current gain for npn transistors is changed from $\beta_F = 250$ to $1.1 \times 250 = 275.0$ and the saturation current is changed from $I_S = 10^{-14} \text{ A}$ to $1.05 \times 10^{-14} \text{ A}$.

14.25 Calculate the gain of the LM741 op-amp in Fig. 14.23 if all resistances are increased by 1%.

14.26 Calculate the gain of the LM741 op-amp in Fig. 14.23 if all resistances are decreased by 1%.

14.7 BiCMOS Op-Amps

14.27 The BiCMOS op-amp in Fig. 14.40 has $I_{DDS} = 500 \mu \text{A}$ and $V_p = -4 \text{ V}$. The biasing current is $I_Q = 200 \mu \text{A}$.

a. If the input biasing current is $I_B = 1.0 \text{ nA}$ at $25^\circ \text{C}$ and doubles for every $10^\circ \text{C}$ temperature rise, find the input biasing current $I_B$ at $75^\circ \text{C}$, $100^\circ \text{C}$, and $125^\circ \text{C}$.

b. If $I_{DDS}$ changes by 2%, find the input offset voltage $V_{OS}$ and the thermal drift $D_v$.

14.28 The biasing current for the op-amp in Fig. 14.40 is $I_Q = 200 \mu \text{A}$, and the compensation capacitance is $C_x = 30 \text{ pF}$. Find (a) the slew rate SR and (b) the unity-gain bandwidth $f_u$. 
CHAPTER 15

INTRODUCTION TO DIGITAL ELECTRONICS

Learning Outcomes
After completing this chapter, students should be able to do the following:

- Describe the definition of logic states.
- List the performance parameters of logic gates.
- List the logic families and describe their internal circuitry.
- List the relative advantages and disadvantages of logic families.
- Design and analyze simple logic gates.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$I_{IL}, I_{IH}$</td>
<td>Input low and input high currents of a logic gate</td>
</tr>
<tr>
<td>$I_{OL}, I_{OH}$</td>
<td>Output low and output high currents of a logic gate</td>
</tr>
<tr>
<td>$I_{Bn}, I_{Cn}, I_{En}$</td>
<td>Base, collector, and emitter currents of an $n$th BJT</td>
</tr>
<tr>
<td>$i_{Dn}, v_{GSn}$</td>
<td>Drain current and gate–source voltage of an $n$th MOSFET</td>
</tr>
<tr>
<td>$i_{DL}, v_{DSL}$</td>
<td>Drain current and drain–source voltage of a load MOSFET</td>
</tr>
<tr>
<td>$K_L, K_n$</td>
<td>Constants of a load MOSFET and a driver NMOS of a logic gate</td>
</tr>
</tbody>
</table>
15.1 Introduction

If we observe the output (v-i) characteristic of a transistor closely, we notice that there are two distinct regions: a low-resistance region and a high-resistance region. In analog electronics, transistors are operated in the active region as amplifying devices, so they exhibit the characteristic of a high output resistance. However, with proper biasing conditions, transistors can also exhibit the characteristic of a low output resistance—that is, a low output voltage. In digital electronics, transistors are operated as on (low output) and off switches.

15.2 Logic States

Electronic circuits used to perform logic functions are known as digital logic circuits or logic gates. They use two binary variables, or states: 0 (low) and 1 (high). The binary states are normally represented by two distinct voltages: voltage $V_H$ for logic 1 and voltage $V_L$ for logic 0. If the logic 1 voltage is higher than the logic 0 voltage (e.g., $V_H = 5$ and $V_L = 0$), the circuit is said to use positive logic. If the logic 1 voltage is lower than the logic 0 voltage (e.g., $V_H = -5$ and $V_L = 0$), the circuit is said to use negative logic. In this chapter, we consider positive logic.

To accommodate variations in component tolerances, temperature, and noise in a logic circuit, two voltage ranges are usually used to define the two logic states. The ranges are illustrated in Fig. 15.1, in which $V_{H1}$ is the lowest voltage that will always be recognized as logic 1 and $V_{L2}$ is the highest voltage that will always be recognized as logic 0. If the voltage lies in the range $V_{H2}$ to $V_{H1}$, the state of the digital circuit is interpreted as logic 1. If the voltage lies in the range $V_{L2}$ to $V_{L1}$, the state of the digital circuit is interpreted as logic 0. The two voltage regions are separated by an undefined, or excluded, region. This is a forbidden band, and the signal voltage is not permitted to lie in this region. The difference $V_{H1} - V_{L2}$ is called the transition region.

### Symbol

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$N$</td>
<td>Fan-out of a logic gate</td>
</tr>
<tr>
<td>$N_{L1}, N_{M1}$</td>
<td>Low logic–noise margin and high logic–noise margin of a logic gate</td>
</tr>
<tr>
<td>$P_{on}, P_{off}, P_{static}$</td>
<td>On-state, off-state, and static low of a logic gate</td>
</tr>
<tr>
<td>$R_{dN}, R_{dp}$</td>
<td>Drain–source resistances of an NMOS and a PMOS</td>
</tr>
<tr>
<td>$t_{pd0}$ or $t_{pLH}$</td>
<td>Propagation delay time from low to high</td>
</tr>
<tr>
<td>$t_{pd1}$ or $t_{pHL}$</td>
<td>Propagation delay time from high to low</td>
</tr>
<tr>
<td>$V_I, V_O$</td>
<td>Input and output voltages of a logic gate</td>
</tr>
<tr>
<td>$V_{IL}, V_{ID}$</td>
<td>Threshold voltages of load and driver MOSFETs of a logic gate</td>
</tr>
<tr>
<td>$V_{IL}, V_{IH}$</td>
<td>Input low and input high voltages of a logic gate</td>
</tr>
<tr>
<td>$V_{OL}, V_{OH}$</td>
<td>Output low and output high voltages of a logic gate</td>
</tr>
<tr>
<td>$V_{TW}, V_{LS}$</td>
<td>Transition width and logic swing voltages of a logic gate</td>
</tr>
</tbody>
</table>
The commonly used logic gates are NOT (inverter), AND, NAND, OR, and NOR. Consider a voltage-controlled switch with a resistance, as shown in Fig. 15.2(a), which is controlled by the input signal \( v_I \) applied between terminals 1 and 2. The output is taken across the switch between terminal 3 and the ground terminal 2. When \( v_I \) is low (around 0), the switch is open, and the output voltage \( v_O \) is high (equal to the DC supply voltage \( V_{CC} \)). When \( v_I \) is high enough to close the switch (generally above a specified threshold voltage), the switch closes, and the output voltage \( v_O \) is low (around 0). The input and output voltages are shown in Fig. 15.2(b). The output is the logical inversion of the input signal, and this circuit is known as an inverter or a \textit{NOT} gate. Inverters are the building blocks of logic gates.

**FIGURE 15.2**  Inverter
The switch can be realized by either a bipolar transistor, as shown in Fig. 15.2(c), or a MOS transistor, as shown in Fig. 15.2(d). Transistors $M_1$ and $Q_1$ are switched between two states: nonconducting (or off) and conducting (or on). $R_P$ is called the pull-up resistance because the output is pulled up toward the positive supply voltage $V_{CC}$ or $V_{DD}$ when the switching transistor is off.

An inverter has only one input voltage. Logic gates usually combine one or more logic variable inputs to produce an output. All of the possible combinations of the input variables and the corresponding outputs are normally listed in a table called a truth table. The truth table for NOT, AND, NAND, OR, and NOR is shown in Table 15.1. If both inputs are high, the output becomes high in an AND gate and low in a NAND gate. If at least one input is high, the output becomes high in an OR gate and low in a NOR gate. The symbols for these logic gates are shown in Fig. 15.3.

Digital circuits are available exclusively in integrated circuits (ICs) and can be classified into families. Each member of a family is made with the same technology, has similar structure, and exhibits the same basic features. There are two MOS-logic families (NMOS, using only $n$-channel MOSFETs, and CMOS, using both $n$- and $p$-channel MOSFETs in a complementary configuration), and two BJT families, (a transistor-transistor logic [TTL] family and an emitter-coupled logic [ECL] family). Each of these families has unique advantages and disadvantages. The choice of a logic family is based on considerations such as logic functions, logic flexibility, speed, noise immunity, operating temperature range, power dissipation, and cost.

MOS transistor logic circuits have advantages over bipolar logic gates because MOSFETs are simpler to fabricate and occupy less space in integrated form than BJTs. A MOSFET can be connected to act as a resistive load to replace a diffused resistor (a resistor made by diffusion during the IC-manufacturing process) in bipolar integrated circuits. The packing density of MOSFETs is extremely high, and MOSFET circuits can be fabricated for LSI, VLSI, and ULSI applications. MOSFETs and metal–Schottky barrier FETs (MESFETs) can also be used in digital integrated circuits. MESFETs are usually fabricated of gallium arsenide (GaAs), whose electron mobility is higher than that of silicon. MESFET circuits are faster than other types of FET circuits and are known for their outstanding speed capabilities.

### Table 15.1 Truth table for NOT, AND, NAND, OR, and NOR

<table>
<thead>
<tr>
<th>Input</th>
<th>NOT</th>
<th>Input</th>
<th>Input</th>
<th>AND</th>
<th>NAND</th>
<th>OR</th>
<th>NOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>$C = \overline{A}$</td>
<td>$A$</td>
<td>$B$</td>
<td>$C = AB$</td>
<td>$C = \overline{AB}$</td>
<td>$C = A + B$</td>
<td>$C = A + \overline{B}$</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>1</td>
<td>0</td>
</tr>
</tbody>
</table>

*FIGURE 15.3* Symbols for logic gates
Depending on the complexity of the internal circuitry of the IC chip, digital IC packages can be classified by degree of integration, as shown in Table 15.2.

### Table 15.2: Classification of digital integrated circuits

<table>
<thead>
<tr>
<th>Degree of Integration</th>
<th>Number of Gates</th>
</tr>
</thead>
<tbody>
<tr>
<td>Small-scale integration (SSI)</td>
<td>Fewer than 10</td>
</tr>
<tr>
<td>Medium-scale integration (MSI)</td>
<td>From 10 to 100</td>
</tr>
<tr>
<td>Large-scale integration (LSI)</td>
<td>From 100 to 1000</td>
</tr>
<tr>
<td>Very-large-scale integration (VLSI)</td>
<td>From 1000 to $10^5$</td>
</tr>
<tr>
<td>Ultra-large-scale integration (ULSI)</td>
<td>More than $10^5$</td>
</tr>
</tbody>
</table>

**KEY POINT OF SECTION 15.3**

- The commonly used logic gates are NOT (inverter), AND, NAND, OR, and NOR. Inverters are the building blocks of logic gates. There are many families of logic gates, among them NMOS, CMOS, TTL, and ECL gates.

### 15.4 Performance Parameters of Logic Gates

Figure 15.2(b) shows the characteristics of an ideal inverter [1–3]. However, the performance of actual inverters differs significantly from the ideal in the following ways:

1. The switch is not ideal; that is, when the switch is closed, it has a finite voltage drop rather than a short circuit.
2. The switch may not open and close instantaneously because of a time delay between the application of the input signal and the propagation of the desired output signal.
3. The input terminal of the inverter usually draws some current from the driving source.
4. The inverter usually drives a load or acts as a source to the next stage(s), and it also should be capable of supplying the driving current.

Manufacturers’ data sheets on logic gates specify many performance parameters. As examples, typical data for TTL and CMOS devices are shown in Table 15.3, where $t_{pd0}$ (also abbreviated as $t_{pHL}$) is the propagation delay time from low to high and $t_{pd1}$ (also abbreviated as $t_{pLH}$) is the propagation delay time from high to low.

#### Table 15.3: Parameters of 54L 74L TTL and 54C 74C CMOS logic gates

<table>
<thead>
<tr>
<th>Family</th>
<th>$V_{CC}$ max (mV)</th>
<th>$I_{IL}$ (mA) max</th>
<th>$V_{IH}$ (mV) min</th>
<th>$I_{IH}$ (μA) max</th>
<th>$V_{OL}$ (mV) min</th>
<th>$I_{OL}$ (μA)</th>
<th>$V_{OH}$ (mV)</th>
<th>$I_{OH}$ (μA)</th>
<th>$t_{pd0}$ (ns)</th>
<th>$t_{pd1}$ (ns)</th>
<th>$P_{D}$ (mW)</th>
<th>$P_{D}$ (1 MHz, 50 pF)</th>
</tr>
</thead>
<tbody>
<tr>
<td>54L 74L</td>
<td>5</td>
<td>0.7</td>
<td>0.18</td>
<td>2.0</td>
<td>10</td>
<td>0.3</td>
<td>2000</td>
<td>2.4</td>
<td>100</td>
<td>31</td>
<td>35</td>
<td>1000</td>
</tr>
<tr>
<td>54C 74C</td>
<td>5</td>
<td>0.8</td>
<td>—</td>
<td>3.5</td>
<td>1.4</td>
<td>0.4</td>
<td>360</td>
<td>2.4</td>
<td>100</td>
<td>60</td>
<td>45</td>
<td>0.01</td>
</tr>
<tr>
<td>54C 74C</td>
<td>10</td>
<td>2.0</td>
<td>—</td>
<td>8.0</td>
<td>1.0</td>
<td>1.0</td>
<td>10</td>
<td>8.0</td>
<td>10</td>
<td>25</td>
<td>30</td>
<td>0.03</td>
</tr>
</tbody>
</table>
15.4.1 Voltage Transfer Characteristic

The voltage transfer characteristic (VTC) gives the relationship between the input voltage \( v_I \) and the output voltage \( v_O \). An inverter with a single power supply \( V_{CC} \) and its VTC are shown in Figs. 15.4(a) and 15.4(b), respectively. The inverter has a threshold voltage of \( V_t = V_{CC}/2 \); that is, the output is high (at \( v_O = V_{CC} \)) for \( v_I < V_{CC}/2 \) and low (at \( v_O = 0 \)) for \( v_I > V_{CC}/2 \). The transition from low to high and vice versa is very sharp at \( v_I = V_{CC}/2 \). Thus, the incremental voltage gain is \( dv_O/dv_I = 0 \) for \( v_I < V_{CC}/2 \) and it is \( dv_O/dv_I = \infty \) for \( v_I = V_{CC}/2 \). Therefore, the inverter exhibits a nonlinear characteristic. If it is connected with feedback, it will oscillate between the low and high states.

A practical inverter does not have a finite threshold voltage; rather, it goes through a transition region from the high state to the low state. The VTC of a practical inverter is shown in Fig. 15.5; the VTC has three distinct regions: the low-input region, \( v_I < V_{IL} \); the transition region, \( V_{IL} \leq v_I \leq V_{IH} \); and the high-input region, \( v_I > V_{IH} \).

The VTC has two transitions, one at \( v_I = V_{IL} \) and one at \( v_I = V_{IH} \), with two corresponding output voltages, \( V_{OH} \) and \( V_{OL} \). The transition voltages are defined as the points at which the slope of the VTC is \(-1\) (that is, \( dv_O/dv_I = -1 \)). The following variables are applicable to all logic circuits:

- \( V_{OH} \) (high-level output voltage) is the minimum output voltage that will establish a high level (logic 1). Data sheets guarantee that the output voltage will exceed this level at all specified operating conditions.
- \( V_{OL} \) (low-level output voltage) is the maximum output voltage that will establish a low level (logic 0). Data sheets guarantee that the output voltage will not exceed this level at all specified operating conditions.
$V_{IL}$ (low-level input voltage) is the maximum positive voltage that can be applied to an input terminal of a gate and still be recognized as logic low (0).

$V_{IH}$ (high-level input voltage) is the minimum positive voltage that can be applied to an input terminal of a gate before the transition starts from the logic high (1) to logic low (0).

In the transition region, the output is undefined. The width of the transition region is a measure of ambiguity, and it is defined by

$$V_{TW} = V_{IH} - V_{IL} \quad (15.1)$$

A low value of $V_{TW}$ is desirable to reduce ambiguity in the input logic state. *Logic swing* is also a measure of the ambiguity in the logic state, and it is defined by

$$V_{LS} = V_{OH} - V_{OL} \quad (15.2)$$

A high value of $V_{LS}$ is desirable to reduce ambiguity and increase noise immunity. A gate for which $V_{OH} = 2.4 \, \text{V}$, $V_{OL} = 0.3 \, \text{V}$, $V_{IH} = 2 \, \text{V}$, and $V_{IL} = 0.7 \, \text{V}$ has

$$V_{TW} = V_{IH} - V_{IL} = 2 - 0.7 = 1.3 \, \text{V}$$

$$V_{LS} = V_{OH} - V_{OL} = 2.4 - 0.3 = 2.1 \, \text{V}$$

### 15.4.2 Noise Margins

*Noise* generally is present in logic circuits, superimposed on input signals. Noise simply refers to extraneous signals, which may arise from inadequate regulation or decoupling of the power supply, electromagnetic radiation, inductive or capacitive coupling from other parts of the system, or line drops. One of the advantages of logic circuits is their tolerance for variations in the input signal, which results from the fact that the input signal is interpreted simply as high or low. *Noise immunity* is a measure of the tolerance for variations in the signal level, and it is that voltage that, applied to the input, will cause the output to change its state. Noise immunity is an important device characteristic. However, *noise margin* is of more use to the designer; it defines the amount of noise a system can tolerate under any circumstances and still maintain the integrity of the logic levels. That is, noise margin measures the ability of a gate to maintain its logic state under varying voltage levels.

In digital circuits, one gate usually drives another; that is, the output of the first gate is the input to the following gate. Thus, the gate whose output is high at $V_{OH}$ will drive an identical gate whose high-level input voltage is $V_{IH}$. This concept is illustrated in Fig. 15.6(a). The difference $V_{OH} - V_{IH}$ represents a margin of safety at the logic high output. It is called the *logic 1, or high, noise margin* and is given by

$$\text{NM}_{\text{H}} = V_{OH} - V_{IH} \quad (15.3)$$
Similarly, the noise margin at the logic low output is called the logic 0, or low, noise margin and is denoted by

\[ N_{ML} = V_{IL} - V_{OL} \]  \hspace{1cm} (15.4)

Therefore, the noise margin is the difference between the guaranteed logic 1 (or 0) level output voltage and the guaranteed logic 1 (or 0) level input voltage. The absolute noise margin is the smaller of the two noise margins; that is,

\[ N_M = \min(N_{ML}, N_{MH}) \]  \hspace{1cm} (15.5)

The noise margins are shown in Fig. 15.6(b). A gate with a high logic swing \( V_{LS} \) and a small transition width \( V_{TW} \) will have good noise immunity. There will be no uncertain or undefined region if \( V_{IL} = V_{IH} = (V_{OH} + V_{OL})/2 \), and the noise margins will be equal: \( N_{ML} = N_{MH} \). However, this will cause the transition region to be eliminated with an abrupt switch in the ideal VTC, as shown in Fig. 15.4(b). For example, a gate for which \( V_{OH} = 2.5 \) V, \( V_{OL} = 0.3 \) V, \( V_{IH} = 2 \) V, and \( V_{IL} = 0.7 \) V has

\[ N_{MH} = V_{OH} - V_{IH} = 2.5 - 2 = 0.5 \text{ V} \]
\[ N_{ML} = V_{IL} - V_{OL} = 0.7 - 0.3 = 0.4 \text{ V} \]
\[ N_M = \min(N_{ML}, N_{MH}) = \min(0.4, 0.5) = 0.4 \text{ V} \]

### 15.4.3 Fan-Out and Fan-In

A gate draws an input current from the input signal and also delivers current to the load gate(s). Note that the current flowing out of a terminal will have a negative value. The gate acts as the load for the
input signal and as the driver for the load gates. Just as there are four voltages, there are four currents associated with a gate and its load, and they are defined as follows:

$I_{OH}$ (high-level output current) is the current flowing into the output terminal when it is in the high state (logic 1).

$I_{OL}$ (low-level output current) is the current flowing into the output terminal when it is in the low state (logic 0).

$I_{IL}$ (low-level input current) is the current flowing into the input terminal when a specified low-level voltage (logic 0) is applied to the input.

$I_{IH}$ (high-level input current) is the current flowing into the input terminal when a specified high-level voltage (logic 1) is applied to the input.

A gate must be capable of accepting more than one input. The number of independent input nodes is known as the fan-in. The output of one gate must be capable of driving more than one input of subsequent gates. The number $N$ of inputs that can be driven by a gate is known as the fan-out. Fan-out is illustrated in Fig. 15.7. More precisely, fan-out is defined as the maximum number of load gates of similar design that can be connected to the output of a logic gate (i.e., the driver gate) without changing its logic state. Since logic gates draw a different amount of current in the logic low and logic high states, the fan-out is the smaller of the numbers for the logic low and logic high states. That is, fan-out $N$ is equal to either $I_{OL}$, $I_{IL}$, or $I_{OH}$, whichever gives the lower natural number. $N$ is given by

$$N = \min \left( \frac{I_{OL}}{I_{IL}}, \frac{I_{OH}}{I_{IH}} \right) \quad (15.6)$$

For example, if $I_{OL} = 2 \text{ mA}$, $I_{IL} = 0.1 \text{ mA}$, $I_{OH} = 100 \mu \text{A}$, and $I_{IH} = 10 \mu \text{A}$,

$$N = \min \left[ \left( \frac{2 \text{ mA}}{0.1 \text{ mA}} \right), \left( \frac{100 \mu \text{A}}{10 \mu \text{A}} \right) \right] = \min(20,10) = 10$$

The fan-out of a logic gate is usually quoted in terms of the number of inverters the gate can drive. This helps in comparing gates and determining the loading effects of the gate in a digital circuit with multiple gates of the same family.

![Figure 15.7](image-url)
EXAMPLE 15.1

**Designing a simple inverter** An inverter, as shown in Fig. 15.8, drives identical inverters and has $V_{OL} = 0.3 \text{ V}$, $I_{OL} = 2 \text{ mA}$, $V_{OH} = 2.4 \text{ V}$, $I_{OH} = 100 \text{ \mu A}$, and $V_{CC} = 5 \text{ V}$. The input currents drawn by each load inverter are $I_{IH} = 0.18 \text{ mA}$ (at logic high) and $I_{IL} = 10 \text{ \mu A}$ (at logic low).

(a) If there are five load inverters, determine the value of pull-up resistance $R_P$ that will ensure a logic 1 output of $V_{OH} = 2.4 \text{ V}$.

(b) If $R_P = 4 \text{ k}\Omega$, find the fan-out $N$.

**SOLUTION**

(a) At high output, all load inverters are connected to the driving inverter, and each draws 0.18 mA. Because of the resistance $R_P$, the output voltage will be lower than $V_{CC}$. Thus,

$$v_O = V_{CC} - R_P(N_{IH} + I_{OH})$$

To ensure that $v_O \geq V_{OH} = 2.4 \text{ V}$, we calculate the value of $R_P$ (for $N = 5$) as

$$R_P = \frac{V_{CC} - V_{OH}}{N_{IH} + I_{OH}} = \frac{(5 - 2.4) \text{ V}}{5 \times 0.18 \text{ mA} + 100 \text{ \mu A}} = 2.6 \text{ k}\Omega$$

At low output, each load inverter draws $I_{IL} = 10 \text{ \mu A}$. Thus,

$$V_{CC} = V_{OL} + R_P(I_{OL} + N_{IL})$$

To ensure that $R_P \geq I_{OL} = 2 \text{ mA}$, we calculate the value of $R_P$ as

$$R_P = \frac{V_{CC} - V_{OL}}{I_{OL} + N_{IL}} = \frac{(5 - 0.3) \text{ V}}{2 \text{ mA} + 5 \times 10 \text{ \mu A}} = 2.29 \text{ k}\Omega$$

**FIGURE 15.8** Logic high fan-out of an inverter
15.4.4 Propagation Delay

A switching device such as a bipolar transistor exhibits junction capacitances. As a result, the output of an inverter may not respond instantaneously to the input signal. In addition, the load gate(s) offers a certain amount of capacitance $C_L$ to the driving inverter, as shown Fig. 15.9(a). $C_L$ is the equivalent input capacitance of the load gate(s), including any capacitance due to the wiring connection. Therefore, the input and output responses of the inverter will exhibit finite rise time $t_r$ and fall time $t_f$, as shown in Fig. 15.9(b). The rise time $t_r$ is the time required for the waveform to rise from 10% to 90% of its final (high) value. Similarly, the fall time $t_f$ is the time required for the waveform to decrease from 90% to 10% of its final (low) value.

The speed of operation of a gate depends on how fast a change in the input propagates through it and causes a change at the output. There will be a delay time between the input and output waveform; this time is commonly known as the propagation delay time $t_{pd}$. The propagation delay time is defined as the time between when the input pulse waveform is at 50% of its logic high value and when the corresponding output pulse waveform is at 50% of its logic high value. For two edges (i.e., falling and rising), there are two delay times, denoted as $t_{pd1}$ (or $t_{pHL}$) for the high-to-low logic and $t_{pd0}$ (or $t_{pLH}$) for the low-to-high logic. The average of $t_{pd1}$ and $t_{pd0}$ is the average propagation time $t_{pd}$:

$$t_{pd} = \frac{t_{pd1} + t_{pd0}}{2}$$

(15.7)

This value is commonly used as a figure of merit to compare the performance of different logic families. For hand calculations of the propagation delays, the input voltage can be assumed to be ideal, as shown in Fig. 15.9(c). Typical values of $t_{pd}$ range from 0.5 ns to 10 ns. Cycle time $t_{cyc}$, another parameter used to compare the performance of logic families, is the time between identical points of successive cycles in a signal waveform. The clock frequency $f_{clk}$, which is the reciprocal of the cycle time, is more often used. Practical digital systems are usually designed to operate with a cycle time 20 to 50 times the propagation time of a single gate.
15.4.5 Power Dissipation

The amount of power \( (P_D) \) consumed by a digital circuit is also an important parameter. Knowing this parameter enables the designer to determine the amount of current that will be drawn from the power supply. The power dissipation has static and dynamic components. As an example, consider the inverter in Fig. 15.10(a), where \( C_L \) is the load capacitance (usually the input capacitance of another gate or the wiring capacitance or internal capacitance of the switching device itself).

![Propagation times](image-url)
Static Power

The power consumed by an inverter depends on its logic state. When the switch \( S_1 \) is closed at logic 0, the current is drawn from the supply. The power delivered by the supply is called the static, or quiescent, power. Thus, the static power at logic 0 is given by

\[
P_{\text{on}} = \frac{V_{\text{CC}}^2}{R_p + R_{\text{on}}}
\]  

(15.8)

where \( R_{\text{on}} \) is the on-state switch resistance, whose value is usually low, and \( R_{\text{on}} \ll R_p \). When the switch is open at logic 1, a small leakage current \( I_{\text{leak}} \) flows through the switch. The static power at logic 1 is given by

\[
P_{\text{off}} = \frac{V_{\text{CC}}^2}{R_p + R_{\text{off}}}
\]  

(15.9)

where \( R_{\text{off}} \) is the off-state switch resistance, whose value is usually very large, and \( R_{\text{off}} \gg R_p \). Thus, for the idealized inverter, \( P_{\text{on}} \approx V_{\text{CC}}^2 / R_p \) and \( P_{\text{off}} \approx 0 \). The static power is usually expressed as an average value. Assuming that a gate, on average, spends half the time in each state, the average static power dissipation becomes

\[
P_{\text{static}} = \frac{1}{2} (P_{\text{on}} + P_{\text{off}})
\]  

(15.10)

\[
\approx \frac{V_{\text{CC}}^2}{2R_p} \quad \text{(for } P_{\text{on}} \approx \frac{V_{\text{CC}}^2}{R_p} \text{ and } P_{\text{off}} \approx 0 \text{)}
\]  

(15.11)

Dynamic Power

An inverter also consumes power each time it changes state. Let us assume that at \( t = 0^− \), the input is high and the switch is closed. Thus, the capacitor \( C_L \) is discharged, and it has no charge, as shown in Fig. 15.10(a). When the switch is opened at \( t = 0^+ \), the capacitor will charge exponentially to the supply voltage \( V_{\text{CC}} \) (approximately) through \( R_p \), as shown in Fig. 15.10(b). The charging current will also flow through \( R_p \), and thus power will be dissipated in \( R_p \).
The charge stored on the capacitor is given by

\[ Q = C_L V_{CC} \]  \hspace{1cm} (15.12)

and the energy drawn from the supply is given by

\[ E = QV_{CC} = C_L V_{CC}^2 \]  \hspace{1cm} (15.13)

Half of this energy is dissipated in the resistance \( R_P \), and the other half is stored in the capacitor as \((1/2)C_L V_{CC}^2\). The next time the input becomes high, the switch is closed, as shown in Fig. 15.10(c), and the capacitor discharges through the switch resistance \( R_{on} \). That is, the energy stored in the capacitor is dissipated as heat in \( R_{on} \). Therefore, every time the capacitor \( C_L \) is charged or discharged, an amount of energy must be provided by the power supply. The energy per cycle is \( C_L V_{CC}^2 \). Since energy per unit time is the power, the \textit{dynamic power dissipation} is given by

\[ P_{\text{dynamic}} = f_{\text{clk}} C_L V_{CC}^2 \]  \hspace{1cm} (15.14)

where \( f_{\text{clk}} \) is the clock frequency of the inverter in hertz. Therefore, the total power that must be supplied by the power supply is given by

\[ P_D = P_{\text{static}} + P_{\text{dynamic}} \]  \hspace{1cm} (15.15)

Since \( P_D \) is dependent on the clock frequency, power dissipation can be a severe problem in digital circuits with frequencies over 100 MHz.

\[ E = QV_{CC} = C_L V_{CC}^2 \]

\[ Q = C_L V_{CC} \]

\[ P_{\text{dynamic}} = f_{\text{clk}} C_L V_{CC}^2 \]

\[ P_D = P_{\text{static}} + P_{\text{dynamic}} \]

\[ \text{EXAMPLE 15.2} \]

\textbf{Finding the delay times and power dissipation of an inverter}  

The inverter shown in Fig. 15.10(a) has \( V_{OL} = 0.3 \text{ V}, \: V_{OH} = 2.4 \text{ V}, \: V_{CC} = 5 \text{ V}, \: R_{on} = 500 \text{ } \Omega, \: R_{off} \approx \infty, \: R_P = 2.6 \text{ k}\Omega, \: C_L = 5 \text{ pF}, \) and \( f_{\text{clk}} = 10 \text{ MHz}. \)

\textbf{(a)} Find the delay times for the output voltage to rise from 0.3 V to 2.4 V (\( t_{\text{pd0}} \)) and to fall from 5 V to 0.3 V (\( t_{\text{pd1}} \)).

\textbf{(b)} Find the power dissipation \( P_D \).

\textbf{SOLUTION}

\textbf{(a)} When the switch is open, the capacitor \( C_L \) charges exponentially from 0.3 V to 5 V. The output voltage \( v_O(t) \) can be expressed in the general form

\[ v_O(t) = v_O(t = \infty) + [v_O(t = 0) - v_O(t = \infty)]e^{-t/\tau} \]  \hspace{1cm} (15.16)

For \( v_O(t = 0) = 0.3 \text{ V} \) and \( v_O(t = \infty) = 5 \text{ V} \), Eq. (15.16) becomes

\[ v_O(t) = 5 - 4.7e^{-t/\tau} \]
15.4.6 Delay-Power Product

It is desirable to have both a low propagation delay time (high speed) and low power dissipation. However, these two requirements conflict with each other. For example, if the power dissipation is reduced by decreasing the supply current, the delay will increase. The delay-power product \(DP\) is the product of average propagation delay time \(t_{pd}\) and power dissipation \(P_D\):

\[
DP = t_{pd}P_D
\]  
(15.17)

DP is a figure of merit for comparing logic gates. A small value of DP indicates that a circuit has a fast switching speed and dissipates very little power. In practice, a trade-off between power dissipation and switching speed is usually required. Typical values of DP range from 5 pJ to 50 pJ.

**KEY POINT OF SECTION 15.4**

- The performance specifications of a logic gate normally include a description of the voltage transfer characteristic, noise margins, fan-out, fan-in, propagation delay, power dissipation, and delay-power product.
15.5 NMOS Inverters

NMOS inverters are the building blocks of NMOS digital circuits [4–6]. They use only $n$-channel MOSFETs, which have low channel resistance because of the greater mobility of electrons in an $n$-channel. If all the MOSFETs are enhancement-type devices, the fabrication procedure becomes simpler. But combining an enhancement-type device (as the driver) and a depletion-type device (as the load) increases the switching speed. An enhancement-type transistor is used as the driver because (1) it will be off when the input voltage $v_I$ is low and (2) its drain and gate voltages have the same polarity. This feature allows direct coupling between stages (i.e., one stage can be connected to the next one without any coupling capacitor). Although depletion-load inverters are generally used in integrated circuits for high-speed switching, we will analyze inverters with both enhancement and depletion MOSFETs.

15.5.1 NMOS Inverter with Enhancement Load

In ICs, an inverter uses a transistor as the load because it requires much less chip area (typically 50 times less than for a diffused resistor with the same value). The resistor in Fig. 15.2(a) can be replaced by a MOSFET.

Enhancement Load

Consider an enhancement-type $n$-channel MOSFET (NMOS) whose gate is connected to the drain terminal, as shown in Fig. 15.11(a). For $v_{GS,L} = v_{DS,L} \leq V_{tL}$ (threshold voltage), the drain current will be zero; that is,

$$i_{DL} = 0$$

$$v_{DS,L} = v_{GS,L}$$

where $L$ in the subscript after $D$, $DS$, and $GS$ refers to the load MOSFET.

**FIGURE 15.11**  $n$-channel enhancement MOSFET as a load with $v_{GS} = v_{DS}$
For $v_{GSL} = v_{DSL} > V_{tL}$, a drain current will flow and the transistor will always operate in the saturation (pinch-down) mode under the following condition:

$$v_{DS} > (v_{GSL} - V_{tL}) = v_{DSL} - V_{tL} = V_{DS(sat)}$$

The corresponding drain current is given by

$$i_{DL} = K_{L}(v_{GSL} - V_{tL})^2$$  \hspace{1cm} (15.18)

where $K_{L}$ is the MOS constant and $V_{tL}$ is the threshold voltage of the load MOSFET. The output characteristics, which are shown in Fig. 15.11(b), indicate that the transistor will act as a nonlinear resistor if operated in the saturation region.

**Static Characteristics**

An NMOS inverter with an enhancement load is shown in Fig. 15.12(a). The substrates of the MOSFETs (MD and ML) are connected to the ground. The substrate in an IC logic gate is common to all devices, and therefore it is connected to the ground (or to the most negative potential). Consequently, there is a nonzero source-to-body voltage $V_{SB}$ (which varies with logic levels) for the device. There is also a body-to-source capacitance, which influences both the frequency and the transient response of the device. However, this effect, called the body effect, is not present if the substrate (i.e., the body) is connected directly to the source. To simplify the analysis, we will assume that the substrates are connected to the source terminals so that $V_{SB} = 0$ for both MD and ML.

The output characteristics of the load and driver transistors are shown in Fig. 15.12(b). Before the application of high input, $v_{I} = V_{OL}$ and $v_{O} = V_{OH}$. The inverter is operating at point A. When $v_{I}$ goes high to $V_{OH}$, transistor MD turns on, and the operating point jumps from A to B and then moves along the $i$-$v$ curve of MD until it finally reaches the quiescent point Q so that $v_{O} = V_{OL}$. When $v_{I}$ goes from high to low to $V_{OL}$, the operating point jumps from Q to A and then moves along the $i$-$v$ curve of ML until it finally reaches $v_{O} = V_{OH}$. Depending on the input voltage $v_{I}$, the circuit operation can be divided into three regions, as shown in Fig. 15.12(c): region I, region II, and region III.

**FIGURE 15.12** NMOS inverter with enhancement load
In region I, the input voltage $v_I \leq V_{tD}$, where $V_{tD}$ is the threshold voltage of the drive transistor $M_D$. Thus, $M_D$ is cut off, so the drain currents of the two transistors must be zero. For the load transistor $M_L$,

$$i_{DL} = 0 = K_L(v_{GSL} - v_{IL})^2$$  \hspace{1cm} (15.19)$$

where $i_{DL}$ = drain current of the load transistor $M_L$
$v_{GSL}$ = gate-to-source voltage of the load transistor $M_L$
$v_{IL}$ = threshold voltage of the load transistor $M_L$
$K_L$ = MOS constant parameter of the load transistor $M_L$

Using KVL, we can write

$$v_{GSL} = V_{DD} - v_O$$  \hspace{1cm} (15.20)$$

Substituting $v_{GSL}$ from Eq. (15.20) into Eq. (15.19) yields

$$0 = K_L(V_{DD} - v_O - v_{IL})^2$$

which gives the output voltage as

$$v_O = V_{OH} = V_{DD} - V_{IL}$$  \hspace{1cm} (15.21)$$

Therefore, the high output voltage $V_{OH}$ is less than $V_{DD}$ by the amount of the threshold voltage of the load transistor $M_L$. The input voltage corresponding to $v_O = V_{OH}$ is

$$V_{IL} = V_{tD}$$  \hspace{1cm} (15.22)$$

In region II, $v_I > V_{tD}$. As $v_I$ becomes slightly greater than $V_{tD}$, $M_D$ begins to conduct and operates in the saturation region. The drain currents of the two transistors will be equal; that is, $i_{DL} = i_{DD}$, which, for both transistors in saturation, gives

$$K_L(v_{GSL} - v_{IL})^2 = K_D(v_{GSD} - v_{tD})^2$$

**NOTE**  D in the subscripts after D, DS, and GS refers to the driver MOSFET.

Substituting the values for the gate-to-source voltages, $v_{GSL} = V_{DD} - v_O$ and $v_{GSD} = v_I$, we get

$$K_L(V_{DD} - v_O - v_{IL})^2 = K_D(v_I - V_{tD})^2$$  \hspace{1cm} (15.23)$$

Solving for $v_O$, we get

$$v_O = V_{DD} - V_{IL} - (v_I - V_{tD})\left(\frac{K_D}{K_L}\right)^{1/2}$$  \hspace{1cm} (15.24)$$

Thus, the output voltage $v_O$ is a linear function of the input voltage $v_I$. The slope of the transfer characteristics is a constant given by

$$\frac{dv_O}{dv_I} = -\sqrt{\frac{K_D}{K_L}} = -\sqrt{K_R}$$  \hspace{1cm} (15.25)$$
where $K_R$ is known as the geometry ratio. The slope changes abruptly from 0 to $-\sqrt{K_R}$ at $V_{IL} = V_{ID}$. The slope, which is inversely proportional to the transition width $V_{TW}$, can be increased and the transition region can be narrowed with a large value of $K_D/K_L$. Since the MOS constant parameters ($K_D$ and $K_L$) are proportional to the ratio $W/L$, the slope can be rewritten as

$$
\frac{dv_O}{dv_I} = -\left[\left(\frac{W/L}{D}\right)\left(\frac{W/L}{L}\right)\right]^{1/2}
$$

(15.26)

The voltage $V_{D(mid)} = v_I = v_O$ can be found from Eq. (15.23):

$$
K_L(V_{DD} - V_{MD} - V_{IL})^2 = K_D(V_{MD} - V_{ID})^2
$$

which can be solved for $V_{MD}$ as follows:

$$
V_{MD} = \frac{V_{DD} - V_{IL} + V_{ID}\sqrt{K_R}}{1 + \sqrt{K_R}}
$$

(15.27)

If $v_I$ is increased sufficiently, $M_D$ will operate at the edge of the saturation region. If $v_{DSD(sat)}$ is the drain–source voltage at the transition point,

$$
v_{DSD(sat)} = V_{GSD} - V_{ID}
$$

(15.28)

Since $v_{DSD(sat)} = v_O$ and $V_{GSD} = v_I$, Eq. (15.28) can be written as

$$
v_O = v_I - V_{ID}
$$

(15.29)

Equating $v_O$ in Eq. (15.24) to the output voltage in Eq. (15.29) gives the input voltage $v_{I(tran)}$ at the transition point between the saturation and nonsaturation (ohmic) regions; that is,

$$
v_O = v_{I(tran)} - V_{ID} = V_{DD} - V_{IL} - (V_{I(tran)} - V_{ID})\sqrt{K_R}
$$

which, solved for $V_{I(tran)}$, yields

$$
V_{I(tran)} = \frac{V_{DD} - V_{IL} + V_{ID}(1 + \sqrt{K_R})}{1 + \sqrt{K_R}}
$$

(15.30)

In region III, $v_I > V_{I(tran)}$. $M_L$ and $M_D$ operate in the saturation and nonsaturation (ohmic) regions, respectively. From Eq. (7.6), the drain current $i_{DD}$ is given by

$$
i_{DD} = K_D[2(v_{GSD} - V_{ID})v_{DSD} - v_{DSD}^2]
$$

(15.31)

Since the two drain currents must be the same, we can find the relation between the input and the output voltages. Thus, from Eqs. (15.18) and (15.31), we get

$$
K_{L}(v_{GSL} - V_{IL}) = K_D[2(v_{GSD} - V_{ID})v_{DSD} - v_{DSD}^2]
$$

(15.32)
Substituting the values for the gate-to-source voltages, \( v_{\text{GSL}} = V_{\text{DD}} - v_{O} \), \( v_{\text{GSD}} = v_{I} \), and \( v_{\text{DS}} = v_{O} \), we get the relationship between the input and output voltages:

\[
K_L(V_{\text{DD}} - v_{O} - V_{\text{IL}})^2 = K_D[2(v_{I} - V_{\text{ID}})v_{O} - v_{O}^2]
\]  

(15.33)

If we use \( dv_{O} / dv_{I} = -1 \), Eq. (15.33) gives

\[
v_{I} = V_{\text{ID}} + 2v_{O} + \frac{v_{O} + V_{\text{IL}} - V_{\text{DD}}}{K_R}
\]

(15.34)

Solving Eqs. (15.33) and (15.34) for the logic high input voltage \( V_{I} = V_{\text{IH}} \) and the corresponding \( V_{O} \), we get

\[
v_{\text{IH}} = \frac{(V_{\text{DD}} - V_{\text{IL}})(2 + 1/K_R)}{\sqrt{1 + 3K_R}} + V_{\text{ID}} - \frac{V_{\text{DD}} - V_{\text{IL}}}{K_R}
\]

(15.35)

\[
V_{O} = \frac{V_{\text{DD}} - V_{\text{IL}}}{\sqrt{1 + 3K_R}}
\]

(15.36)

**NOTE** \( V_{O} \) in Eq. (15.36) is not \( V_{\text{OL}} \), which is found by substituting \( v_{I} = V_{\text{OH}} = V_{\text{DD}} - V_{\text{IL}} \) and \( v_{O} = V_{\text{OL}} \) in Eq. (15.33) and then solving the quadratic equation for \( V_{\text{OL}} \).

The output voltage and the drain currents depend on the input voltage \( v_{I} \). The transfer characteristics are shown in Fig. 15.13 for various values of the ratio \( K_D / K_L \). As the ratio \( K_D / K_L \) becomes larger, a steeper characteristic is obtained, which is highly desirable in digital circuits. At a larger value of \( K_D / K_L \), the low output voltage \( V_{\text{OL}} \) becomes smaller. To guarantee turnoff of the succeeding stages, the low output voltage \( V_{\text{OL}} \) should be less than the threshold voltage of the driver transistors.

![FIGURE 15.13 Transfer characteristics for various values of the ratio \( K_D / K_L \)](image-url)
Body Effect

The previous equations were derived by neglecting the body effect in the load transistor \( M_L \). In IC inverters, the body effect increases the threshold voltage. A higher value of \( V_{IL} \) causes \( V_{OH} \) to drop significantly, and the noise margin is decreased. In addition, a large geometry ratio \( K_R \) (large \( K_D \) and low \( K_L \)) is required to achieve a steep VTC. The threshold voltage \( V_t \) is related to \( V_{SB} \) (see Sec. 7.3.4) by

\[
V_t = V_{t0} + \gamma \sqrt{V_{SB} + 2\phi_f - \sqrt{2\phi_f}}
\]

(15.37)

where

- \( V_{t0} \) = threshold voltage at \( V_{SB} = 0 \), typically 1 V to 1.5 V
- \( \gamma \) = fabrication process constant, typically 0.3 \( \sqrt{V} \) to 1 \( \sqrt{V} \)
- \( 2\phi_f \) = equilibrium electrostatic potential of the p-type body material, typically 0.6 V

Using Eq. (15.37), \( V_{tD0} = V_t \) for \( M_D \), and \( V_{SB} = v_O \) for \( M_L \), we find that \( V_{IL} \) is given by

\[
V_{IL} = V_{tD0} + \gamma \sqrt{V_{OD} + 2\phi_f - \sqrt{2\phi_f}}
\]

(15.38)

This relationship for \( V_{IL}(v_O) \), which is a nonlinear function of \( v_O \), can be used to find the transfer characteristic of the inverter. However, several iterations are required, and the process can be tedious. It is rarely necessary to obtain a detailed analysis for circuit design, and such a task is usually left for computer-aided simulation.

EXAMPLE 15.3

**Designing an enhancement-load NMOS inverter**  Design an enhancement-load NMOS inverter, as shown in Fig. 15.12(a), to obtain a noise margin of \( N_{ML} \leq 0.8 \) V. The threshold voltages are \( V_{IL} = V_{ID} = 1 \) V, and \( K_L = 20 \) \( \mu \)A \( V^2 \). The supply voltage is \( V_{DD} = 5 \) V. Assume load capacitance \( C_L = 0.5 \) pF and clock frequency \( f_{clk} = 5 \) MHz.

(a) Find the design parameter \( K_R = K_D K_L \), neglecting the body effect.

(b) Calculate \( N_{ML} \) if the body effect is included. Assume \( \gamma = 0.5 \sqrt{V} \) and \( 2\phi_f = 0.6 \) V.

(c) Calculate the low-to-high propagation time \( t_{pLH} \).

(d) Calculate the high-to-low propagation time \( t_{pHL} \).

(e) Calculate the delay-power product (DP).

**SOLUTION**

\( N_{ML} = V_{IL} - V_{OL} \geq 0.8 \) V. Thus, for \( V_{IL} = V_{ID} = 1 \) V,

\[
V_{OL} = V_{IL} - N_{ML} \leq 0.2 \text{ V}
\]

Technically, \( V_{OH} = V_{DD} - V_{ID} = 5 - 1 = 4 \) V. However, to simplify the analysis for finding an expression for \( K_R \), we will assume \( V_{OH} = V_{IH} = V_{DD} = 5 \) V. For \( V_{IH} = 5 \) V, \( M_L \) and \( M_D \) operate in the...
saturation and nonsaturation (ohmic) regions, respectively. Thus, for \( v_I = V_{IH} = V_{OH} \) and \( v_O = V_{OL} \), Eq. (15.33) yields

\[
K_D[2(V_{OH} - V_{ID})V_{OL} - V_{OL}^2] = K_L(V_{DD} - V_{OL} - V_{IL})^2
\]  
(15.39)

which gives the desired value of the geometry ratio \( K_R \) as

\[
K_R = \frac{K_D}{K_L} = \frac{(V_{DD} - V_{OL} - V_{IL})^2}{2(V_{OH} - V_{ID})V_{OL} - V_{OL}^2}
\]  
(15.40)

(a) Without the body effect, \( V_{ID} = 1 \) V. For \( V_{OL} = 0.2 \) V and \( V_{OH} = 5 \) V, Eq. (15.40) gives \( K_R = 9.26 \). Since a higher value gives a lower value of \( V_{OL} \), we choose \( K_R = 10.0 \). For this value of \( K_R \), we can find \( V_{OL} \) by solving the quadratic equation in Eq. (15.39):

\[
10[2(5 - 1)V_{OL} - V_{OL}^2] = (5 - V_{OL} - 1)^2
\]

which gives \( V_{OL} = 0.19 \) V or 7.81 V (not physically meaningful). Thus, \( V_{OL} = 0.19 \) V, which is less than 0.2 V.

(b) With the body effect of \( V_O = 0.2 \) V, Eq. (15.38) gives

\[
V_{IL} = V_{ID} + \gamma \frac{|\sqrt{V_O} + 2\phi_I - 2\phi_{V|2}|}{|\sqrt{V_{OL}} + 0.2 + 0.6 - \sqrt{0.6}|} = 1.06 \text{ V}
\]  
(15.41)

With this value of \( V_{IL} = 1.06 \) V, Eq. (15.39) gives

\[
10[2(5 - 1)V_{OL} - V_{OL}^2] = (5 - 1.06 - V_{OL})^2
\]

which gives \( V_{OL} = 0.18 \) V. Thus, \( NM = V_{IL} - V_{OL} = 1 - 0.18 = 0.82 \) V

which is better than the specified 0.8 V.

(c) The NMOS inverter with a load capacitor \( C_L \) is shown in Fig. 15.14(a). As \( v_I \) goes low to \( V_{OL} \), \( M_D \) turns off immediately, and the capacitor \( C_L \) is charged up by the drain current \( i_{DL} \). The operating point, shown in Fig. 15.14(b), moves along the load line of \( M_L \) from point \( Q \) of \( V_{OL} \) to midpoint \( P \) of \( V_{OL(mid)} = (V_{OH} + V_{OL})/2 \). The charging of the capacitor is shown in Fig. 15.14(c). The output voltage is given by

\[
C_L \frac{dv_O}{dt} = i_{DL} = K_L(V_{DD} - v_O - V_{IL})^2
\]  
(15.42)

which can be integrated from \( V_{OL} \) to \( V_{OL(mid)} \) to give the low-to-high propagation time \( t_{pLH} \), as shown in Fig. 15.14(d). To simplify the analysis, we will find the average value of the charging current \( I_{CLH} \) and then find an approximate value of \( t_{pLH} \); that is,

\[
t_{pLH} = \frac{C_L(V_{OL(mid)} - V_{OL})}{I_{CLH}}
\]  
(15.43)

where \( I_{CLH} \) is given by

\[
I_{CLH} = \frac{i_C(V_{OL}) + i_C(V_{OL(mid)})}{2}
\]  
(15.44)
For $V_{OL} = 0.2 \text{ V}$,

$$V_{OH} = V_{DD} - V_{IL} = 5 - 1 = 4 \text{ V}$$
$$V_{O(mid)} = \frac{V_{OH} + V_{OL}}{2} = 2.1 \text{ V}$$

Then $i_C(V_{OL}) = i_{DL}(V_{OL}) = K_L(V_{DD} - V_O - V_{IL})^2 = 20 \mu \times (5 - 0.2 - 1)^2 = 288.8 \mu \text{ A}$

$$i_C(V_{O(mid)}) = K_L(V_{DD} - V_{O(mid)} - V_{IL})^2 = 20 \mu \times (5 - 2.1 - 1)^2 = 72.2 \mu \text{ A}$$

$$I_{CLH} = \frac{288.8 \mu + 72.2 \mu}{2} = 180.5 \mu \text{ A}$$

Thus, $t_{pLH} = 0.5 \text{ pF} \times \frac{(2.1 - 0.2) \text{ V}}{180.5 \mu \text{ A}} = 5.26 \text{ ns}$
(d) As \( v_I \) goes high to \( V_{OH} \), \( M_D \) turns on and the capacitor \( C_L \) discharges through \( M_D \). The operating point moves along the load line of \( M_D \) from point \( B \) of \( V_{OH} \) to midpoint \( D \) of \( V_{O(mid)} = (V_{OH} + V_{OL})/2 \). The output voltage is given by

\[ C_L \frac{dV_O}{dt} = i_{DD} - i_{DL} \]  \hspace{1cm} (15.45)

where

\[ i_{DD} = \begin{cases} K_D (V_{OH} - V_{ID})^2 & \text{for } (V_{OH} - V_{ID}) \geq V_O > V_{O(mid)} \\ K_D [2(V_{OH} - V_{ID})V_O - V_O^2] & \text{for } V_O \leq V_{O(mid)} \end{cases} \]

Using the average value of the capacitor discharging current \( i_{CHL} \), we can find the approximate value of \( t_{pHL} \) from

\[ t_{pHL} = \frac{C_L (V_{OH} - V_{O(mid)})}{i_{CHL}} \]  \hspace{1cm} (15.46)

where \( i_{CHL} \) is given by

\[ i_{CHL} = \frac{i_{DD}(V_{OH}) + i_{DD}(V_{O(mid)}) - i_{DL}(V_{O(mid)})}{2} \]  \hspace{1cm} (15.47)

For \( K_D = K_R K_L = 200 \mu A \ V^2 \),

\[ V_{OL} = 0.2 \ V \]
\[ V_{OH} = V_{DD} - V_{L} = 5 - 1 = 4 \ V \]
\[ V_{O(mid)} = \frac{V_{OH} + V_{OL}}{2} = 2.1 \ V \]

Thus,

\[ i_{DD}(V_{OH}) = K_D (V_{OH} - V_{ID})^2 = 200 \mu \times (4 - 1)^2 = 1800 \mu A \]
\[ i_{DD}(V_{O(mid)}) = K_D [2(V_{OH} - V_{ID})V_O - V_O^2] = 200 \mu \times [2 \times (4 - 1) \times 2.1 - 2.1^2] = 1638 \mu A \]
\[ i_{DL}(V_{O(mid)}) = K_L (V_{DD} - V_O - V_{DL})^2 = 20 \mu \times (5 - 2.1 - 1)^2 = 72.2 \mu A \]
\[ i_{CHL} = \frac{1800 + 1638 - 72.2}{2} = 1683 \mu A \]
\[ t_{pHL} = 0.5 \text{ pF} \times \frac{(4 - 2.1) \text{ V}}{1683 \mu A} = 0.56 \text{ ns} \]

which is much shorter than \( t_{pLH} = 5 \text{ ns} \). The value of \( i_{DL}(V_{O(mid)}) \), which is much smaller than that of \( i_{DD}(V_{OH}) \), can often be neglected.

(e) The propagation time \( t_{pd} \), which is the average of \( t_{pLH} \) and \( t_{pHL} \), is

\[ t_{pd} = \frac{t_{pLH} + t_{pHL}}{2} = \frac{5.26 \text{ ns} + 0.56 \text{ ns}}{2} = 2.91 \text{ ns} \]
15.5.2 NMOS Inverter with Depletion Load

The load in an inverter can be a depletion-type MOSFET. Consider an $n$-channel depletion MOSFET whose gate is connected to the source terminal, as shown in Fig. 15.15(a). With this connection, $v_{GS} = 0$; the output characteristic is shown in Fig. 15.15(b), which indicates that $v_{DS}$ must be zero to obtain a zero drain current.

Static Characteristics

An NMOS inverter with a depletion-load transistor $M_L$ is shown in Fig. 15.16(a), in which the substrates of the MOSFETs are connected to the ground. To simplify the analysis, we will assume that the substrates are connected to the source terminals so that $V_{SB} = 0$ for both $M_D$ and $M_L$. The output characteristics of the load and driver transistors are shown in Fig. 15.16(b). If $v_I = V_{OL}$, then $v_O = V_{OH}$. When $v_I$ goes high (to $V_{OH}$), transistor $M_D$ turns on, and the operating point jumps from $A$ to $B$ and then moves along the $i$-$v$ curve of $M_D$ until it finally reaches the quiescent point $Q$ so that $v_O = V_{OL}$. When $v_I$ goes from high to low (to $V_{OL}$), the operating point jumps from $Q$ to $A$ and then moves along the $i$-$v$ curve of $M_L$ until it finally reaches $v_O = V_{OH}$. Depending on the input voltage $v_I$, the VTC can be divided into four regions, as shown in Fig. 15.16(c).

![Figure 15.15](image-url)

**FIGURE 15.15** $n$-channel depletion-type MOSFET with $v_{GS} = 0$
In region I, the input voltage $v_I \leq V_{ID}$, where $V_{ID}$ is the threshold voltage of the drive transistor $M_D$. $M_D$ is cut off, so the drain currents of the two transistors must be zero. $M_L$ operates in the nonsaturation (ohmic) region. The drain current of load transistor $M_L$ is $i_{DL} = 0$, which, for $V_{DSL} = 0$, gives the output voltage as

$$v_O = V_{OH} = V_{DD} - V_{DSL} = V_{DD}$$

Thus, $V_{OH}$ is not reduced by $V_{IL}$, as it is in the case of an enhancement-load inverter.

In region II, $v_I > V_{ID}$. As $v_I$ becomes slightly greater than $V_{ID}$, $M_D$ begins to conduct and operates in the saturation region. $M_L$ is still in the nonsaturation region. The two drain currents must be equal; that is, $i_{DL} = i_{DD}$, which, for $M_L$ in the nonsaturation region and $M_D$ in the saturation region, gives

$$K_L[2(v_{GSL} - V_{IL})V_{DSL} - V_{DSL}^2] = K_D(v_{GSD} - V_{ID})^2$$

Substituting the values for $v_{GSL} = 0$, $V_{DSL} = V_{DD} - v_O$, and $v_{GSD} = v_I$, we get the relationship between the input and output voltages as

$$K_L[-2V_{IL}(V_{DD} - v_O) - (V_{DD} - v_O)^2] = K_D(v_I - V_{ID})^2$$

(15.49)

$V_{IL}$ can be found by differentiating Eq. (15.49) and setting $dv_O/dv_I = -1$ to solve for $v_I = V_{IL}$.

If $v_I$ is increased sufficiently, both $M_D$ and $M_L$ will operate in the saturation region. At the transition point from nonsaturation to saturation for $M_L$, we get

$$V_{DSL} = V_{DD} - v_O = V_{DSL} = 0 - V_{IL} = -V_{IL}$$

which gives $v_O$ at the edge of the transition as

$$V_{O(tran)} = V_O = V_{DD} + V_{IL}$$

(15.50)
Substituting $v_O$ from Eq. (15.50) into Eq. (15.49) gives the corresponding input voltage $V_I = V_{l(\text{tran}1)}$; that is,

$$K_L[-2V_I(V_{DD} - V_{DD} - V_{l}) - (V_{DD} - V_{DD} - V_{l})^2] = K_D(v_I - V_{id})^2$$

which, for $v_I > V_{id}$, gives

$$V_{l(\text{tran}1)} = v_I = V_{id} - V_{il}\left(\frac{K_L}{K_D}\right)^{1/2} = V_{id} - V_{il}\sqrt{\frac{T}{K_R}}$$  \hspace{1cm} (15.51)

**NOTE** $V_{l(\text{tran}1)}$ can also be found from

$$K_L(v_{GSL} - V_{l})^2 = K_D(v_{GSD} - V_{id})^2$$

for $v_{GSL} = 0$ and $v_{GSD} = v_I = V_{l(\text{tran}1)}$.

In region III, $v_I = V_{l(\text{tran}1)}$. Both $M_L$ and $M_D$ operate in the saturation region. Since the two currents must be equal, $i_{DL} = i_{DD}$, or

$$K_L(v_{GSL} - V_{l})^2 = K_D(v_{GSD} - V_{id})^2$$

Substituting $v_{GSL} = 0$ and $v_{GSD} = v_I$, we get the input voltage when both transistors are in saturation:

$$v_I = V_{id} - V_{il}\left(\frac{K_L}{K_D}\right)^{1/2} = V_{id} - V_{il}\sqrt{\frac{T}{K_R}}$$

The corresponding value of drain current is

$$i_{DL} = i_{DD} = K_L(v_{GSL} - V_{l})^2 = K_L(-V_{il})^2 = K_LV_{il}^2$$  \hspace{1cm} (15.52)

which indicates that the drain current is independent of $v_I$ and remains constant. At this current level, $M_L$ will continue to operate in the saturation region and $M_D$ will be forced into nonsaturation. There will be a quick transition as $M_D$ switches from the saturation to the nonsaturation region. The output voltage will change to

$$V_{O(\text{tran}2)} = v_O = v_{DSD} = v_{GSD} - V_{id} = v_I - V_{id}$$  \hspace{1cm} (15.53)

The input voltage at this transition is the same as $V_{l(\text{tran}1)}$; that is,

$$V_{l(\text{tran}2)} = V_{l(\text{tran}1)} = V_{id} - V_{il}\left(\frac{K_L}{K_D}\right)^{1/2}$$  \hspace{1cm} (15.54)

Note that at $v_I = V_{l(\text{tran}1)} = V_{l(\text{tran}2)}$, there will be two transitions: the first as $M_D$ switches from saturation to nonsaturation at an output voltage of $v_O = V_{DD} + V_{il}$ and then the second as $M_L$ switches from nonsaturation to saturation at an output voltage of $v_O = v_I - V_{id}$.

In region IV, $v_I > V_{l(\text{tran}1)} = V_{l(\text{tran}2)}$. The drive transistor $M_D$ goes into nonsaturation, and $M_L$ operates in the saturation region. Since the two currents must be equal, $i_{DL} = i_{DD}$, which, for $M_L$ in saturation and $M_D$ in nonsaturation, gives

$$K_L(v_{GSL} - V_{l})^2 = K_D[2(v_{GSD} - V_{id})v_{DSD} - v_{DSD}^2]$$
Substituting $v_{GSL} = 0$, $v_{DSD} = v_O$, and $v_{GSL} = v_I$, we get the relationship between the input and output voltages:

$$K_L(-V_{IL})^2 = K_D[2(v_I - V_{ID})v_O - v_O^2]$$  \hspace{1cm} (15.55)

If we set $dv_O \over dv_I = -1$, Eq. (15.55) yields

$$v_I = 2v_O + V_{ID}$$

which, after substitution into Eq. (15.55), gives

$$V_{IH} = \frac{2|V_{IL}|}{\sqrt{3K_R}} + V_{ID}$$ \hspace{1cm} (15.56)

$$v_O = \frac{|V_{IL}|}{\sqrt{3K_R}}$$ \hspace{1cm} (15.57)

The typical loci for the transition points for both the load and the drive transistors are shown in Fig. 15.17. The typical transfer characteristics are shown in Fig. 15.18 for several values of the ratio $K_D / K_L$. The depletion load provides a constant current once $M_L$ goes into saturation.

**Body Effect**

With the body effect, $V_{DL}$ varies with $v_{SB} = v_O$ by

$$V_{DL} = V_{DL0} + \gamma |\sqrt{v_O + 2\phi_f} - \sqrt{2\phi_f}|$$ \hspace{1cm} (15.58)

where $V_{DL0}$ is the threshold voltage at $v_{SB} = 0$. Since $|V_{DL}|$ decreases with falling $v_O$, the load current will decrease slightly and $v_O$ will still drop rapidly with $v_I$.

**FIGURE 15.17** Transition loci for driver and load transistors
EXAMPLE 15.4

Designing a depletion-load NMOS inverter  Design a depletion-load NMOS inverter, as shown in Fig. 15.16(a), to obtain a noise margin of \( \text{NM}_H \geq 2.7 \) V. The threshold voltages are \( V_{IL} = -2.5 \) V, \( V_{DD} = 1 \) V, and \( K_L = 20 \) \( \mu \)A \ V\(^2\). The supply voltage is \( V_{DD} = 5 \) V. Assume \( V_{OH} = V_{DD} = 5 \) V, load capacitance \( C_L = 0.5 \) pF, and frequency \( f_{\text{clk}} = 5 \) MHz.

(a) Find the design parameter \( K_R = K_D / K_L \), neglecting the body effect.
(b) Calculate \( \text{NM}_L \) if the body effect is neglected.
(c) Calculate \( \text{NM}_H \) if the body effect is included. Assume \( \gamma = 0.5 \sqrt{V} \) and \( 2\phi_L = 0.6 \) V.
(d) Calculate the low-to-high propagation time \( t_{pLH} \).
(e) Calculate the high-to-low propagation time \( t_{pHL} \).
(f) Calculate the delay-power product (DP).
(g) Use PSPice/SPICE to verify your results.

SOLUTION

(a) \( \text{NM}_H = V_{OH} - V_{IH} \geq 2.7 \) V. Thus, for \( V_{OH} = V_{DD} = 5 \) V,

\[
V_{IH} = V_{OH} - \text{NM}_H \leq 2.3 \text{ V}
\]
For $V_{IH} \leq 2.3$ V, Eq. (15.56) yields

$$K_R = \frac{(2V_{IL})^2}{3(V_{IH} - V_D)^2}$$

(15.59)

which gives $K_R = 4.93$. Since a higher value gives a lower $V_{IH}$, we choose $K_R = 5$. For this value of $K_R$, Eq. (15.56) gives $V_{IH} = 2.29$ V, and Eq. (15.57) gives $V_O = 0.65$ V.

(b) For $K_D = K_L \times K_R = 100$ $\mu$A/V$^2$, Eq. (15.49) gives

$$20 \mu \times [(-2) \times (-2.5) \times (5 - v_O) - (5 - v_O)^2] = 20 \mu \times 5(v_I - 1)^2$$

(15.60)

Setting $dv_O/dv_I = -1$, we get

$$v_I = 0.2v_O + 0.5$$

Solving these two equations, we get $V_{IL} = 1.46$ V and $v_O = 4.78$ V. For $v_I = V_OH$ and $v_O = V_OL$, Eq. (15.55) yields

$$K_L[2(V_{OH} - V_ID)V_{OL} - V_{OL}^2] = K_L V^2_{IL}$$

(15.61)

which, for $V_OH = 5$ V and $K_R = 5$, gives $V_{OL} = 0.16$ V. Thus,

$$NM_{IL} = V_{IL} - V_{OL} = 1.46 - 0.16 = 1.3$$

(c) With the body effect, at $v_O = 0.65$ V, Eq. (15.58) gives

$$V_{IL} = V_{IL0} + \gamma \sqrt{v_O + 2\phi_I - \sqrt{2\phi_I}}$$

$$= -2.5 + 0.5\sqrt{0.65 + 0.6} - \sqrt{0.6} = -2.33$$

With this value, Eq. (15.56) gives $V_{IH} = 2.20$ V. Thus,

$$NM_{IH} = V_{OH} - V_{IH} = 5 - 2.20 = 2.80$$

which is better than the specified 2.7 V.

(d) The inverter with a load capacitor $C_L$ is shown in Fig. 15.19(a). As $v_I$ goes low to $V_{OL}$, $M_D$ turns off immediately and the capacitor $C_L$ is charged up by the drain current $i_{DL}$. The operating point, shown in Fig. 15.19(b), moves along the load line of $M_L$ from point $Q$ of $V_{OL}$ to midpoint $P$ of $V_{O(mid)} = (V_{OH} + V_{OL})/2$. The charging of the capacitor is shown in Fig. 15.19(c). The output voltage is given by

$$C_L \frac{dv_O}{dt} = i_{DL} = K_L V^2_{IL}$$

(15.63)
which, after integration from \( V_{OL} \) to \( V_{O(mid)} \), gives \( t_{PLH} \) as

\[
    t_{PLH} = \frac{C_L(V_{OH} - V_{OL})}{2KLV_{IL}^2} \tag{15.64}
\]

\[
    = \frac{0.5 \text{ pF} \times (5 - 0.16)}{2 \times 20 \mu \times 2.5^2} = 9.7 \text{ ns}
\]

(e) As \( v_I \) goes high to \( V_{OH} \), \( M_D \) turns on and the capacitor \( C_L \) discharges through \( M_D \). The operating point moves along the load line of \( M_D \) from point \( B \) of \( V_{OH} \) to midpoint \( D \) of \( V_{O(mid)} \). The output voltage is given by

\[
    C_L \frac{dV_O}{dt} = i_{DD} - i_{DL} = i_{DD} - KLV_{IL}^2 \tag{15.65}
\]
\[V_{\text{OL}} = 0.16 \text{ V}, \quad V_{\text{OH}} = V_{\text{DD}} = 5 \text{ V}, \text{ and}\]

\[V_{\text{O(mid)}} = \frac{V_{\text{OH}} + V_{\text{OL}}}{2} = 2.6 \text{ V}\]

Thus, \[i_{\text{DD}}(V_{\text{OH}}) = K_D(V_{\text{OH}} - V_{\text{IL}})^2 = 100 \mu \times (5 - 1)^2 = 1600 \mu \text{A}\]

\[i_{\text{DD}}(V_{\text{O(mid)}}) = K_D[2(V_{\text{OH}} - V_{\text{IL}})V_{\text{OH}} - v_{\text{OH}}^2] = 100 \mu \times [2 \times (5 - 1) \times 2.6 - 2.6^2] = 1404 \mu \text{A}\]

\[i_{\text{DL}}(V_{\text{O(mid)}}) = K_LV_{\text{IL}}^2 = 20 \mu \times 2.5^2 = 125 \mu \text{A}\]

\[i_{\text{CHL}} = \frac{1600 \mu + 1404 \mu - 125 \mu}{2} = 1440 \mu \text{A}\]

Using Eq. (15.46), we have

\[t_{\text{PHL}} = 0.5 \text{ pF} \times \frac{(5 - 2.6) \text{ V}}{1440 \mu \text{A}} = 0.83 \text{ ns}\]

which is much shorter than \(t_{\text{PLH}} = 9.7 \text{ ns}\). Assuming that \(i_{\text{DD}}\) is much greater than \(i_{\text{DL}}\), Eq. (15.65) can be simplified to

\[C_L \frac{dv_{\text{O}}}{dt} = i_{\text{DD}} = K_D(V_{\text{DD}} - V_{\text{IL}})^2\]

which, after integration between \(V_{\text{OH}}\) and \((V_{\text{OH}} + V_{\text{OL}})/2\), gives \(t_{\text{PHL}}\) as

\[t_{\text{PHL}} = \frac{C_L(V_{\text{OH}} - V_{\text{OL}})}{2K_D(V_{\text{DD}} - V_{\text{IL}})^2}\]

\[= \frac{0.5 \text{ pF} \times (5 - 0.16)}{2 \times 100 \mu \times (5 - 1)^2} = 0.76 \text{ ns}\]

(close to 0.83 ns).

**f** The propagation time \(t_{\text{pd}}\), which is the average of \(t_{\text{PLH}}\) and \(t_{\text{PHL}}\), is

\[t_{\text{pd}} = \frac{t_{\text{PLH}} + t_{\text{PHL}}}{2} = \frac{9.7 \text{ ns} + 0.76 \text{ ns}}{2} = 5.2 \text{ ns}\]

The current at \(v_{\text{O}} = V_{\text{OL}}\) becomes

\[i_{\text{DL}} = i_{\text{DD}} = K_LV_{\text{IL}}^2 = 125 \mu \text{A}\]

\[P_{\text{static}} = \frac{5 \times 125 \mu \text{A}}{2} = 312.5 \mu \text{W}\]

Since \(P_{\text{dynamic}} = f_{\text{ck}}C_LV_{\text{DD}}^2 = 5 \text{ MHz} \times 0.5 \text{ pF} \times 5^2 = 62.5 \mu \text{W},\)

\[P_{\text{D}} = P_{\text{static}} + P_{\text{dynamic}} = 312.5 \mu + 62.5 \mu = 375 \mu \text{W}\]

Therefore, the delay-power product is

\[DP = P_{\text{D}} \times t_{\text{pd}} = 375 \mu \text{W} \times 5.2 \text{ ns} = 1.95 \text{ pJ}\]
15.5.3 Comparison of NMOS Inverters

The load lines for three basic types of inverters are shown in Fig. 15.22, superimposed on the drain characteristics of the driver transistor. The resistive load line exhibits linear characteristics, and its high output voltage is dependent on drain resistance $R_D$. An inverter with an enhancement load has a lower drain current for the same value of $v_{DS}$. An inverter with a depletion load shows a constant current over a wide range.
of $v_{DS}$; as a consequence of this characteristic, it will switch a capacitive load more rapidly than the other two inverters. Also, it exhibits a higher noise margin. Depletion-load inverters are commonly used in ICs for high-speed switching.

### KEY POINT OF SECTION 15.5

- An NMOS inverter uses an enhancement MOSFET as the driver. The load is either an enhancement or a depletion MOSFET. Depletion-load inverters allow more rapid switching and thus are commonly used in integrated circuits.

### 15.6 NMOS Logic Circuits

In digital circuits, NMOS inverters are frequently used in transmission gates and NOR and NAND gates.

#### 15.6.1 NMOS Transmission Gates

An NMOS transmission gate is basically an NMOS that is connected to an effective load capacitance $C_L$, as shown in Fig. 15.23. The substrate is connected to the most negative potential in the circuit rather than to the source terminal. The MOSFET may be assumed to be completely bilateral; that is, the drain and source terminals are identical.

To examine the operation of NMOS transmission gates, we will assume that the substrate is connected to zero (that is, $v_{SUB} = 0$) and consider the following cases based on the level of input voltage $v_I$:

- **Case 1**: $v_G = 0$ and $v_I = 0$ or 5 V. The gate will not be positive with respect to either input terminal 1 or output terminal 2, and the transistor will always be cut off. Thus, the input and output terminals will be isolated from each other. The input voltage $v_I$ may have any value without affecting the output voltage $v_O$.

- **Case 2**: $v_G = 5$ V and $v_I = 0$. The gate is at 5 V with respect to input terminal 1, so the transistor is turned on. Terminals 1 and 2 act as drain and source, respectively. The drain-to-source voltage and drain current will be zero. The output voltage will be zero: $v_O = 0$.

- **Case 3**: $v_G = 5$ V, $v_I = 5$ V, and initially $v_O = 0$. The gate and terminal 1 are at 5 V with respect to terminal 2. Terminal 1 acts as a drain, and terminal 2 acts as a source. Thus, the current will flow from terminal 1 to terminal 2, and it will charge the load capacitor until the gate-to-output voltage becomes

![FIGURE 15.23 NMOS transmission gate](image-url)
equal to the threshold voltage $V_t$ of the transistor; that is, $v_G - v_O = V_t$. For a transistor with $V_t = 1$ V, $v_O = v_G - V_t = 5 - 1 = 4$ V.

In case 4, $v_G = 5$ V, $v_I = 0$, and initially $v_O = v_G - V_t = 4$ V. The gate is at 5 V, and terminal 2 is at 4 V with respect to terminal 1. Terminal 2 acts as a drain, and terminal 1 acts as a source. The transistor will be on. Thus, the current will flow from terminal 2 to terminal 1, and it will discharge the load capacitor; that is, $v_O = 0$.

In case 5, $v_G = 0$, $v_I = 0$ or 5 V, and initially $v_O = v_G - V_t = 4$ V. When $v_G$ goes to zero, the situation is similar to that in case 1, and the transistor will be cut off. The input and output terminals are isolated.

Therefore, as long as $v_G$ is high, the transistor transmits the input voltage to the output. If $v_G$ is low, the transistor is turned off, and the input and output terminals are isolated. Depending on the levels of the gate and input voltages, the transmission gate can be operated as (1) an analog switch, (2) a sample-and-hold circuit that converts analog signals to their digital equivalents, or (3) a pass transistor for steering logic signals.

### 15.6.2 NMOS NOR Gates

An NMOS NOR logic gate is shown in Fig. 15.24. Two parallel NMOS transistors are connected in series with a depletion-type load. The transfer characteristic is similar to that of a depletion-load inverter. If both input voltages $v_A$ and $v_B$ are less than the threshold voltage of the driver transistors, $v_O = V_{OH}$ (for logic 1). If $v_A = V_{OH}$ (for logic 1), then $M_1$ will turn on, and the output voltage will drop to low output, $v_O = V_{OL}$. If both $v_A$ and $v_B$ are high (at $V_{OH}$), then both $M_1$ and $M_2$ will turn on, and the output voltage will be low (at $V_{OL}$). The exact value of the output voltage will depend on the transistor parameters, as discussed in Sec. 15.5, and it will vary slightly depending on whether one or both driver transistors are turned on. The logic function for an NMOS NOR gate is shown in Table 15.4.

#### TABLE 15.4 NMOS NOR gate logic function

<table>
<thead>
<tr>
<th>$v_A$</th>
<th>$v_B$</th>
<th>$v_O$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_{OL}$ (0)</td>
<td>$V_{OL}$ (0)</td>
<td>$V_{OH}$ (1)</td>
</tr>
<tr>
<td>$V_{OL}$ (0)</td>
<td>$V_{OH}$ (1)</td>
<td>$V_{OH}$ (1)</td>
</tr>
<tr>
<td>$V_{OH}$ (1)</td>
<td>$V_{OL}$ (0)</td>
<td>$V_{OL}$ (0)</td>
</tr>
<tr>
<td>$V_{OH}$ (1)</td>
<td>$V_{OH}$ (1)</td>
<td>$V_{OL}$ (0)</td>
</tr>
</tbody>
</table>
15.6.3 NMOS NAND Gates

An NMOS NAND gate is shown in Fig. 15.25. Two transistors are connected in series with a depletion-type load. If both input voltages $v_A$ and $v_B$ are less than the threshold voltage of the driver transistors, $v_O = V_{OH}$ (for logic 1). If $v_A = V_{OH}$ and $v_B$ is still less than the threshold voltage $V_{t2}$ of $M_2$, then $M_2$ is still cut off, and $v_O = V_{OH}$. If both $v_A$ and $v_B$ are high (at $V_{OH}$), then both $M_1$ and $M_2$ will turn on, and the output voltage will be low (at $V_{OL}$). The exact value of the output voltage will depend on the transistor parameters. The logic function for an NMOS NAND gate is shown in Table 15.5.

<table>
<thead>
<tr>
<th>$v_A$</th>
<th>$v_B$</th>
<th>$v_O$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_{OL}$ (0)</td>
<td>$V_{OL}$ (0)</td>
<td>$V_{OH}$ (1)</td>
</tr>
<tr>
<td>$V_{OL}$ (0)</td>
<td>$V_{OH}$ (1)</td>
<td>$V_{OH}$ (1)</td>
</tr>
<tr>
<td>$V_{OH}$ (1)</td>
<td>$V_{OL}$ (0)</td>
<td>$V_{OH}$ (1)</td>
</tr>
<tr>
<td>$V_{OH}$ (1)</td>
<td>$V_{OH}$ (1)</td>
<td>$V_{OH}$ (1)</td>
</tr>
</tbody>
</table>

15.7 CMOS Inverters

Complementary, or CMOS, circuits use both $n$-channel and $p$-channel enhancement-type MOSFETs in the same circuit. Because of their very low power consumption, CMOS circuits are commonly used in ICs. A CMOS inverter is shown in Fig. 15.26(a). Transistor $M_P$ is a $p$-channel device, and transistor $M_N$ is an $n$-channel device. Each substrate is tied to its source. The input signal is connected to both transistor gates, and the output terminal is common to both drain terminals. The load characteristics of two CMOS devices are shown in Fig. 15.26(b) for two extreme inputs: $v_I = 0$ and $v_I = V_{DD}$. For $v_I = 0$, $M_N$ is off and its drain current is zero. $M_P$ has the characteristic corresponding to $v_{GSP} = V_{DD}$. For $v_I = V_{DD}$, $M_P$ is off and its drain current is zero. $M_N$ has the characteristic corresponding to $v_{GSN} = V_{DD}$. Thus, their drain currents are zero at these inputs, and the current drawn from the supply is zero.
As the input voltage $v_I$ is varied from zero to the maximum value $V_{DD}$, the output voltage $v_O$ falls from $V_{DD}$ to zero. The transfer characteristic is shown in Fig. 15.26(c). Depending on the input voltage $v_I$, the VTC can be divided into five regions.

In region I, $0 \leq v_I < V_{IN}$, where $V_{IN}$ is the threshold voltage of transistor $M_N$. Since $v_{GSN} = v_I < V_{IN}$, $M_N$ remains off. At a low value of $v_I$, $V_{SGP} = V_{DD} - v_I$ is high and positive. $M_P$ is turned on, and it is driven into the nonsaturation (ohmic) region. Since the channel resistance of the off-transistor $M_N$ is very much greater than that of the on-transistor $M_P$ and since $M_N$ and $M_P$ form a voltage divider, the output voltage is $v_O \approx V_{DD}$, as shown in Fig. 15.26(c). The various voltages are $v_{GSN} = v_I$, $v_{GSP} = -V_{DD} + v_I$, and $v_O = V_{DD}$.

In region II, $V_{IN} \leq v_I \leq V_{MD} = V_{l(tran)}$ and $v_{GSP} = (V_{DD} + v_I) < V_{IP}$, where $V_{IP}$ is the threshold voltage of transistor $M_P$ and $V_{l(tran)}$ is defined by Eq. (15.68). As $v_I$ becomes greater than or equal to $V_{IN}$, $M_N$ conducts and operates in the saturation region, for which $v_{GSN}$ is described by $V_{IN} < v_{GSN} (= v_I) < (V_{DSN} + V_{IN})$.

For $v_{GSP} < V_{IP}$, $M_P$ remains in the ohmic region. With $M_N$ in the saturation region and $M_P$ in the ohmic region, the two drain currents must be equal; that is, $i_{DN} = i_{DP}$. Applying the expressions for the saturation and ohmic regions gives

$$K_n(v_{GSN} - v_{IN})^2 = K_p[2(v_{SGP} + V_{IP})v_{SDP} - V_{SDP}^2]$$

where $K_n$ and $K_p$ are the constants for $n$-type and $p$-type transistors, respectively. Substituting $v_{GSN} = v_I$, $v_{SGP} = V_{DD} - v_I$, and $v_{SDP} = V_{DD} - v_O$ into the above equation, we get the relationship between $v_I$ and $v_O$ as

$$K_n(v_I - V_{IN})^2 = K_p[2(V_{DD} - v_I + V_{IP})(V_{DD} - v_O) - (V_{DD} - v_O)^2]$$  \hspace{1cm} (15.67)$$

$V_{IL}$ can be found by differentiating Eq. (15.67) and setting $dv_O \over dv_I = -1$ to solve for $v_I = V_{IL}$. 

**FIGURE 15.26** CMOS inverter
If $v_I$ is increased further, $v_{GSN}$ increases and $v_{SGP}$ decreases. Both $M_N$ and $M_P$ operate in the saturation region. At the transition point from the ohmic to the saturation region for $M_P$, we get

$$v_{SGP} = v_{SDP} - V_{IP}$$

Substituting the values for $v_{SGP} = V_{DD} - v_I$ and $v_{SDP} = V_{DD} - v_O$, we get

$$V_{DD} - v_I = V_{DD} - v_O - V_{IP}$$

which gives the input voltage at the first transition as

$$V_{I(\text{tran1})} = v_I = v_O + V_{IP}$$

The corresponding output voltage is

$$V_{O(\text{tran1})} = v_O = v_I - V_{IP}$$

whose plot is a straight line that intercepts the output axis at $-V_{IP}$ (a positive quantity), as shown in Fig. 15.26(c). The intersection with the transfer characteristic gives $V_{I(\text{tran1})}$ and $V_{O(\text{tran1})}$. To solve for the value of $V_{I(\text{tran1})}$ or $V_{O(\text{tran1})}$, one of these quantities must be known.

In region III, $v_I = V_{I(\text{tran1})}$. Both $M_N$ and $M_P$ operate in the saturation region. Since the two drain currents must be equal, $i_{DN} = i_{DP}$, or

$$K_n(v_{GSN} - V_{IN})^2 = K_p(v_{SGP} + V_{IP})^2$$

Substituting $v_{GSN} = v_I$ and $v_{SGP} = V_{DD} - v_I$, we get the input voltage at the transition of $M_P$ from the ohmic to the saturation region:

$$V_{I(\text{tran1})} = V_{MD} = \frac{V_{DD} + V_{IP} + V_{IN}\sqrt{K_n/K_p}}{1 + \sqrt{K_n/K_p}} = \frac{V_{DD} + V_{IP} + V_{IN}\sqrt{K_R}}{1 + \sqrt{K_R}} (15.68)$$

which is independent of output voltage $v_O$. For identical transistors, $K_n = K_p$ and $V_{IN} = |V_{IP}|$, and Eq. (15.68) is reduced to

$$V_{I(\text{tran1})} = V_{MD} = \frac{V_{DD}}{2} (15.69)$$

which is desirable to maximize the noise immunity of the circuit. Once we determine the value of $V_{I(\text{tran1})}$ from Eq. (15.69), we can find the output voltage at the edge of the transition of $M_P$ from

$$V_{O(\text{tran1})} = v_O = V_{I(\text{tran1})} - V_{IP}$$

This segment ends when $M_N$ enters the ohmic region, which is defined by

$$v_{GSN} = v_{DSN} + V_{IN}$$
Substituting $v_{GSN} = v_I$ and $v_{DSN} = v_O$ into the preceding equation yields

$$v_I = v_O + V_{IN}$$

which gives the input voltage at the transition of $M_N$ from the saturation region to the ohmic region as

$$V_{I(tran2)} = v_I = v_O + V_{IN}$$

The corresponding output voltage is given by

$$v_O = v_I - V_{IN}$$

which intercepts the input axis at $V_{IN}$, as shown in Fig. 15.26(c). The intersection with the transfer characteristic gives $V_{I(tran2)}$, $V_{O(tran2)}$, $V_{I(tran1)}$, and $V_{O(tran1)}$. Since $V_{I(tran1)}$ and $V_{O(tran2)}$ must be different. There will be two transitions—the first for $M_P$ from the ohmic region to the saturation region at an output voltage of $v_O = v_I + V_{IP}$. Since the two drain currents must be equal, $i_{DN} = i_{DP}$, which, for $M_P$ in the saturation region and $M_N$ in the ohmic region, gives

$$K_n[2(v_{GSN} - V_{IN})v_{DSN} - v_{DSN}^2] = K_p(v_{SGP} + V_{IP})^2$$

Substituting $v_{GSN} = v_I$, $v_{DSN} = v_O$, and $v_{SGP} = V_{DD} - v_I$, we can express the relationship between $v_O$ and $v_I$ as

$$K_n[2(v_I - V_{IN})v_O - v_O^2] = K_p(V_{DD} - v_I + V_{IP})^2$$

$$V_{IH}$$ can be found by differentiating Eq. (15.70) and setting $dv_O / dv_I = -1$ to solve for $v_I = V_{IH}$.

In region V, $V_{I(tran1)} = V_{I(tran2)}$, $V_{I(tran1)} = V_{I(tran2)}$, $V_{IN}$. $V_{I(tran2)}$, and $V_{O(tran2)}$ must be different. There will be two transitions—the first for $M_P$ from the ohmic region to the saturation region at an output voltage of $v_O = v_I + V_{IP}$ and then the second for $M_N$ from the saturation region to the ohmic region at an output voltage of $V_{O(tran2)} = V_{I(tran1)} - V_{IN}$.

In region IV, $v_I = V_{DD} - V_{IP}$. $M_N$ goes into the ohmic region, and $M_P$ still operates in the saturation region. Since the two drain currents must be equal, $i_{DN} = i_{DP}$, which, for $M_P$ in the saturation region and $M_N$ in the ohmic region, gives

$$V_{IH}$$ can be found by differentiating Eq. (15.70) and setting $dv_O / dv_I = -1$ to solve for $v_I = V_{IH}$.

In region V, $v_I > V_{DD} + V_{IP}$, $M_P$ is in the cutoff region, and $M_N$ is in the ohmic region. There will be virtually no current through the transistors, and the output voltage will be zero; that is,

$$i_{DN} = i_{DP} = 0$$

$$v_O = 0$$

**EXAMPLE 15.5**

**Designing a CMOS inverter** Design a CMOS inverter, as shown in Fig. 15.26(a), to operate at a transition voltage $V_{MD} = 2.5$ V. The threshold voltages are $V_{IP} = -1$ V, $V_{IN} = 1$ V, and $K_p = 20$ $\mu$A $\cdot$ V$^{-2}$. The supply voltage is $V_{DD} = 5$ V. Assume $V_{IH} = V_{OH} = 5$ V, load capacitance $C_L = 0.5$ pF, and frequency $f_{clk} = 5$ MHz.

(a) Find the design parameter $K_R = K_n K_p$, neglecting the body effect.

(b) Calculate $N_{ML}$ and $N_{MH}$.

(c) Calculate the propagation delay $t_{pd}$.

(d) Calculate the delay-power product (DP).

(e) Use PSpice/SPICE to verify your results.
SOLUTION

(a) Solving Eq. (15.68) for $K_R$, we have

$$K_R = \frac{(W/L)_N}{(W/L)_P} = \left[ \frac{V_{DD} + |V_{IP}| - V_M}{V_M - V_{IN}} \right]^2$$ (15.71)

which, for $V_{MD} = 2.5 \, \text{V}$, gives $K_R = 1$. Thus, $K_n = K_p = 20 \, \mu\text{A/V}^2$.

(b) $V_{OH} = V_{DD} = 5 \, \text{V}$. Then

$$V_{O(\text{tran1})} = V_{MD} - V_{IP} = 2.5 + 1 = 3.5 \, \text{V}$$

$$V_{O(\text{tran2})} = V_{MD} - V_{IN} = 2.5 - 1 = 1.5 \, \text{V}$$

Substituting numerical values of $V_{DD}$, $V_{IN}$, $V_{IP}$, $K_n$ and $K_p$ in Eq. (15.67), we get

$$v_I^2 = 14 - 8v_I + 2v_O + 2v_O v_O - v_O^2$$

which, at $dv_O/dv_I = -1$, gives $v_O = v_I + 2.5$. Solving these two equations, we get $V_{IH} = v_I = 2.13 \, \text{V}$ at $v_O = 4.63 \, \text{V}$.

Substituting numerical values of $V_{DD}$, $V_{IN}$, $V_{IP}$, $K_n$ and $K_p$ in Eq. (15.70), we get

$$16 - 8v_I + v_I^2 = 2v_O v_O - 2v_O - v_O^2$$

which, at $dv_O/dv_I = -1$, gives $v_I = v_O + 2.5$. Solving these two equations, we get $V_{IH} = v_I = 2.88 \, \text{V}$ at $v_O = 0.38 \, \text{V}$.

At $v_I = V_{DD} + V_{IP} = 5 - 1 = 4$, $M_P$ turns off and $V_{OL} = v_O = 0$. Thus,

$$N_{ML} = V_{IL} - V_{OL} = 2.13 - 0 = 2.13 \, \text{V}$$

$$N_{MH} = V_{OH} - V_{IH} = 5 - 2.88 = 2.12 \, \text{V}$$

(c) The inverter with a load capacitor $C_L$ has an arrangement similar to that shown in Fig. 15.19(a). As $v_I$ goes low to $V_{OL}$, $M_N$ turns off immediately, and capacitor $C_L$ is charged up by the drain current $i_{DP}$. Since $M_P$ is in saturation and $v_I = V_{OL} = 0$, the output voltage is related to the charging current by

$$C_L \frac{dv_O}{dt} = i_{DP} = K_p(V_{DD} - v_I + V_{IP})^2$$

which, after integration between $V_{DD}$ and $V_{DD}/2$, gives $t_{pLH}$ as

$$t_{pLH} = \frac{C_L V_{DD}}{2K_p(V_{DD} - |V_{IP}|)^2} = \frac{0.5 \, \text{pF} \times 5 \, \text{V}}{2 \times 20 \, \mu\text{A/V}^2 \times (5 \, \text{V} - 1 \, \text{V})^2} = 3.91 \, \text{ns}$$ (15.72)

Because of the topology of the CMOS inverter, $t_{PHL}$ will have the same value as $t_{pLH}$. Thus, the propagation time $t_{pd} = t_{pLH} = 3.91 \, \text{ns}$. In practice, a symmetric CMOS does not occupy the minimum chip area, and thus not all CMOS designs are symmetric.

(d) A CMOS inverter draws a negligible current (on the order of nanoamperes) from the power supply in both high and low states. Hence, the static power dissipation is almost zero: $P_{static} = 0$. This is a distinct advantage
for portable CMOS equipment because standby operation of the equipment will not discharge the battery. We have

\[ P_{\text{dynamic}} = f_{\text{clk}} V_{\text{DD}}^2 = 5 \text{ MHz} \times 0.5 \text{ pF} \times 5^2 = 62.5 \mu \text{W} \]

so \[ P_D = P_{\text{static}} + P_{\text{dynamic}} = 0 + 62.5 \mu \text{W} = 62.5 \mu \text{W} \]

Therefore, the delay-power product is

\[ \text{DP} = P_D \times t_{\text{pd}} = 62.5 \mu \text{W} \times 3.91 \text{ ns} = 0.244 \text{ pJ} \]

(e) The PSpice schematic is shown in Fig. 15.27. The PSpice plot of the VTC is shown in Fig. 15.28(a), which gives \( V_{\text{IL}} = 2.1245 \text{ V} \) (expected value is 2.13 V), \( V_{\text{IH}} = 2.8755 \text{ V} \) (expected value is 2.88 V), \( V_{\text{OL}} = 0 \) (expected value is 0) at \( V_{\text{OH}} = 5 \text{ V} \), and \( V_{\text{I(tran)}} = 2.5 \) (expected value is 2.5 V). The VTC values are very close to the hand calculations, as expected. The transient response is shown in Fig. 15.28(b), which gives \( t_{\text{pHL}} = 4.18 \text{ ns} \) and \( t_{\text{pLH}} = 4.255 \text{ ns} \) (for \( L = 50 \mu \text{m} \) and \( W = 100 \mu \text{m} \)). The transient performance, however, will depend on the values of length \( (L) \) and width \( (W) \). A typical value of the transconductance parameter for the NMOS process is \( \mu_n \text{Cox} = 40 \mu \text{A/V}^2 \). [Note: We can find \( dv_O \) \( dv_I = -1 \) points by plotting \( dv(2) \) in Probe.]

---

**FIGURE 15.27** PSpice schematic for Example 15.5

**FIGURE 15.28** PSpice plots for CMOS inverter (using transistor circuit model) for Example 15.5
15.8 CMOS Logic Circuits

Like NMOS inverters, CMOS inverters are frequently used in digital circuits in transmission gates and NOR and NAND gates.

15.8.1 CMOS Transmission Gates

A CMOS transmission gate consists of an NMOS and a PMOS that are connected in parallel and feed an effective load capacitance $C_L$. This arrangement is shown in Fig. 15.29. The substrate of the NMOS ($M_N$) is usually connected to the most negative potential (assumed to be the ground in Fig. 15.29), and the substrate of the PMOS ($M_P$) is connected to the most positive potential (usually the positive supply voltage $V_{DD}$) in the circuit rather than to the source terminal. MOSFETs may be assumed to be completely bilateral; that is, the drain and source terminals of each transistor are identical. The gate control voltage $v_{cnt}$ of the PMOS is the complement of the gate voltage of the NMOS. Assuming that $v_I$ operates between 0 and 5 V, $V_{DD} = 5$ V, $V_{IN} = 1$ V, and $V_{tP} = -1$ V, we will consider the following cases based on the level of input voltage $v_I$.

In case 1, $v_G = 0$, $v_{cnt} = 5$ V, $v_I = 0$ or 5 V, and initially $v_O = 0$ or 5 V. The gate of $M_N$ is never positive with respect to either terminal, so $M_N$ is always cut off. The gate of $M_P$ is never negative with respect to either terminal, so $M_N$ is always cut off. Thus, the input and output terminals are isolated from each other. The input voltage $v_I$ may take on any value without affecting the output voltage $v_O$.

---

**FIGURE 15.29** CMOS transmission gate

---

KEY POINT OF SECTION 15.7

- CMOS circuits use both $n$-channel and $p$-channel enhancement MOSFETs in the same circuit. Because they have very low power consumption and offer very high speed, they are commonly used in ICs.
In case 2, $v_G = 5$ V, $v_{\text{cnt}} = 0$, and $v_I = 0$. The gate-to-terminal-1 voltage of $M_N$ is 5 V, so $M_N$ will be turned on. Terminals 1 and 2 of $M_N$ act as the drain and the source, respectively. The drain-to-source voltage and drain current are zero. The output voltage is zero: $v_O = 0$. With $v_{\text{cnt}} = 0$ and $v_I = v_O = 0$, $M_P$ will be cut off.

In case 3, $v_G = 5$ V, $v_{\text{cnt}} = 0$, $v_I = 5$ V, and initially $v_O = 0$. The gate and terminal 1 of $M_N$ are at 5 V with respect to terminal 2 of $M_N$. Terminal 1 and terminal 2 of $M_N$ act as the drain and the source, respectively. The drain current of $M_N$ flows from terminal 1 to terminal 2 and charges the load capacitor until the gate-to-output voltage becomes equal to the threshold voltage $V_{tN}$; that is,

$$v_{\text{GSN}} = v_G - v_O = V_{IN}$$

For a transistor with $V_{IN} = 1$ V, $v_O = V_G - V_{IN} = 5 - 1 = 4$ V. When $v_O$ rises to $v_G - V_{IN} = 5 - 1 = 4$ V, the gate-to-source voltage of $M_N$ is equal to the threshold voltage $V_{tN}$, and $M_N$ turns off. However, with $v_{\text{cnt}} = 0$ on the gate of $M_P$, $M_P$ is turned on, with terminal 1 and terminal 2 acting as the drain and the source, respectively. After $M_N$ is turned off, $M_P$ is still turned on; the capacitor is charged all the way up to $v_I$, so the output voltage becomes $v_O = v_I$, at which point the drain-to-source voltage on $M_P$ is zero and $M_P$ is turned off. It is important to note that the output of a CMOS transmission gate is the full value of $v_I$.

In case 4, $v_G = 5$ V, $v_{\text{cnt}} = 0$, $v_I = 5$ V, and initially $v_O = v_I$. The gate and terminal 2 of $M_N$ are at 5 V with respect to terminal 1 of $M_N$. Terminal 1 and terminal 2 of $M_N$ act as the source and the drain, respectively. $M_N$ is always turned on, and the drain current flows from terminal 2 to terminal 1, causing the load capacitor to be discharged completely to zero, so $v_O = 0$. $M_P$, whose terminal 2 acts as the source, always remains turned off.

In case 5, $v_G = 0$, $v_{\text{cnt}} = 5$ V, $v_I = 0$ or 5 V, and initially $v_O = v_I$. When $v_G$ goes to zero, the situation is similar to case 1, and both transistors will be cut off. The input and output terminals are isolated.

The advantage of a CMOS transmission gate is that the output voltage $v_O$ is always equal to $v_I$ when the transmission gate is turned on, so $v_O = v_I$. The logic function can be described by

$$v_O = \begin{cases} v_I & \text{if } v_G \text{ is high (at logic 1)} \\ v_O & \text{if } v_G \text{ is low (at logic 0)} \end{cases}$$

With an NMOS transmission gate, the output voltage is reduced by the threshold voltage, so $v_O = v_I - V_{IN}$. The major disadvantage of the CMOS gate is that it requires both a gate voltage $v_G$ and its complement for successful operation.

**Propagation Delay**

For $v_I = V_{DD}$ and $v_O = 0$ (initially), both $M_N$ and $M_P$ are in saturation. Capacitor $C_L$, which arises from the interconnecting area of the two MOSFETs and the load, charges to $V_{DD}$. Thus, the charging time constant $\tau_{LH}$ (for output low to high) can be found from the channel resistance of the complementary pair; that is,

$$\tau_{LH} = C_L(R_L \parallel R_{\text{dN}} \parallel R_{\text{dp}}) \quad (15.73)$$

where $R_L$ is the load resistance. $R_{\text{dN}}$, which is the static resistance of the NMOS, is given by

$$R_{\text{dN}} = \frac{v_{\text{DSN}}}{i_{\text{dN}}} = \frac{V_{DD} - v_O}{K_n(V_{DD} - v_O - V_{IN})^2} \quad (15.74)$$
$R_dP$, which is the static resistance of the PMOS, is given by

$$R_dP = \frac{v_{SDP}}{i_{DP}} = \frac{V_{DD} - v_O}{K_p(V_{DD} - |V_{IP}|)^2} \tag{15.75}$$

The charging time constant $\tau_{LH}$ is usually higher than the discharging time. Assuming $t_{pLH} \approx t_{pHL}$ and $t_{pd} \approx t_{pLH}$, its value can be estimated from $\tau_{LH}$. Thus, the propagation time $t_{pd}$ to charge the capacitor $C_L$ from 0 to $V_{DD}/2$ is given by

$$t_{pd} = \tau_{LH} \ln 2 = 0.69315 \tau_{LH} \tag{15.76}$$

### 15.8.2 CMOS NOR and NAND Gates

A two-input NOR logic gate is shown in Fig. 15.30. Two parallel NMOSs are connected in series with two PMOSs. The substrates of the PMOSs are connected to the most positive potential $V_{DD}$; the substrates of the NMOSs are connected to the most negative potential—that is, the ground. The transfer characteristic is similar to that of a CMOS inverter. If both input voltages $v_A$ and $v_B$ are less than the threshold voltage (assuming $v_A = v_B = V_{OL}$), $M_{N1}$ and $M_{N2}$ are cut off. At the same time, the $p$-channel transistors $M_{P1}$ and $M_{P2}$ are turned on. Thus, the output voltage becomes high: $v_O = V_{OH}$ (at logic 1).

If $v_A = V_{OH}$, then $M_{N1}$ is turned on, and $M_{P1}$ is turned off. In this case, the output voltage drops to low: $v_O = V_{OL}$. If both $v_A$ and $v_B$ are equal to $V_{OH}$, then both $M_{N1}$ and $M_{N2}$ turn on, and the output voltage becomes low. For $v_A = v_B = V_{OL}$, both $M_{N1}$ and $M_{N2}$ are cut off, and the drain currents will be zero. The logic function is the same as that for an NMOS, shown in Table 15.4. If one or both of the logic inputs are at logic high, then at least one PMOS is cut off, and the drain current(s) is again zero. Therefore, the steady-state current is zero, and the power dissipation is essentially zero. Current flow and power dissipation occur only during the transition from one state to another.

A two-input NAND logic gate is shown in Fig. 15.31. Two NMOSs are connected in series, and two PMOSs are connected in parallel. The transfer characteristic is similar to that of a CMOS inverter. If both...
input voltages \( v_A \) and \( v_B \) are less than the threshold voltage (assuming \( v_A = v_B = V_{OL} \) at logic 0), \( M_{N1} \) and \( M_{N2} \) are turned off, and \( M_{P1} \) and \( M_{P2} \) are turned on. In this case, the output voltage becomes high: \( v_O = V_{OH} \) (at logic 1). If \( v_A = V_{OH} \) and \( v_B = V_{OL} \), then \( M_{N1} \) is turned on, \( M_{N2} \) is cut off, \( M_{P1} \) is off, and \( M_{P2} \) is turned on. Because of the high impedance from the drain to the source of \( M_{N2} \), the output voltage becomes high: \( v_O = V_{OH} \). If \( v_A = v_B = V_{OH} \), then both \( M_{N1} \) and \( M_{N2} \) are turned on, and both \( M_{P1} \) and \( M_{P2} \) are turned off. In this case, the output voltage is low: \( v_O = V_{OL} \). The logic function is the same as that for an NMOS, shown in Table 15.5.

CMOS gates have extremely high input resistance, on the order of hundreds of megohms, which is desirable for the driving circuitry. Unfortunately, they are susceptible to damage to the thin gate-oxide layer (typically 0.1 \( \mu \)m for a metal gate and less for a polysilicon gate), which has a breakdown voltage of 50 V to 100 V. Since the capacitance of a typical human body is 100 pF to 300 pF, a person walking across a waxed laboratory floor or brushing against a garment may generate static voltage in excess of 10 kV! Stray electrostatic discharge from a person handling the CMOS can easily release enough energy to cause permanent damage. A charged body can release tens of kilowatts of power over hundreds of nanoseconds. To protect a CMOS, clamping diodes, shown in Fig. 15.30, are built in to CMOS gates to limit any input voltages that fall outside the range from \( V_{SS} \) to \( V_{DD} \). Also, any distributed input resistance \( (R_s) \), which is typically 1.5 k\( \Omega \) for a metal gate and 250 \( \Omega \) for a polysilicon gate, limits the transient gate current.

### 15.8.3 CMOS Families

The CMOS family of gates dates from the late 1960s. The 74Cxx logic families are second-generation CMOS circuits. These are polysilicon rather than metal gates, and the devices are smaller and faster than earlier designs. In addition, outputs are double buffered, as shown in Fig. 15.32. A 74Cxx gate has two cascaded inverters to isolate the logic from the output. As a result, the voltage gain is increased with a very steep VTC.

The 74HCxx series is the third generation of the CMOS family, which continues the trend toward smaller size and considerably less power, typically \( P_D = 500 \mu W \) and \( t_{pd} = 10 \) ns. The 74ACxx series is the fourth generation of the CMOS family, which is faster at the same power, typically \( P_D = 500 \mu W \) and \( t_{pd} = 4 \) ns.
15.9 Comparison of CMOS and NMOS Gates

The major advantages and disadvantages of CMOS and NMOS gates are listed below:

1. The output voltage of a CMOS gate is the full input voltage $v_I$ without the drop due to the threshold voltage, as in the case of NMOS gates.
2. A CMOS gate requires more transistors than an NMOS gate to perform similar logic functions.
3. CMOS gates consume very little power, thereby enabling very-large-scale integration (VLSI). NMOS gates consume more power than CMOS gates and have thermal limitations that make them less attractive for VLSI.
4. CMOS gates draw spikes of current during the transition from one state to another; the current peaks occur when both NMOS and PMOS transistors are in saturation.
5. CMOS gates occupy a larger area and have larger capacitances than NMOS gates.

15.10 BJT Inverters

BJT switches are the building blocks of bipolar logic circuits [7, 2, 5]. The earliest logic device using BJTs was the basic inverter, developed in the 1960s. This device was followed by the resistor-transistor logic (RTL), diode-transistor logic (DTL), and transistor-transistor logic (TTL) families of logic circuits. In RTL, DTL, and TTL families, the BJTs operate by switching between the saturation (on) and cutoff (off) regions of
operation; hence, they are generally called saturation-logic families. Because their BJT saturation-logic circuits experience delay due to the storage time in saturated devices, RTL and DTL circuits are no longer used. In some BJTs (e.g., Schottky BJTs), the time delay required to bring a transistor out of saturation is avoided by preventing the BJT from becoming saturated. Although TTL logic circuits are being challenged by CMOS circuits, TTL technology has improved over the years and continues to be popular. The gate delay of a modern TTL circuit may be as low as 1.5 ns.

There are two other families of logic circuits: the emitter-coupled logic (ECL) circuit and the integrated-injection logic (I\textsuperscript{2}L) circuit. They are operated by switching a constant current between two parts of a circuit and avoiding transistor saturation. The gate delay of an ECL circuit can be less than 1 ns, and ECLs find applications in digital communication circuits and high-speed circuits in supercomputers. I\textsuperscript{2}L circuits, however, have lost ground to CMOS logic circuits in many applications.

### 15.10.1 Voltage Transfer Characteristics

A BJT with a collector (pull-up) resistance \( R_C \) is shown in Fig. 15.33(a). When the input \( v_I \) is low, such that the base-to-emitter voltage is less than the forward-bias cut-in voltage, \( v_{BE} \), transistor \( Q_1 \) is off, and the output \( v_O \) is high; that is, \( v_O = V_{OH} = V_{CC} \). The transistor operates at point \( A \) as shown in Fig. 15.33(b). When \( v_I \) goes high to \( V_{OH} \), the operating point jumps to point \( B \) and moves along the output characteristic (in the active region) toward the quiescent point \( Q \) (in the saturation region). The output voltage becomes low—that is, the collector–emitter saturation voltage, \( V_{CE(sat)} \). The VTC is shown in Fig. 15.33(c).

The collector current at saturation is given by

\[
I_{C(sat)} = \frac{V_{CC} - V_{CE(sat)}}{R_C}
\]  

and the corresponding base current at saturation is given by

\[
I_{B(sat)} = \frac{I_{C(sat)}}{\beta_F} \quad \text{at } V_{CE(sat)}
\]  

\[\text{(15.77)}\]

---

**FIGURE 15.33** BJT inverter
where $\beta_F$ is the forward current gain of the transistor. Normally, the circuit is designed with $I_B$ higher than $I_{B(sat)}$ to ensure that $Q_1$ is driven into saturation. $I_B$ can be found from

$$I_B = \frac{V_1 - V_{BE(sat)}}{R_B}$$

The ratio of $I_B$ to $I_{B(sat)}$ is called the overdrive factor $k_{ODF}$:

$$k_{ODF} = \frac{I_B}{I_{B(sat)}}$$

The ratio of $I_{C(sat)}$ to $I_B$ is called the forced $\beta_F$, given by

$$\beta_{F(forced)} = \frac{I_{C(sat)}}{I_B} = \frac{I_{B(sat)}\beta_F}{I_B} = \frac{\beta_F}{k_{ODF}}$$

$V_{CE(sat)}$, which changes slightly with collector current $I_{C(sat)}$, can be found from

$$V_{CE(sat)} = V_T \ln \left( \frac{I_B\beta_F + I_C\beta_F(1 - \alpha_R)}{I_B\beta_F\alpha_R - I_C\alpha_R} \right)$$

which, for a typical value of reverse current gain $\alpha_R = 0.1$, $I_C = I_{C(sat)}$, and $I_B = I_{B(sat)}$, becomes

$$V_{CE(sat)} = V_T \ln \left( \frac{10 + 9\beta_{F(forced)}I_{B(sat)}}{1 - I_{C(sat)}\beta_{F(forced)}} \right)$$

(See Appendix D.) The value of $V_{CE(sat)}$ usually falls in the range of 0.1 V to 0.3 V. The base-to-emitter junction characteristic is similar to that of a diode, and $V_{BE(sat)}$ usually falls in the range of 0.65 V to 0.8 V.

### 15.10.2 Switching Characteristics

A forward-biased $pn$ junction exhibits two parallel capacitances: a depletion-layer capacitance and a diffusion capacitance. On the other hand, a reverse-biased or zero-biased $pn$ junction has only a depletion capacitance. Under steady-state conditions, these capacitances do not play any role. However, under switching conditions, they contribute to the on and off behavior of the transistor. The typical waveforms and switching times are shown in Fig. 15.34. As the base–emitter voltage $v_{BE}$ rises from zero to $V_B$, the collector current does not respond immediately. There is a delay time $t_d$ before any collector current flows. This delay time is required to charge up the capacitance of the base-to-emitter junction (BEJ) to the forward-bias voltage $V_{BE(cut-in)}$ (approximately 0.7 V). After this delay, the collector current rises to the steady-state value of $I_{C(sat)}$. The rise time $t_r$ depends on the time constant determined by base capacitance $C_B$. Note that when the collector current $i_C$ rises, the output voltage $v_O$ falls. Thus, $t_r$ for $i_C$ in Fig. 15.34(b) corresponds to $t_f$ for $v_O$ in Fig. 15.34(e).
The base current is normally more than that required to saturate the transistor. As a result, the excess minority carrier charge is stored in the base region. The higher the overdrive factor $k_{ODF}$, the greater will be the amount of extra charge stored in the base. This extra charge, which is called the saturating charge, is proportional to the excess base drive $\Delta I_B$ given by

$$\Delta I_B = I_B - I_{B(sat)} = k_{ODF}I_{B(sat)} - I_{B(sat)} = I_{B(sat)}(k_{ODF} - 1)$$

The saturating charge is given by

$$Q_s = \tau_s \Delta I_B = \tau_s I_{B(sat)}(k_{ODF} - 1)$$

where $\tau_s$ is known as the storage time constant of the transistor. When $v_{BE}$ falls to zero, the collector current does not change for a time $t_s$, called the storage time, which is the time required to remove the saturating charge from the base. Once the extra charge is removed, the BEJ capacitance discharges to zero. Fall time $t_f$ depends on the time constant, which is determined by the capacitance of the reverse-biased BEJ.

The propagation time $t_{pHL}$ consists of the delay time $t_d$ and the time $t_{f1}$ for the output to fall from $V_{OH}$ to $(V_{OH} + V_{OL})/2$; that is, $t_{pHL} = t_d + t_{f1}$. The propagation time $t_{pLH}$ consists of the storage time $t_s$ and the time $t_{r1}$ for the output to rise from $V_{OL}$ to $(V_{OH} + V_{OL})/2$. That is, $t_{pLH} = t_s + t_{r1}$.

**FIGURE 15.34** Switching times of bipolar transistors
EXAMPLE 15.6

**Designing a BJT inverter**  Design a BJT inverter, as shown in Fig. 15.35, to drive five identical inverters \((N = 5)\) and to give \(V_{OH} = 3.5 \text{ V}\) and \(V_{IL} = 0.4 \text{ V}\). The transistor has \(I_{C(max)} = 5 \text{ mA}\), \(\beta_F = 100\) to 150, \(V_{BE(cut-in)} = 0.6 \text{ V}\), \(V_{BE(sat)} = 0.8 \text{ V}\), \(t_d = 1 \text{ ns}\), and \(t_s = 2 \text{ ns}\). The supply voltage \(V_{CC}\) is 5 V.

(a) Find the values of \(R_C\) and \(R_B\).

(b) Calculate \(k_{ODF}\) and \(N_{MH}\).

(c) Find the maximum fan-out \(N\) for \(V_{IH} = 2.0 \text{ V}\) and \(k_{ODF} = 1\).

(d) Calculate the propagation delay \(t_{pd}\). Assume that each load is an NMOS and can be represented by a capacitance \(C_B = 2 \text{ pF}\) in series with resistance \(R_B = 1.8 \text{ k}\Omega\).

(e) Calculate the delay-power product (DP) at a frequency \(f_{clk} = 5 \text{ MHz}\).

**SOLUTION**

(a) \(N = 5\), \(V_{OH} = 3.5 \text{ V}\), \(V_{IL} = V_{BE(cut-in)} = 0.6 \text{ V}\), and 
\[V_{OL} = V_{IL} - N M_L = 0.6 - 0.4 = 0.2 \text{ V}\]

For \(V_{CE(sat)} = V_{OL} = 0.2 \text{ V}\) (for \(V_T = 25.8 \text{ mV}\) and \(\beta_F = 100\)), Eq. (15.78) gives \(\beta_F(\text{forced}) = 71.8\). Using Eq. (15.77), we have 
\[R_C = \frac{(V_{CC} - V_{CE(max)})}{I_{C(sat)}} = \frac{(5 - 0.2)}{5 \text{ mA}} = 960 \Omega\]

We choose \(R_C = 1 \text{ k}\Omega\). When the input is low, \(Q_1\) is off and \(v_O = V_{OH}\). The base current \(I_B\) of each load stage is 
\[I_B = \frac{V_{OH} - V_{BE(sat)}}{R_B}\]

**FIGURE 15.35**  BJT inverter driving identical inverters
and \( V_{OH} = V_{CC} - N I_B R_C \). Eliminating \( I_B \) and solving for \( V_{OH} \), we get

\[
V_{OH} = \frac{V_{CC} + N R_C V_{BE(sat)}/R_B}{1 + N R_C/R_B} = \frac{R_B V_{CC} + N R_C V_{BE(sat)}}{R_B + N R_C}
\]  

which, for \( V_{OH} = 3.5 \text{ V} \), \( N = 5 \), and \( V_{BE(sat)} = 0.8 \text{ V} \), gives \( R_B = 9 R_C = 9 \text{ k}\Omega \).

(b) We have

\[
I_B = \frac{I_{C(sat)}}{\beta_{F(\text{forced})}} = \frac{5 \text{ mA}}{71.8} = 70 \mu\text{A}
\]

\[
I_{B(sat)} = \frac{I_{C(sat)}}{\beta_F} = \frac{5 \text{ mA}}{100} = 50 \mu\text{A}
\]

\[
k_{ODF} = \frac{I_B}{I_{B(sat)}} = \frac{70}{50} = 1.40
\]

\[
V_{IH} = R_B I_B + V_{BE(sat)}
\]

\[
= 9 \text{ k}\Omega \times 70 \mu\text{A} + 0.8 \text{ V} = 1.43 \text{ V}
\]

Thus, \( \text{NM}_H = V_{OH} - V_{IH} = 3.5 - 1.43 = 2.07 \text{ V} \). This noise margin is a measure of the safety factor that allows the load transistors to remain saturated despite changes in supply voltage, temperature, and manufacturing tolerances.

(c) When \( Q_1 \) is off, the output voltage \( v_O \) becomes

\[
v_O = V_{CC} - N I_B R_C = V_{CC} - N k_{ODF} I_{B(sat)} R_C = V_{CC} - N \frac{k_{ODF} (V_{CC} - V_{CE(sat)})}{\beta_F}
\]

For regeneration of logic levels at the load gates, \( v_O \geq V_{IH} \), and we find the fan-out \( N \) as

\[
N \leq \frac{\beta_F (V_{CC} - V_{IH})}{k_{ODF} (V_{CC} - V_{CE(sat)})}
\]

\[
\leq 100 \times \frac{5 - 1.4}{1.39 \times (5 - 0.2)} = 53.4
\]

Hence, the maximum fan-out is \( N = 53 \).

(d) When \( Q_1 \) turns off, each load is represented by a capacitance \( C_B \) and a resistance \( R_B \). The equivalent circuit is shown in Fig. 15.36(a). For identical loads, all branches are effectively in parallel as far as the output node is concerned, and Fig. 15.36(a) can be simplified to Fig. 15.36(b). That is, the time constant for low to high can be found from

\[
\tau_{rl} \approx N C_B \left( R_C + \frac{R_B}{N} \right)
\]

\[
= 5 \times 2 \text{ pF} \times \left( 1 \text{ k}\Omega + \frac{9 \text{ k}\Omega}{5} \right) = 28 \text{ ns}
\]
The output voltage \( v_O \) during charging can be found from

\[
v_O = V_{CC} - \frac{R_C V_{CC}}{R_C + R_B/N} e^{-t/t_{r1}}
\]

from which we can find the time required for \( v_O \) to rise from \( V_{OL} = 0.2 \) V to \( V_{O(mid)} \), which is equal to

\[
\left( \frac{V_{OH}}{V_{OL}} \right) = \frac{V_{CC} - V_{OL}}{V_{CC} - V_{O(mid)}}
\]

Thus, \( t_{pLH} = t_s + t_{r1} = 2 + 11.79 = 13.79 \) ns.

When \( Q_1 \) is turned on to saturation, each capacitor discharges through the transistor. The equivalent circuit is shown in Fig. 15.36(c). The time constant for high to low can be found from

\[
\tau_{f1} = C_B R_B
\]

\[
= 2 \, \text{pF} \times 9 \, \text{k\Omega} = 18 \, \text{ns}
\]

The output voltage \( v_O \) during discharging can be found from

\[
v_O = V_{OHE} e^{-t/\tau_n}
\]

\[\text{FIGURE 15.36} \quad \text{Equivalent circuits for Example 15.6}\]
from which we can find the time for \( v_O \) to fall from \( V_{OH} = 3.5 \text{ V} \) to \( (V_{OH} + V_{OL})/2 = 1.85 \text{ V} \); that is,

\[
t_{f1} = \tau_{f1} \ln \left( \frac{2V_{OH}}{V_{OH} + V_{OL}} \right)
\]

\[
= 18 \text{ ns} \times \ln \left( 2 \times \frac{3.5}{3.5 + 0.2} \right) = 11.48 \text{ ns}
\]

Thus, \( t_{PHL} = t_d + t_{f1} = 1 + 11.48 = 12.48 \text{ ns} \). The propagation time is

\[
t_{pd} = \frac{t_{PHL} + t_{PLH}}{2} = 13.135 \text{ ns}
\]

(e) The total power loss in the two junctions is found as follows:

\[
P_{\text{static}} = V_{CE(sat)}I_{C(sat)} + V_{BE(sat)}I_B
\]

\[
= 0.2 \times 5 \text{ mA} + 0.8 \times 70 \mu\text{A} = 1.056 \text{ mW}
\]

\[
P_{\text{dynamic}} = Nf_{\text{clk}}C_BV_{OH}^2 = 5 \times 5 \text{ MHz} \times 2 \text{ pF} \times 3.5^2 = 0.61 \text{ mW}
\]

\[
P_D = P_{\text{static}} + P_{\text{dynamic}} = 1.67 \text{ mW}
\]

Therefore, the delay-power product is

\[
DP = P_D \times t_{pd} = 1.67 \text{ mW} \times 13.135 \text{ ns} = 21.9 \text{ pJ}
\]

**NOTE** The value of \( DP \) is much higher than that of a CMOS or an NMOS inverter.

### KEY POINT OF SECTION 15.10

- A BJT is operated as a switching device. It is usually overdriven to ensure operation in the saturation region. This causes a reduction in the switching speed due to charge recovery. The higher the amount of overdrive, the greater will be the amount of extra charge stored in the base.

### 15.11 Transistor-Transistor Logic Gates

Equation (15.81) shows that, to achieve a high fan-out, \( \beta_F \) should be as high as possible. In 1965 the first TTL family was introduced specifically to increase switching speed without sacrificing fan-out or noise margin and without increasing power dissipation. TTL gates were leading DTL gates in sales by 1970 and dominated the digital IC market for more than 10 years. Many developments in the process and manufacturing technology as well as in circuit design techniques have led to several new generations (families) of TTL gates. Each generation has its advantages and disadvantages relative to the previous generations. As the technology has advanced, it has become economically feasible to obtain high performance by using
more transistors per gate, and several families have emerged, among which are standard TTL gates, high-speed (H) TTL gates, low-power (L) TTL gates, Schottky (S) TTL gates, low-power Schottky (LS) gates, advanced Schottky (AS) gates, advanced low-power Schottky (ALS) gates, and Fairchild advanced Schottky (F or Fast) gates.

The performance parameters of TTL 54/74 families are summarized in Table 15.6. In early TTL families, the improvement in switching speed resulted in more power dissipation. The Schottky series, however, has both lower power dissipation and improved speed. Currently, TTL applications focus on low-power Schottky (LS) and advanced low-power Schottky (ALS) families of gates. The ALS family offers the highest speed (although it is surpassed by the ECL family and rivaled by the CMOS family). As examples, we will analyze the TTL, and ALS types.

### 15.11.1 Standard TTL Gates

The input stage of a TTL gate uses a multiemitter transistor $Q_1$, as shown in Fig. 15.37(a). In isoplanar ICs, multiemitters are normally fabricated within the same base region. Figure 15.37(b) shows a cross section of a three-emitter transistor. $Q_1$ can be viewed as three diodes from the base to the emitter and one from the base to the collector, as shown in Fig. 15.37(c).

A TTL NAND gate with a multiemitter transistor input is shown in Fig. 15.38. The combination of $Q_4$, $D_1$, and $Q_3$ is called the *totem-pole output stage*. The transistor $Q_2$ forms a phase splitter since the collector and emitter voltages are $180^\circ$ out of phase. The circuit operation can be divided into two modes: mode 1 and mode 2.

### TABLE 15.6 Parameters of 54/74 series TTL logic gates

<table>
<thead>
<tr>
<th>Parameter</th>
<th>74</th>
<th>74H</th>
<th>74L</th>
<th>74S</th>
<th>74LS</th>
<th>74AS</th>
<th>74ALS</th>
<th>74F</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_{\text{IL}}$, in V</td>
<td>0.8</td>
<td>0.8</td>
<td>0.8</td>
<td>0.8</td>
<td>0.8</td>
<td>0.8</td>
<td>0.8</td>
<td>0.8</td>
</tr>
<tr>
<td>$V_{\text{IH}}$, in V</td>
<td>2.0</td>
<td>2.0</td>
<td>2.0</td>
<td>2.0</td>
<td>2.0</td>
<td>2.0</td>
<td>2.0</td>
<td>2.0</td>
</tr>
<tr>
<td>$V_{\text{OL}}$, in V</td>
<td>0.4</td>
<td>0.4</td>
<td>0.4</td>
<td>0.5</td>
<td>0.5</td>
<td>0.5</td>
<td>0.5</td>
<td>0.5</td>
</tr>
<tr>
<td>$V_{\text{OH}}$, in V</td>
<td>2.4</td>
<td>2.4</td>
<td>2.4</td>
<td>2.7</td>
<td>2.7</td>
<td>2.7</td>
<td>2.7</td>
<td>2.7</td>
</tr>
<tr>
<td>$t_{\text{pd}}$, in ns at $C_L = 50$ pF</td>
<td>10</td>
<td>6</td>
<td>30</td>
<td>3</td>
<td>10</td>
<td>1.5</td>
<td>4</td>
<td>2.5</td>
</tr>
<tr>
<td>$P_D$, in mW</td>
<td>10</td>
<td>25</td>
<td>1</td>
<td>20</td>
<td>2</td>
<td>8</td>
<td>1</td>
<td>5</td>
</tr>
<tr>
<td>$P_D$, in pJ</td>
<td>100</td>
<td>150</td>
<td>30</td>
<td>60</td>
<td>20</td>
<td>12</td>
<td>4</td>
<td>12.5</td>
</tr>
</tbody>
</table>

![FIGURE 15.37 Multiemitter bipolar transistor](image-url)
During mode 1, the input voltages $v_A$ and $v_B$ are high, at $V_{OH}$; that is, $v_A = v_B = V_{OH}$. The TTL NAND gate is at low-state output voltage, as shown in Fig. 15.39, with $N$ similar load gates. $I_{C3}$ becomes the collector saturation current $I_{CS}$ of $Q_3$. The two emitter junctions of $Q_1$ are reverse biased. The voltage $v_{B1}$ at the base of $Q_1$ is large enough to forward-bias the base–collector junction of $Q_1$ and drive $Q_2$ into saturation. Since the base–collector junction is forward biased and the BEJ is reverse biased,
Q₁ will be operating in its *inverse-active mode*; that is, the roles of the emitter and collector terminals are interchanged. Under this condition, the terminal current relationships become

\[ I_{E1} = -\beta_R I_{B1} \]
\[ I_{C1} = -(1 + \beta_R)I_{B1} \]

where \( \beta_R \) is the inverse-mode current gain of the transistor Q₁.

Assuming that Q₂ and Q₃ are saturated, the base current of Q₁ can be found from

\[ I_{B1} = \frac{V_{CC} - V_{BE2(sat)} - V_{BE3(sat)} - V_{BC1}}{R_1} \quad (15.86) \]

where \( V_{BC1} \) is the base-to-collector voltage of Q₁. Thus, the base current of Q₂ is

\[ I_{B2} = -I_{C1} = (1 + \beta_R)I_{B1} \]

Since the transistor Q₂ is driven into saturation, its collector current is

\[ I_{C2} = \frac{V_{CC} - V_{BE3(sat)} - V_{CE2(sat)}}{R_2} \quad (15.87) \]

Thus, the emitter current of Q₂ is

\[ I_{E2} = I_{B2} + I_{C2} \]

and the current through the recovery resistance \( R_B \) is

\[ I_R = \frac{V_{BE3(sat)}}{R_B} \]

which gives the base current of Q₃ as

\[ I_{B3} = I_{E2} - I_R \]

The maximum collector current to maintain Q₃ in saturation is given by

\[ I_{C3(sat)} = \beta_{F\text{forced}}I_{B3} \]

Since transistor Q₃ is driven into saturation, the output low voltage is

\[ v_O = V_{OL} = V_{CE3(sat)} \]

and the voltage at the collector of Q₂ is

\[ V_{C2} = V_{BE3(sat)} + V_{CE2(sat)} \]

Thus, the voltage difference between \( v_{C2} \) and \( v_O \) is

\[ v_{C2} - v_O = V_{BE3(sat)} + V_{CE2(sat)} - V_{CE3(sat)} \approx V_{BE3(sat)} \]
which is the voltage across the BEJ of Q4 and D1 and will not be sufficient to turn on both Q4 and D1. With Q3 in saturation and the output voltage in its low state, Q4 will be off. Thus, Q3 will sink to the (low-state) load current $I_{L0}$ given by

$$I_{L0} = N_L I_{N0}$$

where $I_{N0}$ is the individual load current for each fan-out at low output (i.e., $I_{N0} = I_{IL}$) and is obtained by multiplying the base current of $Q_1$ by $\beta_R$; that is,

$$I_{N0} = \beta_R \left[ \frac{V_{CC} - V_{CE3(sat)} - V_{BE1(sat)}}{R_1} \right] \quad (15.88)$$

Thus, the low-output fan-out is given by

$$N_L = \frac{I_{C3(sat)}}{I_{N0}}$$

During mode 2, at least one of the inputs $v_A$ or $v_B$ is low, at $V_{OL}$; that is, $v_A$ (or $v_B$) = $V_{OL}$. The TTL NAND gate in the output high state is shown in Fig. 15.40. The BEJ is forward biased through $R_1$ and $V_{CC}$. The base current $I_{B1}$ causes an emitter current through the particular emitter that is connected to the low input. Transistor action forces collector current into $Q_1$. But the collector current of $Q_1$, which equals the reverse-biased saturation current out of the base of $Q_2$, is usually much smaller than its base current. As a result, $Q_1$ will be in saturation. With $v_A$ (or $v_B$) = $V_{OL}$, the base current of $Q_1$ becomes

$$I_{B1} = \frac{V_{CC} - V_{BE1(sat)}}{R_1}$$

![TTL gate in the output high state](image-url)
Transistors Q2 and Q3 are cut off. Transistor Q4 and diode D1 supply the (high-state) load current $I_{L1}$ given by

$$I_{L1} = N_H I_{N1}$$

where $I_{N1}$ is the individual load current for each fan-out at high output (i.e., $I_{N1} = I_{H1}$). Since Q1 is operating in the inverse-action mode, its base current must be multiplied by $\beta_R$ to give $I_{N1} = \beta_R I_{B1}$; that is,

$$I_{N1} = \beta_R \left[ \frac{V_{CC} - V_{BE2(sat)} - V_{BE3(sat)} - V_{BC1}}{R_1} \right]$$  \hspace{1cm} (15.89)

With Q4 in the active region, the base current of Q4 is

$$I_{B4} = \frac{I_{L1}}{1 + \beta_F} = \frac{N_H I_{N1}}{1 + \beta_F}$$

Using KVL around the loop formed by $V_{CC}$, $R_2$, Q4, and D1, we get

$$V_{CC} = R_2 I_{B4} + V_{BE4} + V_{D1} + v_O$$

$$= \frac{R_2 N_H I_{N1}}{1 + \beta_F} + V_{BE4} + V_{D1} + v_O$$  \hspace{1cm} (15.90)

which gives the value of the high-output fan-out as

$$N_H = \frac{(1 + \beta_F)(V_{CC} - V_{BE4} - V_{D1} - v_O)}{R_2 I_{N1}}$$  \hspace{1cm} (15.91)

If $R_4$ were not present, the collector current of Q4 would be $\beta_F I_{B4}$, which would be highly undesirable. To limit the collector current of Q4 to an acceptable value, $R_4$ is introduced. The maximum collector current of Q4 is

$$I_{C4(max)} = \frac{V_{CC} - v_O - V_{CE4(sat)} - V_{D1}}{R_4}$$  \hspace{1cm} (15.92)

In summary, TTL NAND gates have the following features:

1. Short switching time from saturation to cutoff and lower delay time, typically 10 ns as compared to 40 ns for DTL gates
2. High noise margins
3. High fan-out capability
4. Sharp transfer characteristic

**NOTES**

1. In practice, the same load will be connected to the TTL gate circuit during the output high and output low states; that is, $N_L = N_H = N$.
2. From Eq. (15.90), we can find the permissible maximum output voltage as

$$v_{O(height)} = V_{OH} \approx V_{CC} - V_{BE4} - V_{D1} = V_{CC} - 0.7 - 0.7 = V_{CC} \left[ 1 - \frac{1 - 1.4}{V_{CC}} \right]$$

For $V_{CC} = 5$ V, $V_{OH} = 5 - 1.4 = 3.6$ V.
EXAMPLE 15.7

Designing a TTL NAND gate

(a) Design the TTL NAND gate of the circuit in Fig. 15.38. It has two inputs and feeds four identical NAND gates. The output voltage at output high is $V_{OH} = 3.5$ V, and $I_{N1} = 72.5 \, \mu A$. Assume $V_{CC} = 5$ V, $V_D = 0.7$ V, $V_{CE(sat)} = 0.2$ V, $V_{BE} = 0.7$ V, $V_{BC} = 0.7$ V, $\beta_{F(forced)} = 10, \beta_R = 0.1, I_{C3(sat)} = 1$ mA, and $I_{C4(max)} = 1$ mA. Capacitance of each load is, $C_i = 5$ pF.

(b) Use PSpice/SPICE to check your design by plotting the transfer function. Determine $N_{ML}$ and $N_{MH}$.

Model parameters for diodes are

$RS=4 \quad TT=0.1NS$

and for transistors are

$BF=10 \quad BR=0.1 \quad TF=0.1NS \quad TR=10NS \quad VJC=0.85 \quad VAF=50$

SOLUTION

(a) The design can be carried out using the following steps:

Step 1. Using Eq. (15.89), calculate the value of $R_1$:

$$R_1 = \beta_R \frac{V_{CC} - 2V_{BE(sat)} - V_{BC}}{I_{N1}} = 0.1 \times \frac{[5 - (2 \times 0.7) - 0.7]}{72.5 \, \mu A} = 4 \, k\Omega$$

Step 2. Using Eq. (15.91) for $V_O = V_{OH}$, find the value of $R_2$ to meet the output requirement:

$$R_2 = \frac{(1 + \beta_{F(forced)})(V_{CC} - V_{BE4} - V_{D1} - V_{OH})}{N_{I_{N1}}}$$

$$= (1 + 10) \times \frac{(5 - 0.7 - 0.7 - 3.5) \, V}{4 \times 72.5 \, \mu A} = 3.79 \, k\Omega$$

Step 3. Using Eq. (15.86), find the value of $I_{B1}$ at output high:

$$I_{B1} = \frac{(5 - 0.7 - 0.7 - 0.7) \, V}{4 \, k\Omega} = 725 \, \mu A$$

Step 4. Using Eq. (15.88), calculate the low-state load current:

$$I_{N0} = \beta_R \left[ \frac{V_{CC} - V_{CE3(sat)} - V_{BE1(sat)}}{R_1} \right] = 0.1 \left[ \frac{(5 - 0.2 - 0.7) \, V}{4 \, k\Omega} \right] = 102.5 \, \mu A$$

The total load current at low state for $N = 4$ is

$$I_{LO} = 4 \times 102.5 \, \mu A = 410 \, \mu A$$
Step 5. Calculate the high-state $I_{B2}$, $I_{C2}$, and $I_{E2}$. The high-state $I_{B2}$ is

$$I_{B2} = -I_{C1} = (1 + \beta_R)I_{B1} = (1 + 0.1) \times 725 \, \mu A = 798 \, \mu A$$

The high-state $I_{C2}$ is

$$I_{C2} = \frac{V_{CC} - V_{B3(sat)} - V_{CE2(sat)}}{R_2} = \frac{(5 - 0.7 - 0.2) \, V}{3.79 \, k\Omega} = 1082 \, \mu A$$

The high-state $I_{E2}$ is

$$I_{E2} = I_{B2} + I_{C2} = 798 \, \mu A + 1082 \, \mu A = 1.88 \, mA$$

Step 6. Assume that most of $I_{E2}$ flows through the base of Q3 rather than through $R_B$. Let $I_R = 0.4I_{E2} = 0.4 \times 1.88 \, mA = 0.75 \, mA$. Then calculate the value of $R_B$:

$$R_B = \frac{V_{B3(sat)}}{I_R} = \frac{0.7 \, V}{0.75 \, mA} = 933 \, \Omega$$

Choose $R_B = 1 \, k\Omega$.

Step 7. Calculate the maximum permissible collector low-state current to drive Q3 in saturation:

$$I_{B3} = I_{E2} - I_R = 1.88 \, mA - 0.75 \, mA = 1.13 \, mA$$

$$I_{C3(max)} = \beta_R I_{B3} = 10 \times 1.13 \, mA = 11.3 \, mA$$

which is higher than $I_{L0} = 410 \, \mu A$, from step 4. Thus, the design should be satisfactory. (Otherwise steps 1 through 7 should be repeated with a lower value of $N$, a higher value of $\beta_{F(sat)}$, or a lower value of $I_{N1}$.) The overdrive factor is

$$k_{ODF} = \frac{I_{C3(max)}}{I_{L0}} = \frac{11.3 \, mA}{410 \, \mu A} = 27.6$$

Step 8. Calculate the base current of Q4 at output high:

$$I_{B4} = \frac{NI_{N1}}{1 + \beta_R} = \frac{4 \times 72.5 \, \mu A}{1 + 10} = 26.4 \, \mu A$$

Step 9. Using Eq. (15.92), calculate the value of $R_4$ for $V_O = V_{OH}$:

$$R_4 = \frac{V_{CC} - V_{OH} - V_{CE4(sat)} - V_D1}{I_{C4(max)}} = \frac{(5 - 3.5 - 0.2 - 0.7) \, V}{1 \, mA} = 600 \, \Omega$$

The power rating of $R_4$ is $P_{R4} = (1 \, mA)^2R_4 = (1 \, mA)^2 \times 600 = 0.6 \, mW$.

Step 10. Calculate the power rating of $R_2$ due to $I_{C2}$ or $I_{B2}$, whichever has the higher value. Since $I_{C2} = 1082 \, \mu A$ and $I_{B2} = 798 \, \mu A$, use $I_{C2}$.

$$P_{R2} = (1.082 \, mA)^2R_2 = (1.082 \, mA)^2 \times 3.79 \, k\Omega = 4.44 \, mW$$
Step 11. Calculate the power rating of $R_1$ due to $I_{B1}$ at output high or $I_{B1}$ at output low, whichever has the higher value. Since the base current of $Q_1$ at output high is

$$I_{B1} = \frac{V_{CC} - V_{BE2(sat)}}{R_1} = \frac{(5 - 0.7) \text{ V}}{4 \text{ k\Omega}} = 1.075 \text{ mA}$$

and $I_{B1}$ at output low is 725 $\mu$A (from step 3), use $I_{B1}$ at output high.

$$P_{R1} = (1.075 \text{ mA})^2 R_2 = (1.075 \text{ mA})^2 \times 4 \text{ k\Omega} = 4.6 \text{ mW}$$

(b) Each transistor has junction capacitances that will affect the switching speed. To examine the propagation time, we will assume that a load resistance of

$$R_L = \frac{V_{OH}}{I_{L1}} = \frac{3.5 \text{ V}}{4 \times 72.5 \mu\text{A}} = 12 \text{ k\Omega}$$

and an equivalent capacitance of

$$C_L = 4 \times 5 \text{ pF} = 20 \text{ pF}$$

is connected to the output.

The PSpice schematic is shown in Fig. 15.41. The PSpice plots of the characteristics, which are shown in Fig. 15.42, give $V_{OH} = 3.53 \text{ V}$ (expected value is 3.5 V), $V_{IL} = 0.6 \text{ V}$ at $V_O = 3.49 \text{ V}$, $V_{OL} = 42 \text{ mV}$ at $V_O = 5 \text{ V}$, and $V_{IH} = 1.5 \text{ V}$ at $V_O = 62 \text{ mV}$.

$$N_{ML} = V_{IL} - V_{OL} = 0.6 - 0.042 \approx 0.56 \text{ V}$$

$$N_{MH} = V_{OH} - V_{IH} = 3.53 - 1.5 = 2.03 \text{ V}$$

$t_{pHL} = 4.53 \text{ ns}$, $t_{pLH} = 6.68 \text{ ns}$, and $t_{pd} = 5.61 \text{ ns}$. 

![PSpice schematic for Example 15.7](image-url)
15.11.2 High-Speed TTL NAND Gates

Notice from Eq. (15.90) that the voltage drop across $R_2$ caused by the load current flow reduces the output high voltage $v_O$. The output voltage can be increased by replacing $Q_4$ in Fig. 15.38 by a Darlington pair, as shown in Fig. 15.43. Transistors $Q_4$ and $Q_5$ form the Darlington pair. The BEJ of $Q_5$ offers the voltage offset and serves the purpose of diode $D_1$ in Fig. 15.38. Resistor $R_5$ is included to aid the reverse-biased recovery current of $Q_4$. The operation and analysis of this gate are similar to those of the gate in Fig. 15.38.
The current through \( R_5 \) is

\[
I_{R5} = \frac{V_{B4}}{R_5} = \frac{V_O + V_{BE4}}{R_5}
\]

The emitter current of \( Q_5 \) is

\[
I_{E5} = I_{B4} + I_{R5}
\]

which gives its base current as

\[
I_{B5} = \frac{I_{E5}}{1 + \beta_{F5}} = \frac{I_{B4} + I_{R5}}{1 + \beta_{F5}} = \frac{I_{E4}}{(1 + \beta_{F4})(1 + \beta_{F5})} + \frac{I_{R5}}{1 + \beta_{F5}}
\]

Equation (15.90) can be modified to give the output voltage as

\[
V_{CC} = R_2I_{B5} + V_{BE4} + V_{BE5} + V_O
\]

or

\[
v_O = V_{CC} - (R_2I_{B5} + V_{BE4} + V_{BE5}) \quad (15.93)
\]

Also, resistance \( R_B \) is replaced by an active base recovery circuit consisting of \( R_3, R_6, \) and \( Q_6 \). This recovery circuit reduces the amount of diverting current at the base of \( Q_3 \). As a result, the VTC becomes sharper, the noise margin is enhanced, and the delay time is reduced. The recovery circuit and its equivalent are shown in Fig. 15.44[(a) and (b)]. \( I_{BR} \) is the base recovery current of \( Q_3 \).

Assuming that \( Q_6 \) is close to saturation but not saturated, the collector current of \( Q_6 \) is given by

\[
I_{C6} = \frac{I_{BR}}{1 + 1/\beta_{F6}}
\]

which, near the edge of saturation, must be equal to

\[
I_{C6} = \frac{V_{BE3} - V_{CE6(sat)}}{R_6}
\]

![FIGURE 15.44 Base recovery circuit](image-url)
Equating these two and assuming $\beta_F = \beta_F(\text{forced})$, we can find the value of $R_6$ as

$$R_6 = \frac{V_{BE3} - V_{CE6(sat)}}{I_{BR}/(1 + 1/\beta_F(\text{forced}))} \quad (15.94)$$

Suppose that $R_6$ is chosen to yield the same value of $I_{BR}$ as was obtained when a passive resistance $R_B$ was used; that is, $I_{BR} = V_{BE3} / R_B$. Substituting into Eq. (15.94) yields

$$R_6 = \frac{R_B(V_{BE3} - V_{CE6(sat)})}{V_{BE3}/(1 + 1/\beta_F(\text{forced}))} \quad (15.95)$$

In summary, high-speed TTL gates have the following features:

1. Short propagation delay time
2. Low noise immunity
3. Sharp voltage transfer characteristic
4. High fan-out
5. Maximum output voltage limited to, typically, 3.6 V

**EXAMPLE 15.8**

*Designing a TTL NAND gate*

(a) Design the TTL NAND gate of the circuit in Fig. 15.43. It has two inputs and feeds four identical NAND gates. The output voltage at output high is $V_{OH} = 3.5 \text{ V}$, and $I_{N1} = 72.5 \mu\text{A}$. Assume $V_{CC} = 5 \text{ V}$, $V_D = 0.7 \text{ V}$, $V_{CE(sat)} = 0.2 \text{ V}$, $V_{BE} = 0.7 \text{ V}$, $\beta_F(\text{forced}) = 10$, $\beta_R = 0.1$, $I_{C3(sat)} = 1 \text{ mA}$, $I_{C4(max)} = 1 \text{ mA}$, and $I_{BR} = 750 \mu\text{A}$. Capacitance of each load is, $C_i = 5 \text{ pF}$.

(b) Use PSpice/SPICE to check your design by plotting the transfer characteristic. Determine $N_{IL}$ and $N_{IH}$.

Model parameters for transistors are

$$BN=10 \quad BR=0.1 \quad TF=0.1\text{NS} \quad TR=10\text{NS} \quad VJC=0.85 \quad VAF=50$$

**SOLUTION**

(a) The design steps are similar to those in Example 15.7. After completing steps 1, 6, and 9 in Example 15.7 to determine the values of $R_1$, $R_B$, and $R_4$, we follow these steps.

**Step 1.** Using Eq. (15.94), calculate the collector resistance of $Q_6$:

$$R_6 = \frac{V_{BE3} - V_{CE6(sat)}}{I_{BR}/(1 + 1/\beta_F)} = \frac{(0.7 - 0.2) \text{ V}}{0.75 \text{ mA}/(1 + 1/10)} = 733 \Omega$$

Choose $R_3 = 2R_6 = 2 \times 733 = 1466 \Omega$. 
Step 2. Choose a value of $I_{R5}$: Let $I_{R5} = 1130 \mu A$ (the same as the value for the TTL) $\approx 1.2 \ mA$. For $v_O = V_{OH}$, find the value of $R_5$:

$$R_5 = \frac{V_{OH} + V_{BE4}}{I_{R5}} = \frac{3.5 + 0.7}{1.2 \ mA} = 3.5 \ k\Omega$$

Step 3. Calculate the base current of Q4 at output high and the base current of Q5:

$$I_{B4} = \frac{N_H I_{N1}}{1 + \beta_{F4}} = \frac{4 \times 72.5 \ \mu A}{1 + 10} = 26.4 \ \mu A$$

$$I_{B5} = \frac{I_{B4} + I_{R5}}{1 + \beta_{F5}} = \frac{26.4 \ \mu A + 1.2 \ mA}{1 + 10} = 111.5 \ \mu A$$

Step 4. Using Eq. (15.93) for $v_O = V_{OH}$, find the value of $R_2$ that meets the output requirement:

$$R_2 = \frac{V_{CC} - V_{BE4} - V_{BE5} - V_{OH}}{I_{B5}} = \frac{(5 - 0.7 - 0.7 - 3.5) V}{111.5 \ \mu A} = 897 \ \Omega$$

Step 5. Using Eq. (15.89), calculate the value of $R_1$:

$$R_1 = \beta_F \left[ \frac{V_{CC} - 2V_{BE(sat)} - V_{BC1}}{I_{N1}} \right] = 0.1 \left[ \frac{(5 - 2 \times 0.7 - 0.7)}{725 \ \mu A} \right] = 4 \ k\Omega$$

Step 6. Using Eq. (15.86), find the value of $I_{B1}$ at output low, and calculate the high-state $I_{B2}$:

$$I_{B1} = \frac{V_{CC} - V_{BE2(sat)} - V_{BE3(sat)} - V_{BC1}}{R_1} = \frac{(5 - 0.7 - 0.7 - 0.7) V}{4 \ k\Omega} = 725 \ \mu A$$

$$I_{B2} = -I_{C1} = (1 + \beta_R)I_{B1} = (1 + 0.1) \times 725 \ \mu A = 798 \ \mu A$$

Step 7. Calculate the high-state $I_{C2}$, $I_{E2}$, and $I_{B3}$. The high-state $I_{C2}$ for low output is

$$I_{C2} = \frac{V_{CC} - V_{BE3(sat)} - V_{CE2(sat)}}{R_2} = \frac{(5 - 0.7 - 0.2)}{897} V = 4.57 \ mA$$

In the high state,

$$I_{E2} = I_{B2} + I_{C2} = 798 \ \mu A + 4.57 \ mA = 5.37 \ mA$$

For the base current,

$$I_{B3} = I_{E2} - I_R = 5.37 \ mA - 0.75 \ mA = 4.62 \ mA$$

Step 8. Calculate the maximum permissible collector low-state current to drive Q3 in saturation:

$$I_{C3(max)} = \beta_F(\text{forced})I_{B3} = 10 \times 4.62 \ mA = 46.2 \ mA$$
which is higher than $I_{L0} = 410$ $\mu$A, from step 4 of Example 15.7. Thus, the design should be satisfactory. (Otherwise, steps 1 through 8 should be repeated with a lower value of $N$, a higher value of $\beta_{F(sat)}$, or a lower value of $I_{N1}$.) The overdrive factor is

$$k_{ODF} = \frac{I_{C3(sat)}}{I_{L0}} = \frac{46.2 \text{ mA}}{410 \text{ $\mu$A}} = 113$$

which is high; that is, the fan-out can be much more than $N = 4$.

(b) To examine the propagation time, we will assume that a load resistance of

$$R_L = \frac{V_{OH}}{N I_{L0}} = \frac{3.5 \text{ V}}{4 \times 72.5 \text{ $\mu$A}} = 12 \text{ k$\Omega$}$$

and an equivalent capacitance of

$$C_L = 4 \times 5 \text{ pF} = 20 \text{ pF}$$

is connected to the output.

The PSpice schematic is shown in Fig. 15.45. The PSpice plots, which are shown in Fig. 15.46, give $V_{OH} = 3.3$ V (expected value is 3.5 V), $V_{IL} = 1.35$ V at $V_o = 3.3$ V, $V_{OL} = 62$ mV at $v_o = 3.3$ V, and $V_{IH} = 1.6$ V at $V_o = 62$ mV.

$$\text{NM}_L = V_{IL} - V_{OL} = (1.35 - 0.062) \text{ V} \approx 1.29 \text{ V}$$

$$\text{NM}_H = V_{OH} - V_{IH} = 3.3 - 1.6 = 1.7 \text{ V}$$

$t_{pHL} = 1.93$ ns, $t_{pLH} = (57.57 \text{ ns} - 51.77 \text{ ns}) = 5.8$ ns, and $t_{pd} = (t_{pHL} + t_{pLH}) / 2 = 3.87$ ns.
15.1.3 Schottky TTL NAND Gates

In TTL gates, the transistors are driven into saturation. Since the delay time of a TTL gate is a strong function of the storage time of the saturated transistors, a nonsaturating logic gate has an advantage. A Schottky clamped transistor, which is prevented from being driven into saturation, can switch faster than a saturated transistor. Schottky clamped transistors are incorporated in many transistor logic gates.

A Schottky clamped transistor is basically a bipolar transistor with a built-in clamped Schottky diode, as shown in Fig. 15.47(a). Its symbol is shown in Fig. 15.47(b). The forward voltage drop of a Schottky diode is low, typically 0.3 V. When transistor $Q_1$ is in its active region of operation, the base–collector junction is reverse biased. The clamped diode is also reverse biased and does not affect the circuit operation. $Q_1$ behaves as a normal $n^+ p n$ transistor. As transistor $Q_1$ goes into saturation, the base–collector junction becomes forward biased, and it is clamped to the 0.3-V Schottky diode voltage. The excess base current is shunted through the diode, and the transistor is prevented from going heavily into saturation.

Assuming that transistor $Q_1$ is clamped at the edge of saturation, $I_C = \beta I_B$. The diode current can be related to the input and base currents by

$$I_D = I_I - I_B = I_I - \frac{I_C}{\beta_F}$$

![FIGURE 15.47 Schottky clamped transistor](image)

**FIGURE 15.46** PSpice plots for TTL characteristics (using transistor circuit model) for Example 15.8
Using KCL at the collector junction, we get

\[ I_C = I_D + I_L = I_t - \frac{I_C}{\beta_F} + I_L \]

which yields

\[ I_C = \frac{I_t + I_L}{1 + 1/\beta_F} \]

Thus, for an increased value of load current \( I_L \), the value of \( I_C \) is increased and the value of diode current \( I_D \) is reduced. That is, the major part of the input current is diverted into the base of the transistor, keeping the transistor at the edge of saturation. For a small value of load current, the value of \( I_C \) becomes small, and a large part of the input current is shunted through the diode. The base and diode currents change with the load conditions, while the transistor remains at the edge of saturation. The Schottky barrier diode has no minority carrier charge storage, and the transistor is never fully saturated. Thus, the recovery is very quick.

In the Schottky TTL NAND gate shown in Fig. 15.48, all of the transistors except \( Q_4 \) are Schottky clamped transistors. This circuit is similar to the one in Fig. 15.43. The two Schottky diodes from the input terminals to the ground act as clamps, to suppress any ringing that might occur from voltage transients and clamp any negative undershoots at approximately \(-0.3\) V.

The analysis of a Schottky TTL gate is similar to that of a standard TTL gate. When the output transistor \( Q_3 \) is on, \( V_{BE} = 0.7 \) V, the voltage drop across the Schottky diode is clamped to \( V_{BC} = 0.3 \) V, and

\[ V_{CE} = V_{CB} + V_{BE} = -V_{BC} + V_{BE} = -0.3 + 0.7 = 0.4 \) V

Thus, the output voltage of a Schottky gate in its output low state is slightly higher than the value of \( V_{CE(sat)} \) for standard TTL gates. The output voltage in the output high state is essentially the same as that of the standard TTL gate.
In summary, Schottky TTL gates have the following features:

1. Minimal delay time. Because a Schottky clamped transistor operates at a low saturation, the delay time is approximately 2 ns to 5 ns, compared to 10 ns to 15 ns in standard and high-speed TTL gates; that is, the propagation delay time is reduced by a factor of 5 to 10.
2. Sharper voltage transfer characteristic than standard TTL gates
3. Low noise immunity
4. Slightly higher output voltage in the output low state than standard TTL gates, typically 0.4 V

The success of the Schottky gate led to the development of other Schottky gates: low-power Schottky (LS), advanced Schottky (AS), and advanced low-power Schottky (ALS) gates.

**KEY POINT OF SECTION 15.11**

- The TTL families have gone through many developmental stages, which have resulted in improved switching speed and lower power dissipation. Applications focus on the low-power Schottky (LS) and advanced low-power Schottky (ALS) families because they offer the highest speed.

### 15.12 Emitter-Coupled Logic OR/NOR Gates

Storage time is the dominant parameter affecting the delay time of transistors that are driven into saturation. Storage time can be minimized by preventing the transistors from operating at saturation. Schottky TTL gates minimize transistor saturation by clamping the base–collector junction at the edge of the saturation condition. This significantly reduces the storage time and delay time, but slight saturation of the Schottky transistors adds to the switching time. The transistors in ECL gates are never driven into saturation, and so the storage time is virtually zero. The first ECL nonsaturated logic gates were introduced in 1962 by Motorola under the family name MECL I. Since then, the MECL gates have progressed through several generations: MECL II, MECL III, MECL 10K, and MECL 10KH.

An ECL uses an *emitter-coupled pair* as a current switch circuit, as shown in Fig. 15.49(a). It consists of two identical transistors $Q_1$ and $Q_2$, two matched resistors $R_C = R_{C1} = R_{C2}$, and a current

![Current switch circuit](image)

**FIGURE 15.49** Current switch circuit for ECL
source \( I_{EE} \). The input voltage \( v_I \), which is applied to the base of \( Q_1 \), is compared to the reference voltage \( V_{ref} \), which is applied to the base of \( Q_2 \). If \( v_I \) is greater than \( V_{ref} \) by a few hundred millivolts, the source current \( I_{EE} \) flows through \( Q_1 \). Thus, the output voltage becomes

\[
V_{O1} = \frac{v_{C1}}{R_{C1}} \]

as shown in Fig. 15.49(b). On the other hand, if \( v_I \) is less than \( V_{ref} \) by a few hundred millivolts, the source current \( I_{EE} \) flows through \( Q_2 \). The output voltage becomes

\[
V_{O2} = \frac{v_{C2}}{R_{C1}}
\]

as shown in Fig. 15.49(c). Thus, the input voltage \( v_I \) causes the current \( I_{EE} \) to flow through either \( Q_1 \) or \( Q_2 \).

An ECL OR/NOR gate, which is based on a differential pair, is shown in Fig. 15.50. Input transistors \( Q_1 \) and \( Q_3 \) are connected in parallel. If the differential input voltage \( v_d = v_A \) (or \( v_B \)) \(-\) \( V_{ref} \) is more than approximately 100 mV, the output voltage \( v_{O1} \) is directly proportional to \( v_d \). Similarly, the output voltage \( v_{O2} \) is directly proportional to the differential voltage \(-v_d = V_{ref} - v_A \) (or \( v_B \)). However, for the transistors to operate as switches, the differential voltage must be greater than approximately 120 mV. Transistors \( Q_4 \) with \( R_4 \) and \( Q_5 \) with \( R_5 \) operate as emitter followers. The collector terminals are normally placed at zero voltage because it can be proved analytically that placing the ground near the collectors of the transistors results in less noise sensitivity. For this reason, the supply voltages are generally \( V_{CC} = 0 \) and \( V_{EE} = -5.2 \text{ V} \).

The reference circuit consists of resistors \( R_1, R_2, \) and \( R_3 \), diodes \( D_1 \) and \( D_2 \), and transistor \( Q_6 \). The diodes \( D_1 \) and \( D_2 \) provide temperature compensation for the BEJ of \( Q_6 \). Neglecting the base current of \( Q_6 \), we get

\[
I_1 = I_2 = \frac{V_{CC} - 2V_D - V_{EE}}{R_1 + R_2}
\]

The voltage at the base of transistor \( Q_6 \) is

\[
V_{BE6} = V_{CC} - I_1R_1
\]

which gives the reference voltage as

\[
V_{ref} = V_{BE6} - V_{BE6}
\]
The emitter current of Q6 is given by

\[ I_{E6} \approx \frac{V_{\text{ref}} - V_{EE}}{R_3} \]

The circuit operation can be divided into two modes. During mode 1, either input \( v_A \) or input \( v_B \) is at logic high (\( V_{OH} \)); that is, \( v_A \) (or \( v_B \)) = \( V_{CC} - 0.7 \) V. The OR logic is at the \( v_{O2} \) output, and the NOR logic is at the \( v_{O1} \) output. Transistor Q2 is off. That is, \( i_{C2} = 0 \), and \( v_{O2} = V_{CC} \). The output voltage \( V_{OR} \) is given by

\[ V_{OR} = V_{O2} - 0.7 \text{ V} = V_{CC} - 0.7 \text{ V} \]

If \( v_A = V_{OH} = V_{CC} - 0.7 \) V, then Q1 is on, and we get

\[ V_E = v_A - V_{BE1} \]

which gives the emitter current \( I_E \) as

\[ I_E = \frac{V_E - V_{EE}}{R_E} \]

Assuming that \( i_{C1} \approx I_E \), the voltage \( v_{O1} \) can be found from

\[ v_{O1} = V_{CC} - i_{C1}R_{C1} \]

and the output \( V_{NOR} \) becomes

\[ V_{NOR} = v_{O1} - V_{BE5} \]

The current \( I_{E4} \), which is the emitter current of Q4, is

\[ I_{E4} = \frac{V_{E4} - V_{EE}}{R_4} = \frac{V_{OR} - V_{EE}}{R_4} \]

and the emitter current of Q5 is

\[ I_{E5} = \frac{V_{E5} - V_{EE}}{R_5} = \frac{V_{NOR} - V_{EE}}{R_5} \]

During mode 2, inputs \( v_A \) and \( v_B \) are at logic low (\( V_{OL} \)). Transistors Q1 and Q3 are off; that is, \( i_{C1} = 0 \), and \( v_{O1} = V_{CC} \). The output voltage \( V_{NOR} \) is given by

\[ V_{NOR} = V_{OH} = v_{O1} - 0.7 \text{ V} = V_{CC} - 0.7 \text{ V} \]

Transistor Q2 is on, and we get

\[ V_E = V_{\text{ref}} - V_{BE2} \]

which gives the emitter current \( I_E \) as

\[ I_E = \frac{V_E - V_{EE}}{R_E} \]
Assuming that $i_{C2} \approx I_E$, the voltage $v_{O2}$ becomes

$$v_{O2} = V_{B4} = V_{CC} - i_{C2}R_{C2}$$

and the output $V_{OR}$ is

$$V_{OR} = v_{O2} - V_{BE4}$$

The current $I_{E4}$, which is the emitter current of $Q_4$, is

$$I_{E4} = \frac{V_{E4} - V_{EE}}{R_4} = \frac{V_{OR} - V_{EE}}{R_4}$$

and the emitter current of $Q_5$ is

$$I_{E5} = \frac{V_{E5} - V_{EE}}{R_5} = \frac{V_{NOR} - V_{EE}}{R_5}$$

Defining logic high as

$$V_{OH} = V_{CC} - 0.7 = -0.7 \text{ V}$$

and substituting the numerical values, we get the logic low as

$$V_{OL} = v_{O1} - V_{BE5} = V_{CC} - i_{C1}R_{C1} - V_{BE5} = V_{NOR} = -1.63 \text{ V}$$

gives the input and output voltages shown in Table 15.7.

In summary, ECL OR/NOR gates have the following features:

1. No transistor saturation and negligible delay time
2. High power dissipation, typically 50 mW to 70 mW (compared to 2 mW to 10 mW for Schottky TTL circuits)
3. Availability of complementary outputs, which eliminates the need to include separate inverters to provide these complementary outputs
4. High fan-out, typically in the range of 50 to 100
5. Low noise margin, since the logic high and logic low output voltages are only approximately $-0.7 \text{ V}$ and $-1.63 \text{ V}$, respectively
6. Sharp voltage transfer characteristic

**TABLE 15.7** ECL/OR gate logic function

<table>
<thead>
<tr>
<th>$v_A$</th>
<th>$v_B$</th>
<th>$V_{OR}$</th>
<th>$V_{NOR}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_{OL}$  (for 0)</td>
<td>$V_{OL}$ (for 0)</td>
<td>$V_{OL}$ (for 0)</td>
<td>$V_{OH}$ (for 1)</td>
</tr>
<tr>
<td>$V_{OH}$  (for 1)</td>
<td>$V_{OL}$ (for 0)</td>
<td>$V_{OH}$ (for 1)</td>
<td>$V_{OL}$ (for 0)</td>
</tr>
<tr>
<td>$V_{OL}$  (for 0)</td>
<td>$V_{OH}$ (for 1)</td>
<td>$V_{OH}$ (for 1)</td>
<td>$V_{OL}$ (for 0)</td>
</tr>
<tr>
<td>$V_{OH}$  (for 1)</td>
<td>$V_{OH}$ (for 1)</td>
<td>$V_{OH}$ (for 1)</td>
<td>$V_{OL}$ (for 0)</td>
</tr>
</tbody>
</table>
EXAMPLE 15.9

Designing an ECL OR/NOR gate

(a) Design the ECL OR/NOR gate of the circuit in Fig. 15.50. It has two inputs. The desired collector currents are $I_{C1} = 3$ mA and $I_{C4} = I_{C5} = 3$ mA. At logic high, $V_{OR} = -0.7$ V, $V_{NOR} = -1.63$ V, and $v_A$ (or $v_B$) = $V_{OR} = -0.7$ V. Assume $V_{BE} = 0.7$ V, $V_{EE} = -5.2$ V, $V_{BE} = 0.7$ V, and $\beta_F = 100$.

(b) Calculate the maximum fan-out with similar ECL gates if $V_{OR}$ is allowed to fall from $V_{OR(max)} = 0.7$ V to $V_{OR(min)} = 0.75$ V.

(c) Use PSpice/SPICE to check your design by plotting the transfer function. Calculate NM_L and NM_H. Use PSpice/SPICE to find $P_{static}$ if $v_A = v_B = -0.7$ V.

SOLUTION

(a) Assuming $v_A$ (or $v_B$) = $V_{OR} = -0.7$ V, the steps to complete the design are as follows:

Step 1. Calculate $V_{CC}$, $v_A$, and $V_E$. The value of $V_{CC}$ is

$$V_{CC} = V_{OR} + 0.7 = -0.7 + 0.7 = 0$$

The input voltage for logic high is

$$v_A = V_{OH} = V_{CC} - 0.7 = 0 - 0.7 = -0.7$$

The emitter voltage is

$$V_E = v_A - V_{BE1} = -0.7 - 0.7 = -1.4$$

Step 2. Calculate the emitter resistance $R_E$:

$$R_E = \frac{V_E - V_{EE}}{I_E \approx I_C} = \frac{-1.4 V - (-5.2 V)}{3 mA} = \frac{(-1.4 + 5.2) V}{3 mA} = 1.27 \, k\Omega$$

Step 3. Calculate the voltage $v_{O1}$, the collector resistance $R_{C1}$, $R_4$, and $R_5$:

$$v_{O1} = V_{NOR} + V_{BES} = -1.63 + 0.7 = -0.93$$

$$R_{C1} = \frac{V_{CC} - v_{O1}}{I_{C1}} = \frac{V_{CC} - v_{O1}}{I_E} = \frac{0 - (-0.93) V}{3 mA} = \frac{0.93 V}{3 mA} = 310 \, \Omega$$

$$R_4 = \frac{V_{OR} - V_{EE}}{I_{E4} \approx I_{C3}} = \frac{-0.7 V - (-5.2 V)}{3 mA} = 1.5 \, k\Omega$$

$$R_5 = \frac{V_{NOR} - V_{EE}}{I_{ES} \approx I_{C3}} = \frac{-1.63 V - (-5.2 V)}{3 mA} = 1.19 \, k\Omega$$
Step 4. Since the input voltages $v_A$ and $v_B$ are greater than $V_{ref}$ when the circuit is in the logic high state and less than $V_{ref}$ when it is in the logic low state, set $V_{ref}$ at the midpoint between the logic low and logic high levels and calculate the voltage at the base of $Q_6$:

$$V_{ref} = \frac{V_{NOR} + V_{OR}}{2} = \frac{-1.63 - 0.7}{2} = -1.165 \text{ V}$$

$$V_{B6} = V_{ref} + V_{BE6} = -1.165 + 0.7 = -0.465 \text{ V}$$

Step 5. Calculate the values of $R_1$, $R_2$, and $I_1$. Using KVL, we can write

$$I_1(R_1 + R_2) = -V_{EE} - 2V_D$$

$$I_1R_1 = -V_{B6}$$

which gives

$$\frac{I_1(R_1 + R_2)}{I_1R_1} = \frac{-V_{EE} - 2V_D}{-V_{B6}}$$

or

$$1 + \frac{R_2}{R_1} = \frac{-V_{EE} - 2V_D}{-V_{B6}} = \frac{-(5.2) - 2 \times 0.7}{-(-0.465)} = 8.172$$

Choose $R_1 = 300 \text{ Ω}$. Then

$$R_2 = (8.172 - 1)R_1 = (8.172 - 1) \times 300 = 2.15 \text{ kΩ}$$

Calculate the value of $I_1$ as

$$I_1 = I_2 = \frac{V_{B6}}{R_1} = \frac{-0.465}{300} = 1.55 \text{ mA}$$

Step 6. Calculate the value of $R_3$. For good temperature compensation, the current through the emitter of $Q_6$ should be the same as the current through diodes $D_1$ and $D_2$; that is,

$$I_3 = I_2 = \frac{V_{ref} - V_{BE}}{R_3} = \frac{-1.165 - (5.2)}{R_3} = 1.55 \text{ mA}$$

or $R_3 = 2.6 \text{ kΩ}$.

Step 7. Choose $R_{C2}$ to be slightly more than $R_{C1}$ (approximately 3% more for resistors of 1% tolerance)—that is, 2% more than the tolerance value. Thus,

$$R_{C2} = 1.03 \times R_{C1} = 1.03 \times 310 = 319.3 \text{ Ω}$$

Let $R_{C2} = 320 \text{ Ω}$. 
The fan-out can be determined from Fig. 15.51, which shows the emitter-follower output stage of an ECL circuit driving a difference amplifier input stage of an ECL load. This circuit is shown for \( V_{OR} \) at the logic high level. The load transistor \( Q_1 \) is on, and the load emitter current is given by

\[
I_{E} = \frac{V_{OR(max)} - V_{BE} - V_{EE}}{R_E}
\]

\[
= \frac{-0.7 \text{ V} - 0.7 \text{ V} - (-5.2 \text{ V})}{1.27 \text{ k}\Omega} = 2.99 \text{ mA}
\]

The input base current for each fan-out is given by

\[
I_{N1} = \frac{I_E}{1 + \beta_F} = \frac{2.99 \text{ mA}}{1 + 100} = 29.6 \mu\text{A}
\]

Thus, the total load current is

\[
I_{L1} = NI_{N1} = N \times 29.6 \mu\text{A}
\]

The emitter current of \( Q_4 \) is

\[
I_{E4} = \frac{V_{OR(min)} - V_{EE}}{R_E} = \frac{-0.75 \text{ V} - (-5.2 \text{ V})}{1.5 \text{ k}\Omega} = 2.97 \text{ mA}
\]

The base current \( I_{B4} \) required to supply the load current \( I_L \) and the current \( I_{E4} \) is

\[
I_{B4} = \frac{I_{E4} + I_L}{1 + \beta_F} = \frac{V_{CC} - V_{BE} - V_{OR(min)}}{R_{C2}}
\]

which gives

\[
\frac{2.97 \text{ mA} + (N \times 29.6 \mu\text{A})}{1 + 100} = \frac{0 - 0.7 - (-0.75)}{320}
\]

or \( N = 432 \).
(c) The PSpice schematic is shown in Fig. 15.52. The PSpice plots shown in Fig. 15.53 give 
\( V_{OH} = -0.81 \) V (expected value is \(-0.7 \) V), \( V_{IL} = 0.95 \) V at \( V_O = -0.855 \) V, \( V_{OL} = -1.59 \) V at \( V_I = 2 \) V, and \( V_{IH} = 1.33 \) V at \( V_O = -1.55 \) V. Note that \( V(11) \) and \( V(12) \) are the voltages at the OR and NOR terminals, respectively.

\[
\begin{align*}
NM_L &= V_{IL} - |V_{OL}| = 0.95 - 1.59 = -0.64 \text{ V} \\
NM_H &= |V_{OH}| - V_{IH} = 0.81 - 1.33 = -0.52 \text{ V}
\end{align*}
\]

\( t_{pHL} = 1.05 \) ns, \( t_{pLH} = 1.22 \) ns, and \( t_{pd} = 1.13 \) ns. The PSpice output file gives \( P_{\text{static}} = 62 \) mW.

---

**FIGURE 15.52** PSpice schematic for Example 15.9

**FIGURE 15.53** PSpice plots for ECL characteristics (using transistor circuit model) for Example 15.9

(a) VTC  
(b) Transient response
15.13 BiCMOS Inverters

The CMOS inverter is a low-power, compact inverter that exhibits a high input resistance. Its bipolar circuits, however, have low propagation delays. A BiCMOS combines a CMOS with a BJT output buffer and thus incorporates the best features of both technologies. A BiCMOS inverter is shown in Fig. 15.54. The BJT totem-pole output stage provides the high current capability to charge the load capacitance $C_L$ rapidly while maintaining the low-power advantage of the CMOS. In practice, $R_1$ and $R_2$ are polysilicon resistors or MOS resistors. To examine the charging and discharging of $C_L$, we will divide the operation into two modes.

During mode 1, the input voltage $v_I$ is low (at $V_{OL}$). The NMOS $M_N$ is off. $Q_1$ is also off because its base is effectively grounded through $R_1$. The PMOS $M_P$ is on, with $v_{GS} = -V_{DD}$, and it provides the base current to $Q_2$. Thus, the load capacitance $C_L$ is charged up through $Q_2$ (in the active region) to approximately $V_{DD} - V_{BE2}$. At $v_O = V_{DD} - V_{BE2}$, $Q_2$ is cut off, and $C_L$ then continues to charge toward $V_{DD}$ through $M_P$ and $R_2$. Thus, $V_{BE2}$ decreases to approximately 0, and $v_O \approx V_{OH} = V_{DD}$. The base of $Q_2$ also discharges through $R_2$.

During mode 2, the input voltage $v_I$ goes high (to $V_{OH}$). Both $M_P$ and $Q_2$ are off. $M_N$ is turned on. Capacitor $C_L$ discharges through $R_2$ and $M_N$. As a result of the current flow through $R_1$, $Q_1$ is turned on, and the collector current of $Q_1$ causes $C_L$ to discharge rapidly until $V_{BE1}$ drops below cut-in. After that, $Q_1$ turns off, and $C_L$ continues to discharge through $R_2$, $M_N$, and $R_1$ to approximately zero. That is, $V_{OL} \approx 0$. The base of $Q_1$ also discharges through $R_1$.

![BiCMOS inverter](image)
15.13.1 Propagation Delay

The propagation delay time of a BiCMOS is reduced because of the increased charging and discharging currents available from the BJTs. When the input switches from logic high to logic low, M_P supplies the charging current \( I_B \) for Q2. Because of Q1, the minimum value of \( v_I \) is \( V_{BE} \). Thus, \( V_{SGP} = V_{DD} - V_{BE} \), and \( I_B \) is given by

\[
I_B = K_p(V_{DD} - V_{BE} - |V_{IP}|)^2
\]

The charging current supplied by Q2 in the active mode is

\[
I_E = \beta_F I_B
\]

Thus, the time \( t_{PLH} \) required to charge \( C_L \) from \( V_{OL} \) to \((V_{OL} + V_{OH})/2 \approx V_{DD}/2 \) (neglecting the junction capacitances of the transistors) is given by

\[
t_{PLH} \approx \frac{(V_{OL} + V_{OH})C_L}{2I_E} = \frac{V_{DD}C_L}{2\beta_F K_p(V_{DD} - V_{BE} - |V_{IP}|)^2}
\]  \hspace{1cm} (15.96)

If the input switches from logic low to logic high, M_N will provide the discharging path of \( C_L \). Thus, the time \( t_{PHL} \) to discharge \( C_L \) from \( V_{OH} = (V_{OL} + V_{OH})/2 \approx V_{DD}/2 \) to \( V_{OL} \) is given by

\[
t_{PHL} = \frac{(V_{OL} + V_{OH})C_L}{2I_E} = \frac{V_{DD}C_L}{2\beta_F K_n(V_{DD} - V_{BE} - |V_{IP}|)^2}
\]  \hspace{1cm} (15.97)

When the BJTs are identical and the threshold voltages of the MOSFETs have the same magnitude, \( t_{PLH} = t_{PHL} \) and \( t_{pd} = t_{PLH} \).

**EXAMPLE 15.10**

**Finding the propagation delays of a BiCMOS inverter**  The BiCMOS inverter shown in Fig. 15.54 has \( V_{DD} = 5 \) V, \( V_{BE} = 0.7 \) V, \( R_1 = 1 \) kΩ, \( R_2 = 4 \) kΩ, \( C_L = 5 \) pF, and \( \beta_F = 100 \). The threshold voltages are \( V_{IP} = -1 \) V, \( V_{IN} = 1 \) V, and \( K_p = 20 \) μA/V². Assume \( V_{OL} = 0 \) and \( V_{OH} = 5 \) V.

(a) Calculate \( t_{PLH} \), \( t_{PHL} \), and \( t_{pd} \).

(b) Use PSpice/SPICE to plot the VTC and transient response.

**SOLUTION**

(a) Using Eq. (15.96), we have

\[
t_{PLH} = \frac{5 \times 0.5 \text{ pF}}{2 \times 100 \times 20 \text{ μA/V}^2 \times (5 \text{ V} - 0.7 \text{ V} - 1 \text{ V})^2} = 0.06 \text{ ns}
\]

\[
t_{pd} = t_{PHL} = t_{PLH} = 0.06 \text{ ns}
\]

Although \( t_{PLH} \) is independent of the \( L/W \) ratio of the MOSFETs, in PSpice its value will depend on the ratio \( L/W \). This causes the difference between the calculated value in part (a) and the PSpice value in part (b).
The PSpice schematic is shown in Fig. 15.55. The PSpice plot of the VTC, shown in Fig. 15.56(a), gives $V_{IL} = 2.125$ V, $V_{IH} = 2.875$ V, $V_{OL} = 0$ at $V_{OH} = 5$ V, and $V_{I(tran1)} = 2.5$ V (expected value is 2.5 V). The transient response is shown in Fig. 15.56(b), which gives $t_{pHL} = 2.93$ ns, $t_{pLH} = 0.65$ ns, and $t_{pd} = 1.79$ ns, which is lower than $t_{pd} = 3.91$ ns for the CMOS in Example 15.5.

**Figure 15.55** PSpice schematic for Example 15.10

(b) The PSpice schematic is shown in Fig. 15.55. The PSpice plot of the VTC, shown in Fig. 15.56(a), gives $V_{IL} = 2.125$ V, $V_{IH} = 2.875$ V, $V_{OL} = 0$ at $V_{OH} = 5$ V, and $V_{I(tran1)} = 2.5$ V (expected value is 2.5 V). The transient response is shown in Fig. 15.56(b), which gives $t_{pHL} = 2.93$ ns, $t_{pLH} = 0.65$ ns, and $t_{pd} = 1.79$ ns, which is lower than $t_{pd} = 3.91$ ns for the CMOS in Example 15.5.

**Figure 15.56** PSpice plots for BiCMOS inverter (using transistor circuit model) for Example 15.10

NOTE: Figure 15.56 illustrates the very sharp rise and fall times that occur while the BJTs are conducting. The output signal is nearly $V_{DD}$, and the noise margins for this circuit are high, both being more than 2 V.

**KEY POINT OF SECTION 15.13**

- A BiCMOS combines a CMOS with a BJT output buffer and thus incorporates the best features of both technologies.
15.14 Interfacing of Logic Gates

In practice, the load of a logic gate is another logic gate; that is, one gate acts as the driver and another one as the load. The interfacing of different logic gates requires that the circuits operate at a common supply voltage and have logic-level compatibility. Also, the devices must maintain safe power dissipation levels and good noise immunity over the required operating temperatures.

The voltage characteristics required at the output and input terminals of TTL families are shown in Fig. 15.57(a) for $V_{CC} = 5$ V; the voltage characteristics for CMOS families are shown in Fig. 15.57(b). The CMOS gates are designed to switch states at higher voltage levels than are the TTL gates. In interfacing one type of logic gate with another, attention should be given to the logic swing, output drive capability, DC input current, noise immunity, and speed of each type. The typical values of interfacing parameters such as $V_{OL}$, $V_{OH}$, $I_{OL}$, $I_{OH}$, $V_{IL}$, $V_{IH}$, $I_{IL}$, and $I_{IH}$ are shown in Table 15.3 for CMOS and TTL devices.

15.14.1 TTL Driving CMOS

When a TTL device is used to drive a CMOS, the output drive capability of the driving device and the switching levels and input currents of the driven devices are important considerations. The input currents for a CMOS are very small in both the 1 and the 0 state, typically $I_{IL} = I_{IH} = 10$ pA, and the thresholds for a CMOS are typically $V_{IL(max)} = 1.5$ V and $V_{IH(min)} = 3.5$ V. Thus, to obtain some noise immunity, the output of the TTL driver must be no more than $V_{OL} = 1.5$ V at logic 0 and no less than $V_{OH} = 3.5$ V at logic 1. Depending on whether the device is in the 0 or the 1 state, the driver will be sinking or sourcing current.

![Input and output characteristics of logic gates](image-url)
**Current Sinking**

When the output of the TTL device is in the low state (at $V_{OL}$), the collector of Q1, shown in Fig. 15.58(a), is essentially at ground potential. Transistor Q1 must go into saturation to ensure a stable 0 level, typically 0.3 V. To attain this voltage level, there should be a high impedance path from the output to the power supply $V_{CC}$. Since the CMOS has extremely high input impedances, typically $10^{11} \, \Omega$, it will not contribute any significant current through Q1. Thus, the current-sinking capability is not a problem. The voltage level is not a problem either, as CMOS devices have high noise immunity, usually greater than 1 V.

**Current Sourcing**

When Q1, shown in Fig. 15.58(b), is off (at logic 1 output $V_{OH}$), there is a current flow from the $V_{CC}$ terminal of the driver, through a pull-up resistor $R_P$, into the input stages of the load; that is, the driving gate acts as the current source for the load. The total load current ($N_{IH}$) should not reduce the output level below $V_{IH}$ required by the CMOS load. A driver with a built-in pull-up resistor $R_P$, as shown in Fig. 15.58(b), presents no problem in the interface with the CMOS. However, a driver with an open collector requires an external pull-up resistor $R_P$, as shown in Fig. 15.58(c).

![Figure 15.58](image-url)
Consider the diode-transistor arrangement shown in Fig. 15.58(d), which will reduce the output voltage $v_O$ given by

$$v_O = V_{CC} - V_{BE} - V_D - \frac{i_0 R_B}{\beta_F}$$  \hspace{1cm} (15.98)

Depending on the load current $i_0$, the minimum output level of $V_{OH} = 2.4 \, \text{V}$ may not ensure an acceptable state of $V_{IH(min)} = 3.5 \, \text{V}$ for the CMOS device, and this could cause a problem in the logic 1 state. However, the minimum level of $V_{OH} = 2.4 \, \text{V}$ for a TTL gate is often specified at a given load current ($I_{OH} = 200 \, \mu \text{A}$). Since the CMOS draws very little current, an output level of $V_{OH} = 3.5 \, \text{V}$ will be available, but there is no noise immunity. Therefore, a pull-up resistor $R_P$ should be added from the output terminal of the driver, as shown in Fig. 15.58(e). The minimum value of $R_P$ can be found from

$$R_{P(min)} = \frac{V_{DD} - V_{OL(max)}}{I_{OL} - N I_{IL}}$$  \hspace{1cm} (15.99)

where $N$ is the number of CMOS load gates. The maximum value of the external pull-up resistor can be found from

$$R_{P(max)} = \frac{V_{CC} - V_{IH(min)}}{M I_{CEX(max)} - N I_{IH}}$$  \hspace{1cm} (15.100)

where $M$ is the number of TTL drivers and $I_{CEX(max)}$ is the maximum collector–emitter leakage current of $Q_1$ in the high state.

As an example, consider a case where $V_{CC} = V_{DD} = 5 \, \text{V}$, $V_{OL} = 0.4 \, \text{V}$, $V_{IH} = 3.5 \, \text{V}$, $M = N = 1$, $I_{OL} = 2 \, \text{mA}$ (for the TTL), $I_{IL} = 10 \, \text{pA}$ (for the CMOS), $I_{IH} = 10 \, \text{pA}$ (for the CMOS), and $I_{CEX} = 100 \, \mu \text{A}$ (for the TTL). From Eqs. (15.99) and (15.100), we get

$$R_{P(min)} = \frac{5 - 0.4}{2 \, \text{mA} - 1 \times 10 \, \text{pA}} \approx 2.3 \, \text{k}\Omega$$

$$R_{P(max)} = \frac{5 - 3.5}{1 \times 100 \, \mu \text{A} - 1 \times 10 \, \text{pA}} \approx 15 \, \text{k}\Omega$$

### 15.14.2 CMOS Driving TTL

If the CMOS device drives the TTL device, $Q_1$ in Fig. 15.58(a) will be replaced by a MOSFET. The current-sinking capability of the CMOS must be considered when it drives a medium-power TTL. The TTL device typically requires no more than $I_{IL} = 0.18 \, \text{mA}$ in the 0 input state and a maximum of $I_{IH} = 10 \, \mu \text{A}$ in the 1 input state. The CMOS must be capable of sourcing and sinking these currents while maintaining the voltage output levels required by the TTL gates.

**Current Sourcing**

In high-state operation, $V_{DD}$ is normally connected to the output through one or more $p$-channel devices, which must be able to source the total load leakage current of TTL load stages. The $I_{OH}$ of the CMOS stage must match the leakage currents $N I_{IH}$ (for the logic 1 state) with a fan-out of $N$. 
Current Sinking

When the output of the CMOS is in the low state (at $V_{OL}$), an $n$-channel device is on and the output is approximately at ground potential. The CMOS device sinks the current flowing from the TTL input load stage. The $I_{OL}$ of the CMOS stage must match the input currents $N_{IL}$ (for the logic 0 state) with a fan-out of $N$.

KEY POINT OF SECTION 15.14

The load of a logic gate is often another logic gate. It is essential that the logic gates be compatible in terms of supply voltage, logic levels, and noise immunity.

15.15 Comparison of Logic Gates

Broadly speaking there are three logic families: TTL, ECL, and CMOS. Each family has its advantages and disadvantages. The delay-power product (DP), which is the product of power dissipation $P_D$ and propagation delay $t_{pd}$, is the key parameter for high-speed switching. Figure 15.59 compares delay time with power dissipation for various logic families. Note that the first generation of the CMOS family is labeled

![Graph showing comparison of delay time and power dissipation for logic families](Image)

**FIGURE 15.59** Comparison of delay time and power dissipation for logic families
CMOS; HCMOS refers to high-speed CMOS and ACMOS to advanced-type CMOS. The diode-transistor logic (DTL) and resistor-transistor logic (RTL) gates were developed earlier and have the highest delay-power product. Integrated-injection logic (I\textsuperscript{2}L) gates have low power but are relatively slower. The MESGET (gallium arsenide) gates are extremely fast because of the high mobility of electrons in GaAs—typically five to six times higher than that of electrons in $n$-type silicon. Gate delays as low as 20 ps to 100 ps have been reported, but this speed must be weighed against the power dissipation, which is typically 10 mW.

The typical values of various parameters for the TTL, ECL, and CMOS families are shown in Table 15.8. We can draw the following conclusions from these data:

- The TTL (LS) family of logic gates has high speed and low power dissipation and is compatible with many applications. In addition, the technology is mature and hence low in cost.
- The ECL family has very high speed but at a great cost in power. The logic levels of these gates differ from those of other major families, and they require a negative supply.
- The CMOS family of logic gates requires almost zero power in standby and very low power at moderate switching rates. The speed of these gates is now comparable with that of the TTL. This is the technology of choice for most new designs, rivaling the TTL.

**KEY POINT OF SECTION 15.15**

- Different families of logic gates have different advantages and disadvantages that must be taken into account in design. Often a designer must make a trade-off between speed and power dissipation.

**15.16 Design of Logic Circuits**

In the design examples in the preceding sections, we calculated the component values of MOS and bipolar inverters and found their performance parameters, such as $V_{OL}$, $V_{OH}$, $t_{pd}$, $NM_L$, and $NM_H$. Since bipolar and MOS gates are available in IC circuits, designs are seldom done using discrete devices. However, these
examples provided an idea of the inside operation of the gates. Circuit design using gates usually involves applications of logic gates, and the following points should be kept in mind:

- Logic gates are highly nonlinear, and their output is considered as either high or low. The width of the undefined input voltage range should be kept to a minimum so that the noise margin becomes as large as possible.
- The output voltage levels of a gate must be compatible with the input voltage levels of the same or similar gates; that is, a design must address interfacing issues.
- The output of one gate must be capable of driving the inputs of more than one gate; that is, fan-out of a gate should be as large as possible for a circuit consisting of multiple gates.
- The logic gate should consume only as much power as is needed to operate at the required speed. Also, the DC supply must meet the design requirement.

EXAMPLE 15.11

Designing a clock circuit

(a) Design a clock circuit to produce a pulse output at a frequency of \( f_{\text{clk}} = 100 \text{ kHz} \). The DC supply voltage is 5 V.

(b) Use PSpice/SPICE to check your design by plotting the output voltage.

SOLUTION

(a) Given that \( T = 1 \ f_{\text{clk}} = 1 \ 100 \text{ kHz} = 10 \mu\text{s} \), the design steps are carried out as follows:

Step 1. Choose the circuit topology. Figure 15.60(a) shows a circuit that uses inverters; NOR gates could be used instead.

Step 2. Choose the gate types. Although use of CMOS inverters would simplify the analysis, we will use bipolar inverters of type 7404. We will derive a general expression for the clock frequency that can be applied to CMOS inverters also. From the data sheet, the inverter parameters are \( V_{\text{CC}} = 5 \text{ V}, \ V_{\text{OH}} = 3.4 \text{ V}, \ V_{\text{OL}} = 0.5 \text{ V}, \ V_{\text{IH}} = 2 \text{ V}, \ V_{\text{IL}} = 0.8 \text{ V}, \ I_{\text{IH}} = 0.1 \text{ mA}, \) and \( I_{\text{IL}} = 20 \mu\text{A} \).

Step 3. Analyze the circuit. Suppose that, at \( t = 0 \), \( V_{\text{O1}} \) (that is, the output of inverter \( U_1 \)) is high at \( V_{\text{OH}} \), \( v_\text{O} \) is low at \( V_{\text{OL}} \), and \( v_\text{I} \) (the input of inverter \( U_1 \)) is at \( V_{\text{IH}} = V_{\text{OH}} \) (from the previous cycle). Capacitor \( C \) will charge exponentially, with a time constant of \( \tau = RC \). The equivalent circuit during charging is shown in Fig. 15.60(b). The input voltage \( v_\text{I} \) will have a DC component and an exponentially decaying component. Thus,

\[
v_\text{I}(t) - V_{\text{OH}} = R_{\text{IL}} + [v_\text{I}(0^+) - v_\text{O}(0^+)]e^{-t/\tau} = R_{\text{IL}} + (V_{\text{IH}} - 2V_{\text{OH}})e^{-t/\tau} \quad \text{(for } 0 \leq t \leq T_2)\]

At \( t = T_2 \), \( v_\text{I} \) rises to \( V_{\text{IL}} \), and we can find \( T_2 \) from the condition \( v_\text{I}(t = T_2) = V_{\text{IL}} \):

\[
T_2 = RC \ln \left( \frac{V_{\text{IH}} - 2V_{\text{OH}}}{V_{\text{IL}} - V_{\text{OH}} - R_{\text{IL}}} \right) \quad (15.101)
\]
where $I_{IL}$ is the input current in the logic low state. When $v_I$ reaches $V_{IL}$ at $t = T_2$, $v_{O1}$ goes low to $V_{OL}$ and $v_O$ rises to $V_{OH}$ after a delay time of $t_{pd}$. The capacitor feeds this change in $v_O$ back to the input side. As a result, $v_I$ jumps from $V_{IL}$ to $V_{IL} + V_{OH}$.

Now, with $v_I = V_{IL} + V_{OH}$, $v_{O1} = V_{OL}$, and $v_O = V_{OH}$, the capacitor C will discharge exponentially, with a time constant of $\tau = RC$. The equivalent circuit during discharging is shown in Fig. 15.60(c). The input voltage $v_I$ will have a DC component of $V_{OL}$ and a decaying component. Redefining the time origin $t = 0$ (at $t = T_2$) gives the input voltage $v_I$:

$$v_I(t) = V_{OL} + [v_I(0^+) - v_O(0^+)]e^{-t/\tau} = V_{OL} + (V_{OH} - V_{IL} - V_{OL})e^{-t/\tau} \quad (for \ 0 \leq t \leq T_1)$$

At $t = T_1$, $v_I$ falls to $V_{IH}$, and we can find $T_1$ from the condition $v_I(t = T_1) = V_{IH}$:

$$T_1 = RC \ln \left( \frac{V_{OH} + V_{IL} - V_{OL}}{V_{IH} - V_{OL}} \right) \quad (15.102)$$

FIGURE 15.60  Pulse circuit using inverters
When $v_I$ falls to $V_{IH}$ at $t = T_1$, $v_{O1}$ becomes high at $V_{OH}$ and $v_O$ falls to $V_{OL}$ after a delay time of $t_{pd}$. Because the capacitor feeds this change in $v_O$ back to the input side, $v_I$ falls from $V_{IH}$ to $V_{IH} - V_{OH}$, and the cycle is repeated. The voltage waveforms are shown in Fig. 15.60(d). The period $T$ of the output voltage is given by

$$T = T_1 + T_2 = RC \left( \ln \frac{V_{OH} + V_{IL} - V_{OL}}{V_{IH} - V_{OL}} + \ln \frac{V_{IH} - 2V_{OH}}{V_{IH} - V_{OH} - RI_{IL}} \right)$$  \hspace{1cm} (15.103)

**NOTE:** For a CMOS gate, $V_{OL} = 0$, $V_{OH} = V_{DD}$, $I_{IL} = 0$, and $V_{IL} = V_{IH}$ (threshold voltages of the NMOS). Thus, Eq. (15.103) becomes

$$T = T_1 + T_2 = RC \left( \ln \frac{V_{DD} + V_{IL}}{V_{IH}} + \ln \frac{V_{IH} - 2V_{OH}}{V_{IH} - V_{OH}} \right)$$  \hspace{1cm} (15.104)

**Step 4.** Find the values of $R$ and $C$. Using Eq. (15.103), we get

$$10 \mu A = RC \left( \ln \frac{3.4 + 0.8 - 0.5}{2 - 0.5} + \ln \frac{2 - 2 \times 3.4}{0.8 - 3.4 - R \times 0.1 \text{ mA}} \right)$$

Choose a suitable value of $C$: Let $C = 1 \text{ nF}$. Solving for $R$ by iteration, we get $R \approx 8 \text{ k} \Omega$. Then $\tau = RC = 8 \mu s$, $T_1 = 7.22 \mu s$, and $T_2 = 2.76 \mu s$.

**b)** The PSpice simulation is shown in Fig. 15.61. The PSpice plots shown in Fig. 15.62 give $T = 7.53 \mu s$, $T_1 = 6.41 \mu s$, and $T_2 = 1.12 \mu s$. Thus, a higher value of $R$ (say, $R = 10 \text{ k} \Omega$) will be needed.
**Summary**

Electronic circuits are used to perform various logic functions, such as NOT, AND, OR, NOR, and NAND. The transistors are operated as on and off switches. This chapter analyzed the characteristics of NMOS and CMOS inverters, which are the basis of digital circuits and have applications in NAND and NOR gates. The use of a depletion-load transistor allows the logic level to be the same as the supply voltage and also gives the shortest NMOS switching times. NMOS and CMOS transmission gates are commonly used in steering logic function. The power consumption of CMOS gates is very small, essentially zero. In general, CMOS gates have many advantages over NMOS gates.

DTL gates are simple, but their switching speed is low because they include saturated transistors. TTL gates are better than DTL gates in terms of switching speed, component density, and fan-out. I^2L gates have higher component density, higher fan-out, and lower power dissipation than TTL gates. Schottky TTL gates minimize transistor saturation and have higher switching speeds than standard TTL gates. In ECL gates, the transistors are operated in the active region, and saturation is completely avoided, thereby reducing propagation delay time. However, ECL gates have more power dissipation than TTL and I^2L gates. Each gate has its advantages and limitations, and the applications engineer has to decide which types to use in a specific application.

**References**


**Review Questions**

1. What are the advantages and disadvantages of an NMOS inverter with a resistive load?
2. What are the intervals of operation of an NMOS inverter with a resistive load?
3. What are the advantages and disadvantages of an NMOS inverter with a saturated load?
4. What are the intervals of operation of an NMOS inverter with a saturated load?
5. What is the effect of the ratio $K_D / K_L$ on the transfer characteristic of an NMOS inverter with a saturated load?
6. What are the advantages and disadvantages of an NMOS inverter with a depletion load?
7. What are the intervals of operation of an NMOS inverter with a depletion load?
8. What is the effect of the ratio $K_D / K_L$ on the transfer characteristic of an NMOS inverter with a depletion load?
9. What is the function of an NMOS transmission gate?
10. What are the advantages and disadvantages of a CMOS inverter?
11. What are the intervals of operation of a CMOS inverter?
12. What is the function of a CMOS transmission gate?
13. What are the differences between CMOS and NMOS transmission gates?
14. What are the advantages and disadvantages of CMOS and NMOS gates?
15. What is the saturation of a transistor?
16. What is the overdrive factor of a transistor?
17. What is the forced $\beta$ of a transistor?
18. What is delay time?
19. What is the saturating charge of a transistor?
20. What is the storage time of a transistor?
21. What is the fan-out of a gate?
22. What is the noise margin of a gate?
23. What is the pull-down resistance of a gate?
24. What is a multiemitter transistor?
25. What are the advantages and limitations of a TTL gate?
26. What is a totem-pole output stage?
27. What is the active-biased recovery circuit?
28. What are the advantages and limitations of high-speed TTL gates?
29. What is a Schottky clamped transistor?
30. What are the advantages and limitations of Schottky TTL gates?
31. What are the advantages and limitations of ECL gates?
32. What is the purpose of connecting the collectors of an ECL gate to the ground?
33. What problems result from saturation in transistor gates?

**Problems**

The symbol **D** indicates that a problem is a design problem. The symbol **P** indicates that you can check the solution to a problem using PSpice/SPICE or Electronics Workbench.

**15.4 Performance Parameters of Logic Gates**

15.1 **D** An inverter, as shown in Fig. 15.8, drives identical inverters and has $V_{OL} = 0.4$ V, $I_{OL} = 1$ mA, $V_{OH} = 2.4$ V, $I_{OH} = 100$ $\mu$A, and $V_{CC} = 5$ V. The input currents drawn by each load inverter are $I_{IL} = 0.1$ mA (at logic low) and $I_{IH} = 10$ $\mu$A (at logic high).
   
   a. If there are three load inverters, determine the value of pull-up resistance $R_P$ that will ensure a logic 1 output of $V_{OH} = 2.4$ V.
   
   b. If $R_P = 2.5$ k$\Omega$, find the fan-out $N$.

15.2 The inverter shown in Fig. 15.10(a) has $V_{OL} = 0.4$ V, $V_{OH} = 2.4$ V, $V_{CC} = 5$ V, $R_{on} = 200$ $\Omega$, $R_{off} = \infty$, $R_P = 2.5$ k$\Omega$, $C_L = 2$ pF, and $f_{clk} = 100$ kHz.

   a. Find the delay times for the output voltage to rise from 0.4 V to 2.4 V and to fall from 5 V to 0.4 V.

   b. Find the power dissipation $P_D$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
15.5 **NMOS Inverters**

15.3 The parameters of the NMOS inverter in Fig. P15.3 are \( V_{IL} = 1.5 \text{ V} \), \( V_{DD} = 5 \text{ V} \), \( R_D = 1.5 \text{ k}\Omega \), and \( K_L = 0.5 \text{ mA/\text{V}^2} \).

a. Determine the input voltage at the transition point \( V_{I(tran)} \).

b. Calculate the output voltage \( V_O \), the drain current \( i_D \), and the power dissipation \( P_D \) for \( V_I = 2.5 \text{ V} \) and for \( V_I = 5 \text{ V} \).

**FIGURE P15.3**

15.4 The parameters of the NMOS inverter in Fig. P15.3 are \( V_{IL} = 1 \text{ V} \), \( V_{DD} = 5 \text{ V} \), and \( R_D = 1.5 \text{ k}\Omega \). Determine the conduction parameter \( K_L \) so that the output voltage \( V_O \) is 0.5 V at \( V_I = 5 \text{ V} \).

15.5 The parameters of the NMOS inverter in Fig. 15.12(a) with a saturated load are \( V_{IL} = V_{ID} = 1.5 \text{ V} \), \( V_{DD} = 5 \text{ V} \), \( K_L = 0.1 \text{ mA/\text{V}^2} \), and \( K_D = 50 \text{ \mu A/\text{V}^2} \).

a. Determine the input voltage at the transition point \( V_{I(tran)} \).

b. Calculate the output voltage, the drain current, and the power dissipation \( P_D \) for \( V_I = 2.5 \text{ V} \) and for \( V_I = 5 \text{ V} \).

15.6 The parameters of the NMOS inverter in Fig. 15.12(a) with a saturated load are \( V_{IL} = V_{ID} = 1 \text{ V} \) and \( V_{DD} = 5 \text{ V} \). Determine the ratio \( K_D \) \( K_L \) so that (a) \( V_O = 0.25 \text{ V} \) at \( V_I = 5 \text{ V} \) and (b) \( V_O = 0.25 \text{ V} \) at \( V_I = 4.5 \text{ V} \).

15.7 The parameters of the NMOS inverter in Fig. 15.12(a) with an enhancement load are \( V_{IL} = V_{ID} = 1 \text{ V} \), \( V_{DD} = 5 \text{ V} \), \( K_L = 0.25 \text{ mA/\text{V}^2} \), and \( K_D = 1 \text{ mA/\text{V}^2} \).

a. Determine the input voltage at the transition point \( V_{I(tran)} \).

b. Calculate the output voltage and the drain current for \( V_I = 2 \text{ V} \) and for \( V_I = 5 \text{ V} \).

15.8 Design an enhancement load NMOS inverter, as shown in Fig. 15.12(a), to obtain a noise margin of \( N_{ML} \equiv 0.9 \text{ V} \). The threshold voltages are \( V_{IL} = V_{ID} = 1 \text{ V} \), and \( K_L = 40 \text{ \mu A/\text{V}^2} \). The supply voltage is \( V_{DD} = 5 \text{ V} \). Assume \( V_{OH} = V_{IH} = 5 \text{ V} \), load capacitance \( C_L = 2 \text{ pF} \), and clock frequency \( f_{clk} = 1 \text{ MHz} \).

a. Find the design parameter \( K_R \) \( K_D \) \( K_L \) neglecting the body effect.

b. Calculate \( N_{ML} \) if the body effect is included. Assume \( \gamma = 0.5 \sqrt{\text{V}} \) and \( 2\phi_f = 0.6 \text{ V} \).

c. Calculate the low-to-high propagation time \( t_{PLH} \).

d. Calculate the high-to-low propagation time \( t_{PHL} \).

e. Calculate the delay-power product \( (\text{DP}) \).
The parameters of the NMOS inverter in Fig. 15.16(a) with a depletion load are $V_{dd} = -1.5\, \text{V}$, $V_{dd} = 1\, \text{V}$, $V_{dd} = 5\, \text{V}$, $K_D = 1.5\, \text{mA}\, \text{V}^2$, and $K_L = 0.1\, \text{mA}\, \text{V}^2$.

a. Determine the input and output voltages at the first and second transition points.

b. Calculate the output voltage, the drain current, and the power dissipation $P_D$ for $v_1 = 2.5\, \text{V}$ and for $v_1 = 5\, \text{V}$.

The parameters of the NMOS inverter in Fig. 15.16(a) with a depletion load are $V_{dd} = 1\, \text{V}$, $V_{dd} = 1\, \text{V}$, $V_{dd} = 5\, \text{V}$, $K_D = 1.5\, \text{mA}\, \text{V}^2$, and $K_L = 0.1\, \text{mA}\, \text{V}^2$. Determine the ratio $K_D/K_L$ so that (a) $v_O = 0.25\, \text{V}$ at $v_1 = 5\, \text{V}$ and (b) $v_O = 0.25\, \text{V}$ at $v_1 = 4.5\, \text{V}$.

The parameters of the NMOS inverter in Fig. 15.16(a) with a depletion load are $V_{dd} = -1.5\, \text{V}$, $V_{dd} = 1\, \text{V}$, $V_{dd} = 5\, \text{V}$, $K_D = 1.5\, \text{mA}\, \text{V}^2$, and $K_L = 0.1\, \text{mA}\, \text{V}^2$. Determine the value of $V_{il}$ so that $v_O = 0.15\, \text{V}$ at $v_1 = 5\, \text{V}$.

The parameters of the NMOS inverter in Fig. 15.16(a) with a depletion load are $V_{dd} = -2\, \text{V}$, $V_{dd} = 1\, \text{V}$, $V_{dd} = 5\, \text{V}$, $K_D = 1\, \text{mA}\, \text{V}^2$, and $K_L = 0.25\, \text{mA}\, \text{V}^2$.

a. Determine the input and output voltages at the first and second transition points.

b. Calculate the output voltage and the drain currents for $v_1 = 1.5\, \text{V}$ and for $v_1 = 5\, \text{V}$.

d. Calculate the low-to-high propagation time $t_{pHL}$.

e. Calculate the high-to-low propagation time $t_{pHL}$.

f. Calculate the delay-power product (DP).

Design a depletion-load NMOS inverter, as shown in Fig. 15.16(a), to obtain a noise margin of $\text{NMH} \approx 2.5\, \text{V}$. The threshold voltages are $V_{il} = -1.5\, \text{V}$, $V_{dd} = 1\, \text{V}$, and $K_L = 40\, \text{uA}\, \text{V}^2$. The supply voltage is $V_{dd} = 5\, \text{V}$. Assume $V_{OH} = V_{dd} = 5\, \text{V}$, load capacitance $C_L = 2\, \text{pF}$, and frequency $f_{clk} = 1\, \text{MHz}$.

a. Find the design parameter $K_R = K_D K_L$, neglecting the body effect.

b. Calculate $\text{NMH}$ if the body effect is neglected.

c. Calculate $\text{NMH}$ if the body effect is included. Assume $\gamma = 0.5\sqrt{V}$ and $2\phi_F = 0.6\, \text{V}$.

d. Calculate the low-to-high propagation time $t_{pHL}$.

e. Calculate the high-to-low propagation time $t_{pHL}$.

f. Calculate the delay-power product (DP).

The parameters of the NMOS transmission gate in Fig. 15.23 are $V_i = 1.5\, \text{V}$, $K = 0.1\, \text{mA}/\text{V}^2$, $C_L = 5\, \text{pF}$, and $V_G = 5\, \text{V}$.

a. What is the steady-state output voltage if the input voltage is changed in the following sequence: $v_1 = 0$, $v_1 = 5\, \text{V}$, and $v_1 = 3\, \text{V}$?

b. What would the steady-state output voltage be if $v_G$ were switched to 0?

An NMOS RS flip-flop circuit is shown in Fig. P15.15. The input conditions, which are sequential from time 1 to time 4, are also listed in Fig. P15.15. Determine the state of each transistor (on or off) at each time and the logic outputs at $Q_A$ and $Q_B$ for each input condition.

**FIGURE P15.15**
15.7 **CMOS Inverters**

15.16 The parameters of the CMOS inverter in Fig. 15.26(a) are $V_{IN} = 1.5\, V$, $V_{IP} = -1.5\, V$, $V_{DD} = 5\, V$, $K_n = 1\, mA\cdot V^{-2}$, and $K_p = 100\, mA\cdot V^{-2}$.

a. Determine the input and output voltages at the first and second transition points.

b. Calculate the output voltage and the drain currents for $v_1 = 1.5\, V$ and for $v_1 = 4.5\, V$.

15.17 The parameters of the CMOS inverter in Fig. 15.26(a) are $V_{IN} = 1.5\, V$, $V_{IP} = -1.5\, V$, $V_{DD} = 5\, V$, $K_n = 0.2\, mA\cdot V^{-2}$, and $K_p = 100\, mA\cdot V^{-2}$.

a. Determine the input and output voltages at the first and second transition points.

b. Calculate the output voltage and the drain currents for $v_1 = 1.5\, V$ and for $v_1 = 4.5\, V$.

15.18 The parameters of the CMOS inverter in Fig. 15.26(a) are $V_{IN} = 1\, V$, $V_{IP} = -1\, V$, $V_{DD} = 5\, V$, $K_n = 1\, mA\cdot V^{-2}$, and $K_p = 1\, mA\cdot V^{-2}$.

a. Determine the input and output voltages at the first and second transition points.

b. Calculate the output voltage and the drain currents for $v_1 = 1.5\, V$ and for $v_1 = 3.5\, V$.

15.19 The parameters of the CMOS inverter in Fig. 15.26(a) are $V_{IN} = 1.5\, V$, $V_{IP} = -1.5\, V$, and $V_{DD} = 5\, V$. Determine the ratio $K_n / K_p$ so that the transition occurs at an input voltage of $v_1 = V_{I(tran)} = V_{DD} / 2$.

15.20 The parameters of the CMOS inverter in Fig. 15.26(a) are $V_{IN} = 1.5\, V$, $V_{IP} = -1\, V$, and $V_{DD} = 5\, V$. Calculate the output voltages if the input voltage is varied from 0 to 5\, V with a step of 0.25\, V. Assume $K_n / K_p = 1$.

15.21 The parameters of the CMOS inverter in Fig. 15.26(a) are $V_{IN} = 1.5\, V$, $V_{IP} = -1.5\, V$, and $V_{DD} = 5\, V$. Calculate the output voltages if the input voltage is varied from 0 to 5\, V with a step of 0.5\, V. The ratio $K_n / K_p = 1$.

15.22 The parameters of the CMOS inverter in Fig. 15.26(a) are $V_{IN} = 1\, V$, $V_{IP} = -1\, V$, $V_{DD} = 5\, V$, $K_n = 1\, mA\cdot V^{-2}$, and $K_p = 1\, mA\cdot V^{-2}$.

a. Determine the input and output voltages at the first and second transition points.

b. Calculate the output voltage and the drain currents for $v_1 = 1.5\, V$ and for $v_1 = 3.5\, V$.

15.23 Design a CMOS inverter, as shown in Fig. 15.26(a), to operate at a transition voltage $V_M = 2.5\, V$. The threshold voltages are $V_{IP} = -1.5\, V$, $V_{IN} = 1\, V$, and $K_p = 40\, \mu A\cdot V^{-2}$. The supply voltage is $V_{DD} = 5\, V$. Assume $V_{IH} = V_{OH} = 5\, V$, load capacitance $C_L = 2\, pF$, and frequency $f_{clk} = 1\, MHz$.

a. Find the design parameter $K_R = K_n / K_p$, neglecting the body effect.

b. Calculate $N_{ML}$ and $N_{MH}$.

c. Calculate the propagation delay $t_{pd}$.

d. Calculate the delay-power product (DP).

15.8 **CMOS Logic Circuits**

15.24 The parameters of the CMOS transmission gate in Fig. 15.29 are $V_{IN} = 1.5\, V$, $V_{IP} = -1.5\, V$, $V_{DD} = 5\, V$, $K_n = 0.1\, mA\cdot V^{-2}$, $K_p = 0.1\, mA\cdot V^{-2}$, $C_L = 2\, pF$, $\bar{V}_G = 5\, V$, and $v_G = 0$.

a. What is the steady-state output voltage if the input voltage is changed in the following sequence: $v_1 = 0$, $v_1 = 5\, V$, and $v_1 = 3\, V$?

b. What would the steady-state output voltage be if the gate voltages were switched to $\bar{V}_G = 0$ and $v_G = 5\, V$?

15.25 The parameters of the CMOS transmission gate in Fig. 15.29 are $V_{IN} = 1.5\, V$, $V_{IP} = -1.5\, V$, $V_{DD} = 5\, V$, $K_n = 0.1\, mA\cdot V^{-2}$, $K_p = 0.1\, mA\cdot V^{-2}$, $C_L = 2\, pF$, $\bar{V}_G = 5\, V$, and $v_G = 0$. Use PSpice SPICE to plot the transfer characteristic $v_O$ versus $v_1$ if $v_1$ is varied from 0 to 5\, V with a step increment of 0.5\, V. Indicate the regions over which the $n$-channel and the $p$-channel transistors are either turned on or cut off.
15.26 A CMOS RS flip-flop circuit is shown in Fig. P15.26. The input conditions, which are sequential from time 1 to time 4, are also listed. Determine the state of each transistor (on or off) at each time and the logic outputs at $Q_A$ and $Q_B$ for each input condition.

**FIGURE P15.26**

<table>
<thead>
<tr>
<th>Time</th>
<th>$v_A$</th>
<th>$v_B$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

15.27 A CMOS logic circuit is shown in Fig. P15.27. The input conditions, which are sequential from time 1 to time 4, are also listed. Determine the state of each transistor (on or off) at each time and the logic output $v_O$.

**FIGURE P15.27**

<table>
<thead>
<tr>
<th>Time</th>
<th>$v_A$</th>
<th>$v_B$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

15.10 **BJT Inverters**

For Probs. 15.28–15.30, assume that the PSpice SPICE model parameters are

$$\text{CCS}=2\text{PF} \quad \text{TF}=0.1\text{NS} \quad \text{TR}=10\text{NS} \quad VJC=0.85 \quad \text{VAF}=50$$

($\beta_F$ is as specified).

15.28 The bipolar transistor in Fig. 15.33(a) is specified to have $\beta_F$ in the range of 10 to 50. The load resistance is $R_C=10\,\text{k}\Omega$. The DC supply voltage is $V_{CC}=5\,\text{V}$, and the input voltage to the base circuit is $v_I=5\,\text{V}$. If $V_{CE(sat)}=0.1\,\text{V}$ and $V_{BE(sat)}=0.7\,\text{V}$, find (a) the value of $R_B$ that results in saturation with an overdrive factor of 5, (b) the forced $\beta_F$, and (c) the power loss in the transistor $P_D$.

15.29 The bipolar transistor in Fig. 15.33(a) is specified to have $\beta_F$ in the range of 8 to 40. The load resistance is $R_C=1\,\text{k}\Omega$. The DC supply voltage is $V_{CC}=5\,\text{V}$, and the input voltage to the base circuit is $v_I=5\,\text{V}$. If $V_{CE(sat)}=0.2\,\text{V}$ and $V_{BE(sat)}=0.7\,\text{V}$, find (a) the value of $R_B$ that results in saturation with an overdrive factor of 5, (b) the forced $\beta_F$, and (c) the power loss in the transistor $P_D$. 
Design a BJT inverter, as shown in Fig. 15.35, to drive four identical inverters \((N = 4)\) and to give \(V_{OH} = 2.5\,\text{V}\) and \(N_{ML} = 0.2\,\text{V}\). The transistor has \(I_{C(max)} = 4\,\text{mA}\), \(\beta_F = 80\) to 120, \(V_{BE\text{(cut-in)}} = 0.5\,\text{V}\), \(V_{BE\text{(sat)}} = 0.7\,\text{V}\), \(I_d = 2\,\text{ns}\), and \(t_s = 2\,\text{ns}\). The supply voltage \(V_{CC}\) is 5\,\text{V}.

a. Find the values of \(R_C\) and \(R_B\).

b. Calculate \(k_{ODF}\) and \(N_{MH}\).

c. Find the maximum fan-out \(N\) for \(V_{IH} = 2.0\,\text{V}\) and \(k_{ODF} = 1\).

d. Calculate the propagation delay \(t_{pd}\). Assume that the base–emitter junction capacitance of each load transistor is \(C_B = 1.5\,\text{pF}\) (during switching).

e. Calculate the delay-power product (DP) at a frequency \(f_{clk} = 1\,\text{MHz}\).

15.30 Design the gate of the circuit in Fig. P15.31. It has two inputs and feeds one similar gate with one input. Assume \(V_{CC} = 5\,\text{V}\), \(V_D = 0.7\,\text{V}\), \(V_{CE\text{(sat)}} = 0.1\,\text{V}\), \(V_{BE} = 0.7\,\text{V}\), \(\beta_F\text{(forced)} = 10\), and \(I_{C\text{(sat)}} = 1\,\text{mA}\).

b. If the output voltage must be maintained at more than 30\% of \(V_{CC}\), what is the maximum value of fan-out \(N\)? From the transfer function, calculate \(N_{ML}\) and \(N_{MH}\).

For Probs. 15.31–15.42, assume that the PSpice SPICE model parameters are

\[
\begin{align*}
CCS &= 2\,\text{PF} \\
TF &= 0.1\,\text{NS} \\
TR &= 10\,\text{NS} \\
VJC &= 0.85\,\text{V} \\
A &= 50 \\
\end{align*}
\]

\((\beta_F\text{ is as specified}).\)

15.31 Design the gate of the circuit in Fig. P15.33. It has two inputs and feeds three similar NAND gates. Assume \(V_{CC} = 5\,\text{V}\), \(V_D = 0.7\,\text{V}\), \(V_{CE\text{(sat)}} = 0.2\,\text{V}\), \(V_{BE} = 0.7\,\text{V}\), \(\beta_F\text{(forced)} = 10\), and \(I_{C\text{(sat)}} = 1\,\text{mA}\). From the transfer function, calculate \(N_{ML}\) and \(N_{MH}\).
15.34 A TTL gate is shown in Fig. P15.34.

a. Calculate the currents $I_1$, $I_2$, $I_C$, and $I_B$ and the voltages $V_I$ and $V_O$ for input conditions $v_A = v_B = 0$ and $v_A = v_B = 5$ V.

b. Calculate the fan-out for the output low condition. Assume $I_{N0} = 50$ μA and $I_{L0} = 200$ μA.

**FIGURE P15.34**

15.35 Design the TTL NAND gate of the circuit in Fig. 15.38. It has two inputs and feeds two similar NAND gates. The output voltage at output high is $V_{OH} = 3.5$ V, and $I_{N1} = 650$ μA. Assume $V_{CC} = 5$ V, $V_D = 0.7$ V, $V_{CE(sat)} = 0.2$ V, $V_{BE} = 0.7$ V, $β_{F(forced)} = 10$, $β_R = 0.2$, $I_{C3(sat)} = 1$ mA, and $I_{C4(max)} = 2$ mA. From the transfer function, calculate $N_{ML}$ and $N_{MH}$.

15.36 A TTL gate is shown in Fig. P15.36. Calculate the output voltage and all currents of the transistors and fan-out for the conditions $v_A = v_B = 5$ V and $v_A = v_B = 0$. Assume $V_{CC} = 5$ V, $V_{CE(sat)} = 0.2$ V, $V_{BE} = 0.7$ V, $V_{BC1} = 0.7$ V, $β_{F(forced)} = 10$, and $β_R = 0.4$.

**FIGURE P15.36**

15.37 A TTL gate is shown in Fig. P15.37. Calculate the output voltage and all currents of the transistors and fan-out for the conditions $v_A = v_B = 5$ V and $v_A = v_B = 0$. Assume $V_{CC} = 5$ V, $V_{OH} = 3$ V, $V_{CE(sat)} = 0.2$ V, $V_{BE} = 0.7$ V, $V_{BC1} = 0.7$ V, $β_{F(forced)} = 10$, and $β_R = 0.4$. 
15.38 Design the TTL NAND gate of the circuit in Fig. 15.36. It has two inputs and feeds four identical NAND gates. The output voltage at output high is $V_{OH} = 2.8$ V, and $I_{N1} = 72.5$ μA. Assume $V_{CC} = 5$ V, $V_D = 0.7$ V, $V_{CE(sat)} = 0.3$ V, $V_{BE} = 0.8$ V, $\beta_{F(forced)} = 20$, $\beta_R = 0.1$, $I_{C3(sat)} = 2$ mA, and $I_{C4(max)} = 2$ mA. From the VTC, find $N_{ML}$ and $N_{MH}$.

15.39 Design the high-speed TTL NAND gate of the circuit in Fig. 15.43. It has two inputs and feeds four similar NAND gates. The output voltage at output high is $V_{OH} = 3.5$ V, and $I_{N1} = 650$ μA. Assume $V_{CC} = 5$ V, $V_{CE(sat)} = 0.2$ V, $V_{BE} = 0.7$ V, $\beta_{F(forced)} = 10$, $\beta_R = 0.2$, $I_{C3(max)} = 1$ mA, $I_{C4(max)} = 2$ mA, and $I_{BR} = 0.7$ mA. From the VTC, calculate $N_{ML}$ and $N_{MH}$.

15.40 Design the TTL NAND gate of the circuit in Fig. 15.43. It has two inputs and feeds four identical NAND gates. The output voltage at output high is $V_{OH} = 2.8$ V, and $I_{N1} = 65$ μA. Assume $V_{CC} = 5$ V, $V_D = 0.7$ V, $V_{CE(sat)} = 0.3$ V, $V_{BE} = 0.7$ V, $\beta_{F(forced)} = 40$, $\beta_R = 0.1$, $I_{C3(sat)} = 2$ mA, $I_{C4(max)} = 2$ mA, and $I_{BR} = 700$ μA. From the VTC, calculate $N_{ML}$ and $N_{MH}$.

15.41 A Schottky clamped transistor is shown in Fig. P15.41. If the input current is $I_1 = 2$ mA, calculate $I_D$, $I_B$, and $I_C$ for the conditions $I_L = 5$ mA and $I_L = 20$ mA. Assume $\beta_F = 25$, $V_{BE} = 0.7$ V, and $V_D = 0.3$ V.

15.42 A Schottky clamped transistor is shown in Fig. P15.42.

a. If the input current is $I_1 = 2$ mA, calculate $I_D$, $I_B$, and $I_C$ for the condition $I_L = 0$.

b. Determine the maximum value of load current $I_L$ that the transistor can sink and still remain at the edge of saturation. Assume $\beta_F = 25$, $V_{BE} = 0.7$ V, $V_{CE(sat)} = 0.2$ V, and $V_D = 0.3$ V.
15.12 Emitter-Coupled Logic OR/NOR Gates

15.43 a. Design the ECL OR / NOR gate of the circuit in Fig. 15.50. It has two inputs. The desired collector currents are \( I_{C1} = 2 \text{ mA} \) and \( I_{C4} = I_{C5} = 1 \text{ mA} \). At logic 1, \( V_{OR} = -0.7 \text{ V} \), \( V_{NOR} = -1.5 \text{ V} \), and \( V_A \) (or \( V_B \)) = \( V_{OR} = -0.7 \text{ V} \). Assume \( V_{EE} = -5.2 \text{ V} \), \( V_{BE} = 0.7 \text{ V} \), and \( \beta_F = 50 \).

b. Calculate the maximum fan-out with similar ECL gates if \( V_{OR} \) is allowed to fall from \( V_{OR(max)} = 0.7 \text{ V} \) to \( V_{OR(min)} = -0.75 \text{ V} \).

15.44 Repeat Prob. 15.43 if \( V_{CC} = 3 \text{ V} \), \( V_{OR} = -3.2 \text{ V} \), \( V_{NOR} = 1.5 \text{ V} \), and \( V_A \) (or \( V_B \)) = \( V_{OR} = 0.5 \text{ V} \).

15.45 An ECL logic gate with one input is shown in Fig. P15.45. Calculate the resistances \( R_{C1}, R_{C2}, R_E, \) and \( R_4 \). The desired emitter current of all transistors is \( 2 \text{ mA} \). At logic 1, the outputs are \( V_{OR} = 2.5 \text{ V} \) and \( V_{NOR} = 3.5 \text{ V} \). Assume \( V_{CC} = 5 \text{ V}, V_{EE} = 0, V_{BE} = 0.7 \text{ V} \), and \( \beta_F = 200 \). From the transfer function, calculate \( N_{M_L} \) and \( N_{M_H} \).

15.46 a. Design the ECL OR / NOR gate of the circuit in Fig. 15.50. It has two inputs. The desired collector currents are \( I_{C1} = 2 \text{ mA} \) and \( I_{C4} = I_{C5} = 1 \text{ mA} \). At logic 1, \( V_{OR} = -0.7 \text{ V} \), \( V_{NOR} = -1.63 \text{ V} \), and \( V_A \) (or \( V_B \)) = \( V_{OR} = -0.7 \text{ V} \). Assume \( V_{EE} = -5.2 \text{ V} \), \( V_{BE} = 0.7 \text{ V} \), and \( \beta_F = 50 \).

b. Calculate the maximum fan-out with similar ECL gates if \( V_{OR} \) is allowed to fall from \( V_{OR(max)} = -0.7 \text{ V} \) to \( V_{OR(min)} = -0.75 \text{ V} \).

c. From the VTC, calculate \( N_{M_L} \) and \( N_{M_H} \).

15.13 BiCMOS Inverters

15.47 The BiCMOS inverter shown in Fig. 15.54 has \( V_{DD} = 5 \text{ V} \), \( V_{BE} = 0.7 \text{ V} \), \( R_1 = 2 \text{ k}\Omega \), \( R_2 = 6 \text{ k}\Omega \), \( C_1 = 2 \text{ pF} \), and \( \beta_F = 150 \). The threshold voltages are \( V_{TP} = -1.5 \text{ V} \), \( V_{IN} = 1.5 \text{ V} \), and \( K_p = K_n = 40 \mu\text{A} \text{ V}^2 \). Assume \( V_{OL} = 0 \) and \( V_{OH} = 5 \text{ V} \).

a. Calculate \( t_{PLH}, t_{PHL}, \) and \( t_{pd} \).

b. Use PSpice SPICE to plot the VTC and transient response.
CHAPTER 16

INTEGRATED ANALOG CIRCUITS AND APPLICATIONS

Learning Outcomes

After completing this chapter, students should be able to do the following:

- Design and analyze precision rectifiers.
- Design and analyze nonlinear voltage limiters.
- List the differences between comparators and op-amps.
- Apply comparators and op-amps in generating waveforms such as the zero-crossing detector, Schmitt trigger, and square-wave, triangular-wave, and sawtooth-wave generators.
- Describe the internal structure and the principle of operation of commonly used analog-integrated circuits such as the NE/SE-566 VCO, the 555 timer, NE/SE-565 PLL, Teledyne 9400 V/F and F/V converters, LF198 sample-and-hold amplifier, MC1408 and NE/SE-5018 D/A converters, and NE/SE-5034 A/D converter.
- Apply some ICs and design circuits for waveform generation, conversion, detection, or signal processing.

Symbols and Their Meanings

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$, $A_f$</td>
<td>Open-loop and closed-loop voltage gains</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Feedback factor</td>
</tr>
</tbody>
</table>
16.1 Introduction

Electronic circuits are commonly employed in generating waveforms of various shapes for control and interfacing purposes. This chapter examines the basic principles of operation of circuits used to generate waveforms and the applications of some commonly used ICs [1–3].

16.2 Circuits with Op-Amps and Diodes

Many applications—such as peak signal detectors, precision rectifiers, comparator circuits, and limiters—require nonlinear functions. A diode, however, typically has a finite voltage drop of 0.7 V, which distorts the output voltage of a circuit with one or more diodes, especially for low-voltage signals. Op-amp circuits with diodes can reduce the effect of diode drop and are used for precision signal processing.

Let us take the op-amp circuit shown in Fig. 16.1(a) in which the diode provides a unidirectional current flow. If the input voltage $v_S$ is negative, the output voltage $v_{O1}$ of the op-amp becomes negative, causing the diode to be reverse biased; no current flows through the load $R_L$ because the current $i_i$ flowing into the op-amp is zero. Thus, the output voltage is zero: $v_O = 0$ for $v_S \leq 0$. But the voltage $v_{O1}$ will reach the negative saturation limit of the op-amp. If the input voltage $v_S$ is positive, the output voltage $v_{O1}$ of the op-amp becomes positive, causing the diode to be forward biased and supply the load current $i_L$. Since $v_d = 0$ and $i_i = 0$, the output voltage is $v_O = v_S$ for $v_S \geq 0$. The transfer characteristic of $v_O$ versus $v_S$ is shown in Fig. 16.1(b).

For the diode to start conduction, only a very small input voltage is required: $v_{S(min)} = V_D \cdot A_o$, where $V_D$ is the diode drop (typically 0.7 V) and $A_o$ is the open-circuit gain of the op-amp (typically $2 \times 10^5$). This circuit exhibits the characteristic of a diode. However, the effect of the diode drop is negligible. Thus, this circuit is called a superdiode. The drawback of this circuit is that, for negative values of $v_S$, the voltage $v_{O1}$ will swing to the negative saturation limit, thereby slowing the speed of op-amp operation.
16.2.1 Most Positive Signal Detectors

The value of the most positive of a number of input signals can be detected by the circuit shown in Fig. 16.2. The inputs to the unity follower are the input signals to be detected. The diode that has the highest positive signal will conduct, and that signal will appear on the output of the circuit. The current source $I_{DC}$ keeps the diode current constant irrespective of the value of the input signal, and it maintains a constant diode drop. As a result, the voltage drop across the conducting diode does not vary with variations in the input signal. Without the current source, the current and voltage of the conducting diode will change with the level of the most positive voltage, and the output voltage will vary. The unity follower acts as a buffer stage, providing a very high resistance at the diodes and a very low resistance at the output. If each diode is replaced by the superdiode shown in Fig. 16.1, there will be no voltage drop at the diodes; however, this arrangement will increase the complexity of the circuit.

16.2.2 Precision Peak Voltage Detectors

A precision peak voltage detector is used in many applications, such as monitoring the peak temperature of a day (or month) or initiating the action of a device if the signal exceeds a reference value. The circuit of Fig. 16.3 will detect the peak input voltage. The superdiode circuit allows the capacitor to charge to the
peak input signal. The unity follower offers a high resistance to the capacitor and supplies the capacitor voltage to the output without discharging any charge of the capacitor. Resistance $R$, when connected, will allow the capacitor to discharge slowly so that the circuit can adjust to a lower input voltage. Otherwise, the capacitor will maintain its previous higher voltage and will not indicate the correct peak voltage.

### 16.2.3 Precision Half-Wave Rectifiers

A diode requires a minimum voltage, typically 0.7 V, to conduct. In a single-phase half-wave rectifier, one diode conducts. If the input voltage is less than 0.7 V, the output of the rectifier will be zero. Therefore, diode rectifiers are not suitable for rectification of low voltage. An op-amp circuit with two diodes, as shown in Fig. 16.4(a), can rectify a very small voltage in the range of microvolts. The circuit operation can be divided into two intervals: interval 1 and interval 2. We will consider the circuit operation with a sinusoidal input voltage $v_S = V_m \sin \omega t$. 

![Precision half-wave rectifier](image_url)
During interval 1, $0 \leq \omega t \leq \pi$. The input voltage is positive. The voltage $v_{O1}$ at the output of the first op-amp is negative, and diode $D_2$ is off. Diode $D_1$ conducts, and current $i_f$ through $R_F$ equals input current $i_S$. Since the current flowing into the op-amp is zero, $i_S = i_f$ and

$$i_S = i_f = \frac{v_S}{R_1}$$

$$v_d = R_F i_f + v_{O2}$$

For an ideal op-amp, $v_d = 0$, and voltage $v_{O2}$ becomes

$$v_{O2} = -R_F i_f = -R_F i_S = -R_F i_S$$  \text{ (for $R_F = R_1$)}

$$= -v_S$$

Thus, the output voltage at the output of the second inverting op-amp is

$$v_O = -v_{O2} = v_S$$  \text{ (for $v_S \geq 0$, $R_3 = R_2 = R_1$, and $v_S \geq 0$)}

During interval 2, $\pi \leq \omega t \leq 2\pi$. The input voltage is negative. The voltage $v_{O1}$ at the output of the first op-amp is positive, and diode $D_2$ conducts. As a result, the voltage $v_{O1}$ is clamped to approximately the voltage of one diode. Diode $D_1$ remains off, and no current flows through $R_F$. The voltage $v_{O2}$ becomes zero. Thus, the output voltage at the output of the second op-amp is $v_O = -v_{O2} = 0$.

The output voltage $v_O$ is almost independent of the diode characteristics. This is because diode $D_1$ is included in series with the op-amp. Since the op-amp gain is very high, tending to infinity, the effect of the diode and its voltage drop becomes insignificant. The output waveforms are shown in Fig. 16.4(b).

If the directions of the diodes are reversed, the output voltage will correspond to the negative part of the input voltage; there is no need for the second inverting op-amp. This arrangement is shown in Fig. 16.5(a). When the input voltage $v_S$ is positive, the voltage $v_{O1}$ becomes negative, making diode $D_2$ conduct and diode $D_1$ turn off. As a result, the output voltage becomes zero; that is, the output voltage $v_O = 0$ for $v_S \geq 0$. On the other hand, when the input voltage $v_S$ is negative, the voltage $v_{O1}$ becomes positive, making diode $D_2$ turn off and diode $D_1$ conduct. As a result, the output voltage will be equal and opposite to the input voltage; that is, the output voltage $v_O = -v_S$ for $v_S < 0$. The transfer characteristic is shown in Fig. 16.5(b), and the voltage waveforms are shown in Fig. 16.5(c).

![Diagram](https://example.com/diagram.png)

**FIGURE 16.5** Alternate precision half-wave rectifier
16.2.4 Precision Full-Wave Rectifiers

The half-wave rectifier of Fig. 16.5(a) can be modified to operate as a precision full-wave rectifier if we use the following algebraic relationship:

\[ v_O = 2v_S - v_S = v_S \quad \text{(for the positive interval of the input voltage)} \]

This situation is shown in Fig. 16.6(a). Let us consider the case with \( R_1 = R_2 = R_F = R \) and \( R_3 = R_4 = 2R \). We will divide the circuit operation into two intervals, interval 1 and interval 2, and use a sinusoidal input voltage \( v_S = V_m \sin \omega t \).

During interval 1, \( 0 \leq \omega t \leq \pi \). \( v_S \) is positive, and \( v_{O2} = -v_S \). The voltage at the output of the second op-amp can be found from

\[ v_O = -\left(\frac{R_3}{R_2} v_{O2} + \frac{R_3}{R_4} v_S\right) \quad (16.1) \]

which, for \( R_3 = R_4 = 2R \) and \( R_2 = R \), becomes

\[ v_O = -2v_{O2} - v_S = -2(-v_S) - v_S = v_S \quad \text{(for } v_S \geq 0) \]

During interval 2, \( \pi \leq \omega t \leq 2\pi \). \( v_S \) is negative, and \( v_{O2} = 0 \). The voltage at the output of the second op-amp can be found from

\[ v_O = -\left(\frac{R_3}{R_2} v_{O2} + \frac{R_3}{R_4} v_S\right) \quad (16.2) \]

which, for \( R_3 = R_4 = 2R \) and \( R_2 = R \), becomes

\[ v_O = -2v_{O2} - v_S = -2 \times 0 - v_S = -v_S \quad \text{(for } v_S < 0) \]

Thus, the output voltage is the inverted version of the input voltage, as shown in Fig. 16.6(b).
16.2.5 Precision Clamping Circuits

A precision clamping circuit is sometimes used in signal processing to add just enough DC voltage to the input voltage so that the sum never crosses the zero level. Figure 16.7(a) shows a superdiode circuit with an input voltage \( v_S \) and a DC reference voltage \( V_{\text{ref}} \). As long as the op-amp voltage \( v_d \) is positive, \( v_{O1} \) will be positive, and diode \( D_1 \) will conduct, thereby making the (−) terminal behave as a virtual ground. To consider the circuit operation, let us take a sinusoidal input voltage \( v_S = V_m \sin \omega t \) and \( V_{\text{ref}} = 0 \).

During the interval \( 0 \leq \omega t \leq \pi \), \( v_S \) is positive, \( v_d \) is negative, \( v_{O1} \) is negative, and diode \( D_1 \) is reverse biased. No current flows through the capacitor, and it cannot charge. During the interval \( \pi < \omega t \leq 3\pi \), \( v_S \) is negative, \( v_d \) is positive, \( v_{O1} \) is positive, and diode \( D_1 \) conducts. The (−) terminal behaves as a virtual ground. A charging current flows through the capacitor, with point A being at a higher potential than point B. The capacitor is charged to the peak negative voltage \( V_m \). For the interval \( \omega t > 3\pi \),

\[
v_d = -(V_m + v_S) = -V_m(1 + \sin \omega t)
\]

and diode \( D_1 \) remains off; that is, the output voltage \( v_O \) is

\[
v_O = -v_d = V_m(1 + \sin \omega t)
\]
The voltage waveforms are shown in Fig. 16.7(b). With a reference voltage of $V_{\text{ref}}$, the capacitor will charge to $(V_m + V_{\text{ref}})$, and the output voltage $v_O$ can be expressed by

$$v_O = V_m \sin \omega t + V_m + V_{\text{ref}}$$  \hspace{1cm} (16.3)

For example, for $V_m = 10 \text{ V}$ and $V_{\text{ref}} = 5 \text{ V}$, $v_O = 10 \sin \omega t + 15 \text{ V}$, and for $V_m = 10 \text{ V}$ and $V_{\text{ref}} = -5 \text{ V}$, $v_O = 10 \sin \omega t + 5 \text{ V}$. If the direction of diode $D_1$ is reversed, the capacitor will charge when $v_d$ becomes negative during the interval $0 \leq \omega t \leq \pi / 2$. The output voltage will then be reversed; that is,

$$v_O = -(V_m \sin \omega t + V_m + V_{\text{ref}})$$

A resistor R, represented in Fig. 16.7(a) by a light line, can be connected across the capacitor so that the capacitor can discharge slowly and the circuit can adjust to an input voltage of lower amplitude. The resistor can also provide a path for the DC biasing current of the op-amp. As shown in Fig. 16.7(c), voltage followers may be connected to the input and output sides so that the clamping circuit draws no current from the signal source and can deliver load current without affecting the charge on the capacitor.

### 16.2.6 Fixed-Voltage Limiters

A limiter restricts the output voltage to a specified value. A negative output voltage can be limited approximately to zero by connecting a diode across the feedback resistor $R_F$ of the inverting amplifier in Fig. 3.11, as shown in Fig. 16.8(a). If the input voltage $v_S$ is positive, the output voltage $v_O$ tends to be negative, and diode $D_1$ conducts, limiting the output to a negative value $-V_D$ of the diode voltage drop. If the input voltage is negative, the output voltage becomes positive, and the diode is reverse biased. The output voltage follows the input voltage with a change in polarity. The transfer characteristic is shown in Fig. 16.8(b). If the direction of the diode is reversed, as shown in Fig. 16.9(a), the positive output voltage is limited to $V_D$, as shown by the transfer characteristic in Fig. 16.9(b).

### 16.2.7 Adjustable Voltage Limiters

Output voltage can be limited to an adjustable level by choosing resistors with appropriate values. A circuit that limits the negative output voltage is shown in Fig. 16.10(a). If the output is positive, diode $D_1$ is reverse biased, and the circuit operates as an inverting amplifier. If the output voltage is negative, its transfer

![FIGURE 16.8 Negative voltage limiter](image)
FIGURE 16.9 Positive voltage limiter

FIGURE 16.10 Adjustable negative voltage limiter
characteristic changes its slope at $V_{O(min)}$ and then varies with a slope of $-R_3/R_1$. The circuit operation can be divided into two intervals: interval 1 and interval 2.

During interval 1, diode $D_1$ is reverse biased and remains off. The circuit operates as an inverting amplifier, and the slope of the transfer characteristic is $-R_F/R_1$.

During interval 2, diode $D_1$ conducts, and the limiting circuit is active. Let $V_D$ be the forward diode drop. $D_1$ is turned on when the potential $V_x$ becomes $-V_D$. Voltage $V_x$ can be found from Thevenin’s equivalent circuit, as shown in Fig. 16.10(b). By the superposition theorem (considering sources $v_O$ and $V_A$ separately), Thevenin’s equivalent voltage is given by

$$V_{Th} = \frac{R_2v_O}{R_2 + R_3} + \frac{R_3V_A}{R_2 + R_3}$$

Diode $D_1$ turns on when $V_x = -V_D$; that is,

$$V_x = -V_D = V_{Th} = \frac{R_2v_O}{R_2 + R_3} + \frac{R_3V_A}{R_2 + R_3}$$

which gives the negative clamping output voltage $V_{O(min)}$ as

$$V_{O(min)} = v_O = -V_D - (V_A + V_D)\frac{R_3}{R_2} = -V_D\left(1 + \frac{R_3}{R_2}\right) - V_A\frac{R_3}{R_2}$$

from which the positive input voltage corresponding to $V_{O(min)}$ can be found:

$$V_{S(max)} = \frac{R_1}{R_F}V_{O(min)} = \frac{R_1}{R_F}\left[\frac{R_3}{R_2}V_A + \left(1 + \frac{R_3}{R_2}\right)V_D\right]$$

($V_{Th} + V_D$) acts a reference voltage $V_{ref}$. Then $v_S$ can be compared to $V_{ref}$. $V_{S(max)}$ is the threshold voltage at which the change of slope takes place. The equivalent circuit in Fig. 16.10(c) has two input signals, $v_S$ and $(V_{Th} + V_D)$, and it can be characterized as a summing amplifier. The output voltage during the clamped condition can be expressed as

$$v_O = -\frac{R_Fv_S}{R_1} - \frac{R_F}{(R_2 || R_3)}\left(\frac{R_2v_O}{R_2 + R_3} + \frac{R_3V_A}{R_2 + R_3} + V_D\right)$$

which, after solving for $v_O$, gives

$$v_O = -\frac{1}{1 + R_F/R_3}\left[\frac{R_F}{R_1}v_S + \frac{R_F}{R_2}V_A + \frac{R_F}{(R_2 || R_3)}V_D\right]$$

For $R_F R_3 >> 1$, which is normally the case, Eq. (16.7) is reduced to

$$v_O = -\frac{R_3}{R_1}v_S - \frac{R_3}{R_2}V_A - \left(1 + \frac{R_3}{R_2}\right)V_D \quad \text{(for } v_S > V_{S(max)})$$

which describes the transfer characteristic of the limiter, as shown in Fig. 16.10(d). The slope beyond the break point is $-R_3/R_1$, and this slope should be made small by choosing $R_1 >> R_3$. Note that $|V_{O(min)}|$ must be less than the saturation voltage $|V_{sat}|$ of the op-amp.

The positive voltage can be limited by adding another diode $D_2$, as shown in Fig. 16.11(a). $V_x$ and $V_y$ limit the negative voltage and positive voltage, respectively. $V_y$ can be found from Eq. (16.4):

$$V_y = V_D = V_{Th1} = \frac{R_4v_O}{R_4 + R_5} - \frac{R_5V_B}{R_4 + R_5}$$

(16.9)
Similarly, the positive clamping voltage can be found from Eq. (16.5):

\[
V_{O(\text{max})} = v_O = V_D + (V_B + V_D) \frac{R_5}{R_4} = V_D \left(1 + \frac{R_5}{R_4}\right) + V_B \frac{R_2}{R_4} \tag{16.10}
\]

The negative input voltage corresponding to \(V_{O(\text{max})}\) can be found from

\[
V_{S(\text{min})} = -\frac{R_1}{R_F} V_{O(\text{max})} = -\frac{R_1}{R_F} \left[\frac{R_5}{R_4} V_B + \left(1 + \frac{R_5}{R_4}\right) V_D\right] \tag{16.11}
\]

If \(V_{\text{Th1}}\) is Thevenin’s equivalent voltage due to \(V_B\), \((V_{\text{Th1}} - V_D)\) acts as reference voltage \(V_{\text{ref1}}\). Then \(v_S\) can be compared to \(V_{\text{ref1}}\). \(V_{S(\text{min})}\) is the threshold voltage at which the change of slope takes place. Using Eq. (16.7), we can find the output voltage during the positive voltage clamping to be

\[
v_O = -\frac{R_5}{R_1} v_S + \frac{R_5}{R_4} V_B + \left(1 + \frac{R_5}{R_4}\right) V_D \quad (\text{for } v_S < V_{S(\text{min})}) \tag{16.12}
\]

which describes the transfer characteristic of an adjustable positive and negative voltage limiter, as shown in Fig. 16.11(b). This is a practical limiter and is commonly used. It is also called a soft limiter since the output voltage will increase slightly if the input voltage is increased beyond the break points. All DC supply voltages of the limiter are generally made the same magnitude; that is, \(V_A = V_B = V_{CC} = V_{EE}\).
EXAMPLE 16.1

Designing a negative voltage–limiting circuit  Design a negative voltage limiter as shown in Fig. 16.10(a) for $V_A = 12$ V. The circuit should limit the negative output voltage to $V_{O(min)} = -5$ V. The voltage gain without limiting is $A_f = -4$. The diode should be fully turned on at a forward current of $i_D = 0.1$ mA, and its forward voltage drop is $V_D = 0.7$ V. The slope after the break point is to be limited to $-1/20$. Determine the values of $R_1$, $R_2$, $R_3$, and $R_F$.

SOLUTION

Equation (16.6) gives the diode current at clamping as

$$i_D = \frac{V_{S(max)}}{R_1} = -\frac{1}{R_F} V_{O(min)}$$

so

$$0.1 \times 10^{-3} = -\frac{5}{R_F}$$

or $R_F = 50$ kΩ

Since $A_f = -4 = -R_F/R_1$,

$$R_1 = \frac{R_F}{4} = \frac{50}{4} = 12.5$$ kΩ

Since slope $S = -R_3/R_1 = -1/20$,

$$R_3 = \frac{R_1}{20} = \frac{12.5}{20} = 625$$ Ω

From Eq. (16.5),

$$-5 = -0.7 - (12 + 0.7) \frac{R_3}{R_2}$$

$$\frac{R_3}{R_2} = 0.3386$$

$$R_2 = \frac{R_3}{0.3386} = 625 \div 0.3386 = 1846$$ Ω

EXAMPLE 16.2

Finding the limiting voltages of an op-amp limiting circuit  The adjustable limiter in Fig. 16.11(a) has $R_1 = 15$ kΩ, $R_F = 60$ kΩ, $R_2 = 4$ kΩ, $R_3 = 1$ kΩ, $R_4 = 5$ kΩ, $R_5 = 1$ kΩ, $V_A = 15$ V, $V_B = -15$ V, and $V_D = 0.7$ V. Determine (a) the positive clamping voltage $V_{O(max)}$ and the corresponding input voltage $V_{S(max)}$, (b) the negative clamping voltage $V_{O(min)}$ and the corresponding input voltage $V_{S(min)}$, and (c) the output voltage when the input voltage is $V_S = 5$ V.
Integrated Analog Circuits and Applications

SOLUTION

\[ R_1 = 15 \, \text{k}\Omega, \ R_F = 60 \, \text{k}\Omega, \ R_2 = 4 \, \text{k}\Omega, \ R_3 = 1 \, \text{k}\Omega, \ R_4 = 5 \, \text{k}\Omega, \ R_5 = 1 \, \text{k}\Omega, \ V_A = 15 \, \text{V}, \ -V_B = -15 \, \text{V}, \ V_D = 0.7 \, \text{V}, \ \text{and} \ v_S = 5 \, \text{V}. \]

(a) From Eq. (16.10),
\[ V_{O\text{max}} = 0.7 + (15 + 0.7) \times \frac{1 \, \text{k}}{5 \, \text{k}} = 3.84 \, \text{V} \]
From Eq. (16.11),
\[ V_{S\text{min}} = -V_{O\text{max}} \frac{R_1}{R_F} = -3.84 \times \frac{15 \, \text{k}}{60 \, \text{k}} = -0.96 \, \text{V} \]

(b) From Eq. (16.5),
\[ V_{O\text{min}} = -0.7 - (15 + 0.7) \times \frac{1 \, \text{k}}{4 \, \text{k}} = -4.625 \, \text{V} \]
From Eq. (16.6),
\[ V_{S\text{max}} = -V_{O\text{min}} \frac{R_1}{R_F} = 4.625 \times \frac{15 \, \text{k}}{60 \, \text{k}} = 1.15625 \, \text{V} \]

(c) From Eq. (16.8),
\[ v_O = -5 \times \frac{1 \, \text{k}}{15 \, \text{k}} - 15 \times \frac{1 \, \text{k}}{4 \, \text{k}} - \left(1 + \frac{1 \, \text{k}}{4 \, \text{k}}\right) \times 0.7 = -4.958 \, \text{V} \]

EXAMPLE 16.3

Designing an adjustable voltage-limiting circuit
(a) Design an adjustable limiter as shown in Fig. 16.11(a) to satisfy the following specifications: \( V_A = 12 \, \text{V}, \ -V_B = -12 \, \text{V}, \ V_{O\text{min}} = -5 \, \text{V}, \ V_{O\text{max}} = 6 \, \text{V}, \) voltage gain \( A_f = -4, \) slope after break with a positive input (for \( v_S > 0 \)) \( S_1 = -1 \, 20, \) and slope after break with a negative input (for \( v_S < 0 \)) \( S_2 = -1 \, 25. \) Assume that diodes are fully on at a diode current of \( i_D = 0.1 \, \text{mA} \) and the corresponding on-state diode voltage is \( V_D = 0.7 \, \text{V}. \)

(b) Use PSpice/SPICE [4] to plot the transfer characteristic \( v_O \) versus \( v_S = V(6). \) Assume \( V_{CC} = 12 \, \text{V} \) and \( -V_{EE} = -12 \, \text{V}. \)

SOLUTION

(a) The steps in completing the design are as follows:

Step 1. The diode current can be related to \( V_{O\text{min}} \) and \( R_F \) by
\[ i_D = -\frac{V_{O\text{min}}}{R_F} \text{ or } 0.1 \, \text{mA} = -\frac{5}{R_F} \]
which gives \( R_F = 50 \, \text{k}\Omega. \)

Step 2. The voltage gain \( A_f \) is given by
\[ A_f = -\frac{R_F}{R_1} \text{ or } -4 = -\frac{50 \, \text{k}\Omega}{R_1} \]
which gives \( R_1 = 12.5 \, \text{k}\Omega. \)
Step 3. The slope after break with a positive input (for $v_S > 0$) is

$$S_1 = \frac{-R_3}{R_1} \text{ or } -\frac{1}{20} = -\frac{R_3}{12.5 \text{ k}\Omega}$$

which gives $R_3 = 625 \Omega$.

Step 4. The slope after break with a negative input (for $v_S < 0$) is

$$S_2 = \frac{-R_5}{R_1} \text{ or } -\frac{1}{25} = -\frac{R_5}{12.5 \text{ k}\Omega}$$

which gives $R_5 = 500 \Omega$.

Step 5. The value of $R_2$ can be found from Eq. (16.5):

$$V_{O(\text{min})} = -V_D - (V_A + V_D) \frac{R_3}{R_2} \text{ or } -5 = -0.7 - (12 + 0.7) \frac{625}{R_2}$$

which gives $R_2 = 1846 \Omega$.

Step 6. The value of $R_4$ can be found from Eq. (16.10):

$$V_{O(\text{max})} = V_D + (V_B + V_D) \frac{R_5}{R_4} \text{ or } 6 = 0.7 + (12 + 0.7) \frac{500}{R_4}$$

which gives $R_4 = 1198 \Omega$.

(b) The limiter circuit for PSpice simulation is shown in Fig. 16.12. The transfer characteristic (for voltage-controlled voltage source model) is shown in Fig. 16.13, which gives $V_{S(\text{max})} = 1.41 \text{ V}$, $V_{S(\text{min})} = -1.72 \text{ V}$, $V_{O(\text{max})} = 6.7 \text{ V}$ (expected value is 6 V), and $V_{O(\text{min})} = -5.5 \text{ V}$ (expected value is -5 V). The discrepancies between the design values and the PSpice results may be attributed to the use of an ideal op-amp versus a macromodel and the use of a specific diode versus a diode of type D1N4148. If we run the simulation using voltage-controlled voltage source model, the PSpice and calculated values will agree more closely.
16.2.8 Zener Voltage Limiters

The output can be limited by adding two zener diodes across the output terminals of a noninverting or an inverting amplifier. An arrangement for a noninverting amplifier circuit is shown in Fig. 16.14(a). Zener diodes limit the output voltage between $V_{Z1} + V_D$ and $V_{Z2} + V_D$, where $V_D$ is the voltage drop of a zener diode in the forward direction. Without zener action, the output voltage can be found by using Eq. (3.18) for the noninverting amplifier:

$$v_O = \left(1 + \frac{R_F}{R_1}\right)v_S$$

If the output voltage is to be limited to a maximum voltage $V_{O(max)} = v_O = (V_{Z1} + V_D)$ and a minimum voltage $V_{O(min)} = v_O = -(V_{Z2} + V_D)$, it can be expressed as

$$v_O = \begin{cases} V_{Z1} + V_D & \text{for } v_S > V_{S(max)} \\ -(V_{Z2} + V_D) & \text{for } v_S < V_{S(min)} \\ \left(1 + \frac{R_F}{R_1}\right)v_S & \text{for } V_{S(min)} \leq v_S \leq V_{S(min)} \end{cases}$$

The transfer characteristic is shown in Fig. 16.14(b).

**EXAMPLE 16.4**

**Designing a zener voltage-clamping circuit** Design an output voltage-clamping circuit as shown in Fig. 16.14(a) so that the normal slope of the transfer characteristic is $S = v_O / v_S = 10$, $V_{O(max)} = 5.7$ V, and $V_{O(min)} = -7.7$ V. Determine the zener voltages $V_{Z1}$ and $V_{Z2}$. Assume $V_D = 0.7$ V.

**SOLUTION**

$$V_{O(max)} = 5.7$$ V, and $V_{O(min)} = -7.7$ V. Since $S = 1 + \frac{R_F}{R_1} = 10$,

$$\frac{R_F}{R_1} = 10 - 1 = 9$$
16.2.9 Hard Limiters

The limiter in Fig. 16.11(a) can be made to switch between \( V_{O\text{(min)}} \) and \( V_{O\text{(max)}} \) if the input voltage \( v_S \) becomes greater than zero or less than zero, respectively. This can be accomplished by making the feedback resistance \( R_F \) in Fig. 16.11(a) very large, tending to infinity. That is, for \( R_F \to \infty \), the gain of the circuit becomes very large, tending to infinity, and the output voltage becomes \( V_{O\text{(min)}} \) if \( v_S > 0 \) and \( V_{O\text{(max)}} \) if \( v_S < 0 \). The circuit shown in Fig. 16.15(a) compares the input signal \( v_S \) with the reference signal \( V_{\text{ref}} = 0 \), and if the input signal is greater than or less than zero, the output changes its value. This circuit is known as a hard limiter, although it exhibits finite slopes beyond the breaks.

The equations given earlier for adjustable voltage limiters can be applied to the limiter in Fig. 16.15(a), except that \( R_F = \infty \) and the gain \( A_F = -R_F \) \( R_1 \) is very large, tending to infinity. The transfer characteristic is shown in Fig. 16.15(b). The voltage limiting can also be accomplished by connecting two zener diodes, as shown in Fig. 16.16(a), for which \( V_{O\text{(max)}} = (V_{Z1} + V_D) \) and \( V_{O\text{(min)}} = -(V_{Z2} + V_D) \). Practical circuits exhibit finite slopes beyond the breaks because of the finite resistances of the zener diodes. These finite slopes are represented in Fig. 16.16(b) by light lines.

If \( R_1 = 5 \, \text{k\Omega} \),
\[
R_F = 9 \times 5 \, \text{k} = 45 \, \text{k\Omega}
\]
Since \( V_{O\text{(max)}} = (V_{Z1} + V_D) \),
\[
V_{Z1} = V_{O\text{(max)}} - V_D = 5.7 - 0.7 = 5 \, \text{V}
\]
Since \( V_{O\text{(min)}} = -(V_{Z2} + V_D) \),
\[
V_{Z2} = -(V_{O\text{(min)}} + V_D) = -(7.7 + 0.7) = 7 \, \text{V}
\]

\[
V_{Z1} = 5 \text{V}
\]

\[
V_{Z2} = 7 \text{V}
\]

\[
A_F = \frac{R_F}{R_1} = 45 \text{V} / 5 \text{V} = 9 \text{V}
\]

\[
V_{CC} - V_{EE} + V_{A} - V_{B} = \frac{R_3}{R_1} V_{O\text{(max)}} - \frac{R_3}{R_1} V_{O\text{(min)}}
\]

\[
\text{Slope} = -\frac{R_3}{R_1}
\]

\[
\text{Slope} = -\frac{R_5}{R_1}
\]

\[
\text{Slope} = -\frac{R_4}{R_1}
\]

FIGURE 16.15 Hard limiter
EXAMPLE 16.5

Designing a hard voltage limiter

(a) Design a hard limiter like the one in Fig. 16.15(a). The circuit should limit the negative output voltage to \( V_{O_{\text{min}}} = -5 \text{ V} \) and the positive output voltage to \( V_{O_{\text{max}}} = 5 \text{ V} \). The magnitude of the slopes after the break points should be less than or equal to \( \frac{1}{20} \). The diode drop is \( V_D = 0.7 \text{ V} \). The DC supplies are given by \( V_A = V_B = 12 \text{ V} \). Determine the values of \( R_1, R_2, R_3, R_4, \) and \( R_5 \).

(b) Use PSpice/SPICE to plot the transfer characteristic. Assume \( V_{CC} = 12 \text{ V}, \ V_{EE} = -12 \text{ V}, \) and \( v_S = -6 \text{ V} \) to \( 6 \text{ V} \).

SOLUTION

(a) Choose \( R_1 = 10 \text{ k}\Omega \). Since the slope \( |S_1| = R_3 \ R_1 = 1 \ 20 \),

\[
R_3 = \frac{R_1}{20} = \frac{10 \text{ k}\Omega}{20} = 500 \text{ \Omega}
\]

From Eq. (16.5),

\[
5 = -0.7 - \frac{(12 + 0.7)R_3}{R_2}
\]

\[
\frac{R_3}{R_2} = 0.3386
\]

\[
R_2 = 1477 \text{ \Omega}
\]

Since the slope \( |S_2| = R_5 \ R_1 = 1 \ 20 \),

\[
R_5 = \frac{R_1}{20} = \frac{10 \text{ k}\Omega}{20} = 500 \text{ \Omega}
\]
From Eq. (16.10),
\[ 5 = 0.7 + \frac{(12 + 0.7)R_5}{R_4} \]
\[ R_5 = 0.3386 \]
\[ R_4 = 1477 \, \Omega \]

(b) The hard limiter for PSpice simulation is shown in Fig. 16.17. The transfer characteristic is shown in Fig. 16.18, which gives \( V_{S_{\text{max}}} = 55.86 \, \text{mV} \), \( V_{S_{\text{min}}} = -27.93 \, \text{mV} \), \( V_{O_{\text{max}}} = 4.50 \, \text{V} \), and \( V_{O_{\text{min}}} = -4.55 \, \text{V} \). Discrepancies between the design values and the PSpice results come from use of an ideal op-amp versus a macromodel and use of a specific diode versus a diode of type D1N4148.

**FIGURE 16.17** Comparator circuit for PSpice simulation

**FIGURE 16.18** PSpice transfer characteristic for Example 16.5
A comparator compares a signal voltage $v_S$ on one input terminal with a known voltage, called the reference voltage $V_{\text{ref}}$. The symbol of a comparator, which is similar to that of an op-amp, is shown in Fig. 16.19(a). A comparator gives a digital output voltage $v_O$. Thus, it can be considered a simple one-bit analog-to-digital (A/D) converter, which produces a digital 1 output ($v_O = V_H$) if the input voltage $v_S$ is above the reference level $V_{\text{ref}}$ and a digital 0 output ($v_O = V_L$) if the input voltage $v_S$ falls below the reference level $V_{\text{ref}}$. The output levels $V_L$ and $V_H$ may be of opposite polarity (i.e., $V_H$ positive and $V_L$ negative or vice versa), or both $V_L$ and $V_H$ may be either positive or negative. The transfer characteristic of an ideal comparator is shown in Fig. 16.19(b). The output may be symmetric or asymmetric.

A practical comparator has a finite voltage gain in the range from 3000 to 200,000 and takes a finite amount of time (in the range from 10 ns to 1 $\mu$s) to make a transition from one level to another (e.g., $V_L$ to $V_H$). The transfer characteristic of a practical comparator is shown in Fig. 16.19(c). The input voltage swing required to produce the output voltage transition is in the range of about 0.1 mV to 4 mV. The output of a comparator must switch rapidly between the levels. The bandwidth must be wide because the wider the bandwidth, the faster the switching speed will be. Some typical parameters (listed here for the LM111 comparator) are as follows:

- Operates from a single 5-V power supply
- Input current: 150 nA (maximum)
- Offset current: 20 nA (maximum)
- Differential input voltage: $\pm$30 V
- Voltage gain: 200 V/mV (typical)
### 16.3.1 Comparators versus Op-Amps

A comparator is designed to operate under open-loop conditions, usually as a switching device, whereas an op-amp is generally operated under closed-loop conditions as a linear amplifier. Otherwise, comparators are very similar to op-amps. Like an op-amp, a comparator has an offset voltage (typically 4 mV), a biasing current (typically 150 nA), and an offset current (typically 20 nA). The characteristics of comparators and op-amps are listed in Table 16.1.

### 16.3.2 Output-Side Connection

Comparators are often used as an interface between digital and analog signals. The power supply at the analog side ($V_{CC}$ and $V_{EE}$, typically ±15 V) is different from that at the digital side ($V_{CC}$, typically 0 to 5 V). Comparators generally have an open-collector output stage, which allows separate power supplies for the analog and digital parts. A block diagram of the LM111 comparator is shown in Fig. 16.20. The output terminals are the collector and the emitter of an npn transistor. If the input voltage to the transistor is low, the transistor is off, and the output logic level is 1; that is, the output is high (5 V), and the current flows through the pull-up resistor $R_P$ to the digital load. On the other hand, if the input voltage to the transistor is high, the transistor is driven into saturation, and the output logic level is 0; that is, the output is low at the saturation voltage of the transistor (typically 0.2 V), and no current flows through the pull-up resistor $R_P$ to the digital load.

### 16.3.3 Threshold Comparators

The voltage at which a comparator changes from one level to another is called the crossover (or threshold) voltage. Its value can be adjusted by adding resistors, as shown in the noninverting comparator.
in Fig. 16.21(a). From the superposition theorem, the voltage $V_+$ at the noninverting terminal is given by

$$V_+ = \frac{R_1}{R_1 + R_F} V_{\text{ref}} + \frac{R_F}{R_1 + R_F} v_S$$

(16.13)

Ideally, the crossover will occur when $V_+ = 0$; that is,

$$R_1 V_{\text{ref}} + R_F v_S = 0$$

which gives the low threshold voltage of the comparator $V_{L_t} = v_S$ (for changing from low to high) as

$$V_{L_t} = -\frac{R_1}{R_F} V_{\text{ref}}$$

(16.14)

Thus, the output voltage becomes high ($V_H$) at the positive saturation voltage ($+V_{\text{sat}}$) when $V_+ > 0$ (i.e., $v_S > V_{L_t}$). The transfer characteristic is shown in Fig. 16.21(b).

If the input signal $v_S$ is connected to the inverting terminal, as shown in Fig. 16.22(a), the output will change from high ($V_H$) to low ($V_L$). This situation is shown in Fig. 16.22(b). The high threshold voltage of the comparator $V_{H_t} = v_S$ (for changing from high to low) is given by

$$V_{H_t} = \frac{R_1}{R_1 + R_F} V_{\text{ref}}$$

(16.15)

FIGURE 16.21 Noninverting threshold comparator

![Noninverting threshold comparator](image)

FIGURE 16.22 Inverting configuration for noninverting threshold comparator

![Inverting configuration](image)
Thus, the output voltage becomes low ($V_L$) at the negative saturation voltage ($-V_{sat}$) when $v_S > V_+$ (i.e., $v_S > V_{HH}$). The transfer characteristic is shown in Fig. 16.22(b).

If both the input signal $v_S$ and the reference signal $V_{ref}$ are connected to the inverting terminal, as shown in Fig. 16.23(a), the output will be the inversion of the output in Fig. 16.21(b). That is, the output will change from high ($V_H$) to low ($V_L$) when the input is $v_S = V_{Lt}$. This situation is shown in Fig. 16.23(b).

In both inverting and noninverting configurations, the output voltage is limited to the saturation voltage of the comparator. The output voltage can, however, be set to specified limits by external limiters such as zener diodes connected across the output terminals of Figs. 16.21(a), 16.22(a), and 16.23(a). This approach is illustrated in Fig. 16.23(c), in which resistance $R$ is connected to limit the current through the zener diodes.

**FIGURE 16.23** Inverting threshold comparator

KEY POINTS OF SECTION 16.3

- A comparator compares a signal voltage on one input terminal with a known voltage, called the reference voltage, on the other input terminal. It is designed to operate under open-loop conditions, usually as a switching device. Comparators are often used as an interface between digital and analog signals.
- A comparator can be used as a threshold comparator in either inverting or noninverting mode. The voltage at which the comparator changes from one level to another is called the crossover (or threshold) voltage.

### 16.4 Zero-Crossing Detectors

A comparator can be used as a zero-crossing detector, as shown in Fig. 16.24(a). Input signal $v_S$ is compared with a reference signal of 0. When $v_S$ passes through zero in the positive direction, the output $v_O$ is driven into negative saturation ($-V_{sat}$) as a result of the very high gain of the comparator. Conversely, when $v_S$ passes through zero in the negative direction, the output is driven into positive saturation ($+V_{sat}$). The input and output waveforms are shown in Fig. 16.24(b).
Diodes D₁ and D₂ in Fig. 16.24(a) protect the comparator from damage due to excessive input voltage \(v_S\). Because of these diodes, the differential input voltage \(v_d\) of the comparator is clamped to approximately 0.7 V or \(-0.7\) V. These diodes, called clamp diodes, are external to the comparator. It is up to the designer to determine if the diodes are needed to protect the circuit. Resistance \(R\) is connected in series with input signal \(v_S\) to limit the current through D₁ and D₂. Resistance \(R_x\) is used to reduce the effect of comparator offset problems.

If \(v_S\) is such that it crosses zero very slowly, then \(v_O\) may not switch quickly from one saturation voltage to the other. Instead, \(v_O\) may fluctuate between two saturation voltages \(+V_{sat}\) and \(-V_{sat}\) as a result of input offset voltage or noise signals at the comparator input terminals. Therefore, a zero-crossing detector will not be suitable for a low-frequency signal or a signal with noise superimposed on it.

**KEY POINT OF SECTION 16.4**

- A zero-crossing detector is a special application of a comparator in which the input signal is compared with a reference signal of 0.

### 16.5 Schmitt Triggers

A Schmitt trigger compares a regular or irregular waveform with a reference signal and converts the waveform to a square or pulse wave. A Schmitt trigger is often known as a squaring circuit. It is also known as a bistable multivibrator because it has two stable states, low and high. It can remain in one state indefinitely; it moves to the other stable state only when a triggering signal is applied. Schmitt triggers can be classified into two types depending on the type of op-amp configuration used: inverting or noninverting.
16.5.1 Inverting Schmitt Trigger

In an inverting Schmitt trigger, the input signal is applied to the inverting terminal of the comparator. The inverting threshold comparator in Fig. 16.22(a) can operate as an inverting Schmitt trigger if the resistance \( R_F \) is connected to the output side. This arrangement is shown in Fig. 16.25(a). The voltage divider consisting of \( R_1 \) and \( R_F \) will feed a fraction \( \beta = R_1 (R_1 + R_F) \) of the output voltage back to the positive terminal of the comparator. If \( A \) is the open-loop gain of the comparator, the closed-loop voltage gain \( A_f \) is given by

\[
A_f = \frac{v_O}{v_S} = \frac{-A}{1 - \beta A}
\]  

(16.16)

If \( \beta A > 1 \), which is usually the case, the feedback signal \( V_+ = \beta v_O = \beta A v_S \) will be greater than its original value. For any change in \( v_S \), the output \( v_O \) will continue to build up toward the saturation limit.
Transfer Characteristics

To start with, let us assume that $v_S$ is negative and the output is at the positive saturation voltage, $V_H = +V_{sat}$. If $v_S$ is increased from 0, there will be no change in the output until $v_S$ reaches a value of $v_S = V_+ = \beta V_{sat}$. If $v_S$ begins to exceed $V_{Ht} = \beta V_{sat}$, the differential voltage $v_d$, which will be negative, will be amplified by the voltage gain $A$ of the comparator; that is, $v_O$ will be negative, thereby making $V_+$ negative also. The result will be an increase in the magnitude of the differential voltage $v_d$, and $v_O$ will become more negative. This regenerative process will continue until eventually the comparator saturates, with its output voltage equal to the negative saturation voltage, $v_O = V_L = -V_{sat}$, and $V_+ = -\beta V_{sat}$. Increasing $v_S$ beyond $v_S = \beta V_{sat}$ will have no effect on the state of the output voltage. The transfer characteristic for increasing $v_S$ is shown in Fig. 16.25(b).

If $v_S$ is decreased further while the output is low, there will be no change in the output until $v_S$ goes negative, with a value of $v_S = V_+ = -\beta V_{sat}$. If $v_S$ begins to exceed $V_{Lt} = -\beta V_{sat}$, the differential voltage $v_d$, which will be positive, will be amplified by the gain of the comparator; that is, $V_+$ will be positive. The result will be an increase in the differential voltage $v_d$, and $v_O$ will become more positive. This regenerative process will continue until eventually the comparator saturates, with its output voltage equal to the positive saturation voltage, $v_O = V_H = +V_{sat}$, and $V_+ = \beta V_{sat}$. Decreasing $v_S$ further ($v_S < -\beta V_{sat}$) will have no effect on the state of the output voltage. The transfer characteristic for decreasing $v_S$ is shown in Fig. 16.25(c).

The complete transfer characteristics are shown in Fig. 16.25(d). A Schmitt trigger exhibits a hysteresis, or deadband, condition. That is, when the input of the Schmitt trigger exceeds $V_{Ht} = +V_{th}$, its output switches from $+V_{sat}$ to $-V_{sat}$, and when the input goes below $V_{Lt} = -V_{th}$, the output reverts to its original state, $+V_{sat}$.

Every time input voltage $v_S$ exceeds certain levels, called the positive (or upper) threshold voltage $+V_{th}$ and the negative (or lower) threshold voltage $-V_{th}$, it changes the state of output voltage $v_O$. If the input signal is a sine wave, the output will be a square wave, as shown in Fig. 16.25(e). $+V_{th}$ and $-V_{th}$ are given by

$$+V_{th} = V_{Ht} = \frac{R_1}{R_1 + R_F}(+V_{sat}) \quad (16.17)$$

$$-V_{th} = V_{Lt} = \frac{R_1}{R_1 + R_F}(-V_{sat}) \quad (16.18)$$

where $V_{sat} = |+V_{sat}| = |-V_{sat}|$ and $V_{th} = |+V_{th}| = |-V_{th}|$.

Effect of Positive Feedback

$R_F$ provides positive feedback. As soon as the output voltage begins to change, positive feedback increases the differential voltage $v_d$, which, in turn, further changes the output voltage. Once a transition is initiated by a change in the input signal $v_S$, the positive feedback forces the comparator to complete the transition from one state to another rapidly and to operate in saturation, either positive or negative. Positive feedback leads to rapid transition of the output. Oscillations, which normally occur in the active region and hence prevail for a short time, are avoided.
EXAMPLE 16.6

Designing a Schmitt trigger with a hysteresis band

(a) Design a Schmitt trigger as in Fig. 16.25(a) so that \( V_{th} = |+V_{th}| = |-V_{th}| = 5 \) V. Assume \( V_{sat} = |+V_{sat}| = 14 \) V.

(b) Use PSpice/SPICE [4] to plot the hysteresis characteristic for \( v_s = 10 \sin (800 \pi t) \).

SOLUTION

(a) The steps required to design the Schmitt trigger are as follows:

Step 1. Find the values of \( R_1 \) and \( R_F \). From Eq. (16.17),

\[
1 + \frac{R_F}{R_1} = \frac{V_{sat}}{V_{th}}
\]

so \( R_F = \frac{14}{5} = 2.8 \). Let \( R_1 = 10 \) kΩ; then

\( R_F = 1.8 \times R_1 = 18 \) kΩ (use a 20-kΩ potentiometer)

Step 2. Choose the value of offset minimizing resistance \( R_x \):

\[ R_x = R_1 \parallel R_F = 10 \) kΩ \parallel 18 \) kΩ = 6.43 kΩ

(b) The circuit for PSpice simulation is shown in Fig. 16.26. The comparator is simulated by the PSpice macro-model of the LM111. To obtain the negative output voltage swing, terminal 1 is connected to the negative power supply instead of to the ground.

The transfer characteristic and the output voltage \( v_O = V(U1:OUT) \) (using EX16-6.SCH) are shown in Fig. 16.27[(a) and (b)], respectively. The simulated values are \( V_{th} = 5.02 \) V (expected value is 5 V), \( V_{Lt} = -5.28 \) V (expected value is -5 V), \( V_{Ht} = 14.05 \) V (expected value is 14 V), and \( V_L = -14.83 \) V (expected value is -14 V).
16.5.2 Noninverting Schmitt Trigger

In a noninverting Schmitt trigger, the input signal is applied to the noninverting terminal of the comparator, and the transfer characteristics are inverted. The noninverting threshold comparator in Fig. 16.21(a) can be operated as a noninverting Schmitt trigger if the resistance \( R_F \) is connected to the output side. This arrangement is shown in Fig. 16.28(a). Resistance \( R_F \) will feed a current signal, a fraction \( \beta = 1 \) \( R_F \) of the output voltage, back to the positive terminal of the comparator, providing positive shunt-shunt feedback. As soon
as the output voltage begins to change, the positive shunt-shunt feedback increases the feedback current $i_f$, which in turn increases the differential voltage $v_d$ and hence further changes the output voltage. Once a transition is initiated by a change in the input signal $v_s$, positive feedback forces the comparator to complete the transition from one state to another rapidly and to operate in saturation, either positive or negative.

Transfer Characteristics

To start with, let us assume that $v_s$ is negative and the output is at the negative saturation voltage, $V_L = -V_{sat}$. If $v_s$ is increased from a relatively large negative value, there will be no change in the output until $v_s$ reaches a value of $v_s = V_{Lt} = -V_{sat} \frac{R_1}{R_F}$. If $v_S$ begins to exceed $V_{Lt}$, the differential voltage $v_d$, which will be positive, will be amplified by the voltage gain $A$ of the comparator. That is, $v_O$ will be positive, thereby making $v_s$ positive also. The result will be an increase in the magnitude of the differential voltage $v_d$, and $v_O$ will become more positive. This regenerative process will continue until eventually the comparator saturates, with its output voltage equal to the positive saturation voltage, $V_H = +V_{sat}$. Increasing $v_s$ further ($v_s \geq V_{Lt}$) will have no effect on the state of the output voltage.

If $v_s$ is decreased while the output is high, there will be no change in the output until $v_s$ decreases to a value of $v_s = V_{Ht} = +V_{sat} \frac{R_1}{R_F}$. If $v_s$ begins to decrease beyond $V_{Ht}$, the differential voltage $v_d$ will be negative and will be amplified by the gain of the comparator. As a result, $v_O$ will become more negative. This regenerative process will continue until eventually the comparator saturates, with its output voltage equal to the negative saturation voltage $-V_{sat}$. Decreasing $v_s$ further ($v_s \leq V_{Ht}$) will have no effect on the state of the output voltage. The complete transfer characteristics are shown in Fig. 16.28(b).

16.5.3 Schmitt Trigger with Reference Voltage

The switching voltage of a Schmitt trigger circuit is defined as the average of $V_{Lt}$ and $V_{Ht}$. For the circuits in Figs. 16.25(a) and 16.28(a), $V_{Lt} = -V_{Ht}$, and hence the switching voltage $V_{st}$, which is the width of the hysteresis band, is zero; that is, $V_{st} = (V_{Lt} + V_{Ht}) / 2 = 0$. However, some applications require shifting the crossover voltage in either the positive or the negative direction along the $v_s$-axis. This can be accomplished by adding a reference voltage $V_{ref}$ to the circuit in Fig. 16.25(a), as shown in Fig. 16.29(a) for a noninverting Schmitt trigger.
noninverting Schmitt trigger. The complete transfer characteristics are shown in Fig. 16.29(b). Assuming that $V_{Lt}$ and $V_{Ht}$ are symmetric about the zero-axis, the switching voltage is given by

$$V_{st} = \frac{R_F}{R_1 + R_F} V_{ref} \tag{16.20}$$

Thus, the upper and lower crossover voltages become

$$V_{Ht} = V_{st} + \frac{R_1}{R_1 + R_F} (+V_{sat}) \tag{16.21}$$

$$V_{Lt} = V_{st} + \frac{R_1}{R_1 + R_F} (-V_{sat}) \tag{16.22}$$

The direction of the hysteresis loop in Fig. 16.29(b) can be reversed by applying a reference voltage $V_{ref}$ to the circuit in Fig. 16.28(a). This arrangement is shown in Fig. 16.30(a), and the corresponding transfer characteristics are shown in Fig. 16.30(b).

### 16.5.4 Effects of Hysteresis on the Output Voltage

To understand the effect of the *hysteresis*, or *deadband*, condition, consider a sinusoidal signal with a noise signal superimposed on it, as shown in Fig. 16.31(a). If there is no hysteresis, the output voltage will change to its saturation limit when the input signal $v_S$ crosses zero, as shown in Fig. 16.31(b). However, if the output is made to change when the input signal exceeds specified voltage limits $V_{Ht}$ and $V_{Lt}$, there will be less switching of the output voltage, as shown in Fig. 16.31(c). As a result, any unwanted signal (e.g., noise) will not make the output change. Also, a deadband can be used to reduce the number of contact-bounces in a system such as a temperature-control system, in which the heating element turns on or off when the temperature falls below or rises above the set value.
EXAMPLE 16.7

Designing a Schmitt trigger with a shifted hysteresis band

(a) Design a Schmitt trigger as in Fig. 16.29(a) so that \( V_{Ht} = 7 \) V and \( V_{Lt} = 3 \) V. Assume \( V_{sat} = | -V_{sat} | = 14 \) V and an input frequency of \( f = 400 \) Hz. Determine the values of \( R_1, R_F, \) and \( V_{ref}. \)

(b) Use PSpice/SPICE to plot the hysteresis characteristic for \( v_S = 10 \sin (800 \pi t) \). Use the macromodel of the \( \mu A741 \) op-amp.

SOLUTION

(a) The Schmitt trigger can be designed using the following steps:

**Step 1.** Find the values of \( R_1 \) and \( R_F. \) From Eqs. (16.21) and (16.22), we can find the input width of the hysteresis band (HB) as follows:

\[
\text{HB} = V_{Ht} - V_{Lt} = \frac{2R_1}{R_1 + R_F} V_{sat}
\]

which gives

\[
1 + \frac{R_F}{R_1} = \frac{2V_{sat}}{V_{Ht} - V_{Lt}} = \frac{2 \times 14}{7 - 3} = 7
\]

Let \( R_1 = 10 \) kΩ; then \( R_F = (7 - 1) \times R_1 = 60 \) kΩ.

**Step 2.** Determine the value of reference voltage \( V_{ref}. \)

\[
V_{st} = \frac{R_F}{R_1 + R_F} V_{ref} = \frac{V_{Ht} + V_{Lt}}{2} = \frac{7 + 3}{2} = 5 \text{ V}
\]

which gives \( V_{ref} = 5.83 \) V.

(b) We will use op-amp \( \mu A741 \) rather than comparator LM111 to illustrate the advantage of a comparator in a Schmitt trigger. The circuit for PSpice simulation is shown in Fig. 16.32.
The transfer characteristic and the output voltage (using EX16-7.SCH) are shown in Fig. 16.33[(a) and (b)], which gives $V_H = 7.03\ \text{V}$ (expected value is 7 V), $V_L = 3.03\ \text{V}$ (expected value is 3 V), and $V_{sat} = 14.6\ \text{V}$ (expected value is 14 V). The transition from low to high and vice versa is not very sharp, as a result of the low slew rate of the op-amp (compared to that of a comparator, shown in Fig. 16.27).

The Schmitt trigger circuit for PSpice simulation is shown in FIGURE 16.32. The transfer characteristic and the output voltage are shown in FIGURE 16.33.

**FIGURE 16.32**  Schmitt trigger circuit for PSpice simulation

**FIGURE 16.33**  Transfer characteristic and input and output voltages for Example 16.7

**KEY POINTS OF SECTION 16.5**

- A Schmitt trigger compares a regular or irregular waveform with a reference signal and converts the waveform to a square or pulse wave. A Schmitt trigger is often known as a *squaring circuit*. It is also known as a *bistable multivibrator* because it has two stable states, low and high. Schmitt triggers can be classified into two types depending on the type of op-amp configuration used: inverting or noninverting.
- A Schmitt trigger exhibits a *hysteresis*, or *deadband*, condition; that is, when the input of the Schmitt trigger exceeds $+V_{th}$, its output switches from $+V_{sat}$ to $-V_{sat}$, and when the input goes below $-V_{th}$, the output reverts to its original state, $+V_{sat}$.
- When a general-purpose op-amp (e.g., μA741) is used to generate the characteristic of a hysteresis loop, the transition from low to high and vice versa is not as sharp as it is with a comparator.
16.6 Square-Wave Generators

A square wave can be generated if the output of an op-amp is forced to swing repetitively between positive saturation $+V_{\text{sat}}$ and negative saturation $-V_{\text{sat}}$. This can be accomplished by connecting a bistable multivibrator (or Schmitt trigger) with an $RC$ circuit in the feedback loop. The circuit implementation is shown in Fig. 16.34(a). This square-wave generator is also called a free-running or astable multivibrator because the output does not have any stable state. The output of the op-amp will be in either positive saturation or negative saturation, depending on whether the differential input voltage $v_d$ is positive or negative.

Assuming that the voltage across capacitor $C$ is zero, the voltage at the inverting terminal is zero initially; that is, $v_- = 0$ at the instant the DC supply voltages $V_{\text{CC}}$ and $V_{\text{EE}}$ are turned on. However, at the same instant, the voltage $v_+$ at the noninverting terminal will have a very small value that will depend on the output offset voltage $V_{\text{OO}}$; that is,

$$v_d = v_+ - v_- = v_+ = V_{\text{OO}}$$

which has a positive value. But $v_d$ will be amplified as a result of the very large gain of the op-amp (typically $2 \times 10^5$) and will drive the output of the op-amp to positive saturation $+V_{\text{sat}}$. Capacitor $C$ will start charging toward $+V_{\text{sat}}$ through $R$. However, as soon as the voltage across $C$, which is equal to $v_-$, is slightly more than $v_+$, then $v_d = v_+ - v_-$ will become negative, and the output of the op-amp will switch to negative saturation $-V_{\text{sat}}$. The operation of the circuit can be divided into two modes: mode 1 for $v_d > 0$ and mode 2 for $v_d < 0$.

During mode 1, $v_d > 0$ and the output voltage of the op-amp is at positive saturation $+V_{\text{sat}}$. The voltage $v_+$ becomes

$$v_+ = \frac{R_1}{R_1 + R_F} (+V_{\text{sat}})$$

(16.23)

Capacitor $C$ will again start charging toward $+V_{\text{sat}}$ through $R$. As soon as the voltage across $C$ is slightly more than $v_+$, then $v_d = v_+ - v_-$ will become negative, and the output of the op-amp will be forced to switch to negative saturation $-V_{\text{sat}}$. 

![Square-wave generator circuit](image)
During mode 2, \( v_d < 0 \) and the op-amp’s output voltage is at negative saturation \(-V_{\text{sat}}\). The voltage \( v_+ \) follows the voltage divider rule and can be found from

\[
v_+ = \frac{R_1}{R_1 + R_F} (-V_{\text{sat}})
\]  

(16.24)

As long as \( v_d \) is negative, the output will remain in negative saturation. The capacitor will discharge and then recharge toward \(-V_{\text{sat}}\) through \( R \). When the voltage across \( C \) is slightly more negative than \( v_+ \), then \( v_d = v_+ - v_- \) will become positive, and the output of the op-amp will be forced to switch to positive saturation \(+V_{\text{sat}}\). Then mode 1 begins again, and the cycle is repeated.

Voltage \( v_+ \) acts as the reference. The capacitor voltage \( v_C \) tries to follow \( v_+ \). However, as soon as the magnitude of \( v_- \) becomes slightly greater than that of \( v_+ \), \( v_+ \) switches its polarity. As a result, the output swings from positive to negative and vice versa. The waveforms of the output voltage and capacitor voltage are shown in Fig. 16.34(b).

Assuming that \(+V_{\text{sat}}\) is the output voltage and the capacitor has an initial voltage of \(-V_{\text{th}}\) during mode 1, the equivalent circuit during the charging period is as shown in Fig. 16.34(c). Using KVL, we can write

\[
V_{\text{sat}} = R_i C + \frac{1}{C} \int i_C dt - V_{\text{th}}
\]

which gives the charging current \( i_C(t) \) as

\[
i_C(t) = \frac{V_{\text{sat}} + V_{\text{th}}}{R} e^{-t/RC}
\]  

(16.25)

where

\[
V_{\text{th}} = \frac{R_1}{R_1 + R_F} V_{\text{sat}}
\]  

(16.26)

The capacitor voltage \( v_C(t) \) can be found from

\[
v_C(t) = V_{\text{sat}} - (V_{\text{sat}} + V_{\text{th}}) e^{-t/RC}
\]  

(16.27)

At \( t = t_1 \), the capacitor is recharged to \( V_{\text{th}} \). That is, \( v_C(t = t_1) = V_{\text{th}} \). From Eq. (16.27), we get

\[
V_{\text{th}} = V_{\text{sat}} - (V_{\text{sat}} + V_{\text{th}}) e^{-t_1/RC}
\]  

(16.28)

which gives

\[
t_1 = -RC \ln \frac{V_{\text{sat}} - V_{\text{th}}}{V_{\text{sat}} + V_{\text{th}}} = -RC \ln \frac{V_{\text{sat}} - R_1 V_{\text{sat}}/(R_1 + R_F)}{V_{\text{sat}} + R_1 V_{\text{sat}}/(R_1 + R_F)}
\]

\[
= -RC \ln \frac{R_F}{2R_1 + R_F} = RC \ln \frac{2R_1 + R_F}{R_F}
\]  

(16.29)

The period \( T \) of the output voltage is given by

\[
T = t_1 + t_2 = 2t_1 = 2RC \ln \frac{2R_1 + R_F}{R_F} = 2RC \ln \left( 1 + \frac{2R_1}{R_F} \right)
\]  

(16.30)

Thus, the frequency of the output voltage \( f_o \) is given by

\[
f_o = \frac{1}{T} = \frac{1}{2RC \ln (1 + 2R_1/R_F)}
\]  

(16.31)
Equation (16.31) shows that the output frequency depends not only on the time constant $\tau = RC$ but also on the relationship between $R_1$ and $R_F$. If $R_F = 1.164 R_1$, Eq. (16.31) is reduced to

$$f_o = \frac{1}{2RC}$$ \hfill (16.32)

**NOTES**

1. The inputs of the op-amp are subjected to large differential voltages. To prevent excessive differential current flow to the op-amp, a series resistance $R_S$, typically on the order of $100 \, \text{k}\Omega$, can be connected to each of the inverting and noninverting terminals of the op-amp.
2. The peak-to-peak output voltage can be reduced by connecting a pair of zener diodes back to back at the output terminal.

**EXAMPLE 16.8**

**Designing a square-wave generator**

(a) Design the square-wave generator shown in Fig. 16.34(a) so that $f_o = 5 \, \text{kHz}$. Assume $+V_{sat} = | -V_{sat} | = 14 \, \text{V}$.

(b) Use PSpice/SPICE to check your design. Use the macromodel of the LM111 for sharper transition.

**SOLUTION**

(a) The steps used to design the square-wave generator are as follows:

**Step 1.** Choose the value of $R_1$: Let $R_1 = 10 \, \text{k}\Omega$.

**Step 2.** To simplify the design, choose $R_F = 1.164 R_1$ and find $R_F$:

$$R_F = 1.164 R_1 = 1.164 \times 10 \, \text{k}\Omega = 11.64 \, \text{k}\Omega \quad \text{(use a 20-k}\Omega \text{ potentiometer)}$$

**Step 3.** Choose a value of $C$: Let $C = 0.01 \, \mu\text{F}$.

**Step 4.** Find the value of $R$ from Eq. (16.32):

$$R = \frac{1}{2 C f_o} = \frac{1}{2 \times 0.01 \, \mu\text{F} \times 5 \, \text{kHz}} = 10 \, \text{k}\Omega$$

(b) The circuit for PSpice simulation is shown in Fig. 16.35.
16.7 Triangular-Wave Generators

A triangular-wave generator can be produced by integrating the square-wave output of a Schmitt trigger circuit. This can be accomplished by cascading an integrator with a bistable multivibrator (or Schmitt trigger), as shown in Fig. 16.37(a), which consists of a comparator \(A_1\) and an integrator \(A_2\). Comparator \(A_1\) continuously compares the noninverting input \(v_+\) at point \(P\) with the inverting input \(v_-\) (i.e., 0 V). Thus, the differential voltage is \(v_d = v_+ - v_- = v_+\). Because of the very large gain of the op-amp, the output of \(A_1\) will be at negative saturation \(-V_{\text{sat}}\) or positive saturation \(+V_{\text{sat}}\) when \(v_+\) goes slightly below or above 0, respectively.

To examine the principle of operation of the triangular-wave generator, let us assume that when the supply voltages \(V_{CC}\) and \(-V_{EE}\) are switched on, the voltage at the noninverting terminal begins slightly above 0 as a result of the input offset voltage of the op-amp. Because of the high gain of the op-amp, the output of \(A_1\) will be switched to positive saturation \(+V_{\text{sat}}\). Therefore, the output of the op-amp is forced to swing repetitively between positive saturation \(+V_{\text{sat}}\) and negative saturation \(-V_{\text{sat}}\). The output voltages of \(A_1\) and \(A_2\) are shown in Fig. 16.37(b). The operation of the circuit can be divided into two modes.

During mode 1, \(v_+ > 0\) and the output of \(A_1\) is at positive saturation \(+V_{\text{sat}}\), which is the input to the inverting integrator \(A_2\). The output of \(A_2\) will be a negative-going ramp. Thus, one side of \(R_F\) will be at...
+ $V_{sat}$, and one side of $R_1$ will be at the negative-going ramp of $A_2$. When the negative-going ramp exceeds a certain value $-V_{th}$, the voltage at point P will be slightly below 0, and the output of $A_1$ will switch from positive saturation $+V_{sat}$ to negative saturation $-V_{sat}$.

The equivalent circuit for determining the condition under which the circuit switches over to negative saturation is shown in Fig. 16.37(c). Since the current flowing into the op-amp is negligible, $I_{R1} = -I_{RF}$; that is,

$$\frac{-V_{th}}{R_1} = \frac{-V_{sat}}{R_F}$$

which gives the condition for $v_+ < 0$ as

$$V_{th} = \frac{R_1}{R_F} (V_{sat})$$ (16.33)

During mode 2, $v_+ < 0$ and the output of $A_1$ is at negative saturation $-V_{sat}$, which is the input to the integrator $A_2$. The output of $A_2$ will be a positive-going ramp. When the positive-going ramp exceeds a certain value $V_{th}$, the voltage at point P will be slightly above 0, and the output of $A_1$ will switch from negative saturation $-V_{sat}$ to positive saturation $+V_{sat}$. When this occurs, mode 1 starts again, and the cycle is repeated.

The equivalent circuit for determining the condition under which the circuit switches over to positive saturation is shown in Fig. 16.37(d). Neglecting the current flowing into the op-amp, we have $I_{R1} = -I_{RF}$; that is,

$$V_{th} = \frac{V_{sat}}{R_F}$$
which gives the condition for $v_+ > 0$ as

$$V_{th} = \frac{R_1}{R_F} ( +V_{sat} )$$  \hspace{1cm} (16.34)

where $V_{sat} = | +V_{sat} | = | -V_{sat} |$. The peak-to-peak output amplitude of the triangular wave is given by

$$v_{pp} = V_{th} - ( -V_{th} ) = \frac{2R_1}{R_F} V_{sat}$$  \hspace{1cm} (16.35)

The period and the frequency of the output voltage can be determined from the time that is required to charge the capacitor from $-V_{th}$ to $V_{th}$ or from $V_{th}$ to $-V_{th}$. Let us consider the capacitor voltage during mode 2, when the input voltage to the integrator is $-V_{sat}$; that is,

$$v_C(t) = -\frac{1}{RC} \int v_O dt - V_{th} = -\frac{1}{RC} \int ( -V_{sat} ) dt - V_{th}$$

$$\quad = \frac{V_{sat}}{RC} t - V_{th}$$  \hspace{1cm} (16.36)

At half period, $t = t_1 = T/2$ and $v_C(t = T/2) = V_{th}$, and Eq. (16.36) yields

$$V_{th} = \frac{V_{sat}}{RC \left( \frac{T}{2} \right)} - V_{th}$$

which gives the period as

$$T = \frac{4RCV_{th}}{V_{sat}}$$  \hspace{1cm} (16.37)

Substituting the value of $V_{th}$ from Eq. (16.34), we have for the period of the triangular wave

$$T = \frac{4RC}{V_{sat}} \times \frac{R_1 V_{sat}}{R_F} = \frac{4RCR_1}{R_F}$$  \hspace{1cm} (16.38)

which gives the frequency of oscillation as

$$f_o = \frac{R_F}{4RCR_1}$$  \hspace{1cm} (16.39)

**EXAMPLE 16.9**

**Designing a triangular-wave generator**

(a) Design the triangular-wave generator shown in Fig. 16.37(a) so that $f_o = 4$ kHz and $V_{th} = | -V_{th} | = 5$ V. Assume $+V_{sat} = |-V_{sat}| = 14$ V.

(b) Use PSpice/SPICE to check your design.

**SOLUTION**

(a) The steps used to design the triangular-wave generator are as follows:

**Step 1.** Find the values of $R_1$ and $R_F$. Equation (16.34) gives

$$\frac{R_1}{R_F} = \frac{V_{th}}{V_{sat}} = \frac{5}{14} = 0.36$$
Let $R_1 = 10 \, \text{k}\Omega$; then

$$R_F = \frac{R_1}{0.36} = \frac{10 \, \text{k}\Omega}{0.36} = 28 \, \text{k}\Omega \quad \text{(use a 30-k}\Omega\text{ potentiometer)}$$

**Step 2.** Choose a suitable value of $C$: Let $C = 0.01 \, \mu\text{F}$.

**Step 3.** Find the value of $R$. Equation (16.39) gives

$$R = \frac{R_F}{4f_0CR_1}$$

$$= \frac{28 \, \text{k}\Omega}{4 \times 4 \, \text{kHz} \times 0.01 \, \mu\text{F} \times 10 \, \text{k}\Omega} = 17.5 \, \text{k}\Omega \quad \text{(use a 20-k}\Omega\text{ potentiometer)}$$

**b)** The circuit for PSpice simulation is shown in Fig. 16.38. The op-amp is simulated by PSpice macromodel LM111.

The PSpice plots of the voltages at the output of amplifiers $A_1$ and $A_2$ (using EX16-9.SCH) are shown in Fig. 16.39. The results are $V_{\text{sat}} = 14.4 \, \text{V}$ (expected value is 14 \, \text{V}), $-V_{\text{sat}} = -14.8 \, \text{V}$ (expected value is $-14 \, \text{V}$), $V_{\text{th}} = 5.3 \, \text{V}$ (expected value is 5 \, \text{V}), $-V_{\text{th}} = -5.2 \, \text{V}$ (expected value is $-5 \, \text{V}$), and $T = 312.1 - 62.6 = 249.5 \, \mu\text{s}$ (expected value is 250 \, \mu\text{s}). If the integrator time constant $\tau = RC$ is made much smaller than the period of the output waveform, the triangular wave can be made very close to a sine wave.

**FIGURE 16.38** Triangular-wave generator for PSpice simulation

**FIGURE 16.39** Output waveforms for Example 16.9
16.8 Sawtooth-Wave Generators

In a triangular waveform, the rise time is always equal to the fall time. That is, the same amount of time is required for a triangular wave to swing from \(-V_{\text{th}}\) to \(+V_{\text{th}}\) as from \(+V_{\text{th}}\) to \(-V_{\text{th}}\). On the other hand, a sawtooth waveform has unequal rise and fall times. The rise time may be faster than the fall time or vice versa. The triangular-wave generator in Fig. 16.37(a) may be converted to a sawtooth generator by adding a variable DC voltage \(V_{\text{ref}}\) to the noninverting terminal of the op-amp. The addition of \(V_{\text{ref}}\), which acts as a reference signal for the integrator \(A_2\), can be accomplished by using a potentiometer, as shown in Fig. 16.40(a). The voltage waveforms are shown in Fig. 16.40(b). As with the triangular-wave generator, the operation of the circuit can be divided into two modes.

During mode 1, \(v_+ > 0\) and the output of \(A_1\) is at positive saturation \(+V_{\text{sat}}\), which is the input signal to the integrator \(A_2\). The equivalent circuit for the operation of the integrator is shown in Fig. 16.40(c). At the beginning of this mode, the output voltage is \(V_{\text{th}}\), and the voltage at the inverting terminal is \(v_1 \approx V_{\text{ref}}\). Thus, the initial voltage on the capacitor is

\[
v_C(t = 0) = v_1 - v_0 = V_{\text{ref}} - V_{\text{th}}
\]

![Diagram of a sawtooth-wave generator](image)

**FIGURE 16.40** Sawtooth-wave generator
The instantaneous voltage across the capacitor $v_C(t)$ is given by

$$V_{\text{ref}} - v_O(t) = \frac{1}{C} \int i_C \, dt + v_C(t = 0) = \frac{1}{C} \int \frac{V_{\text{sat}} - V_{\text{ref}}}{R} \, dt + V_{\text{ref}} - V_{\text{th}}$$

which gives the instantaneous output voltage as

$$v_O(t) = V_{\text{th}} - \frac{V_{\text{sat}} - V_{\text{ref}}}{RC} \, t$$ \hspace{1cm} (16.41)

At the end of this mode (at $t = t_1$), the output voltage has changed to $-V_{\text{th}}$, whose value can be found from Eq. (16.33). That is, $v_O(t = t_1) = -V_{\text{th}}$. From Eq. (16.41),

$$-V_{\text{th}} = V_{\text{th}} - \frac{V_{\text{sat}} - V_{\text{ref}}}{RC} \, t_1$$

which gives the duration of mode 1 as

$$t_1 = \frac{2RCV_{\text{th}}}{V_{\text{sat}} - V_{\text{ref}}}$$ \hspace{1cm} (16.42)

During mode 2, $v_+ < 0$ and the output of $A_1$ is at negative saturation $-V_{\text{sat}}$, which is the input to the integrator $A_2$. The equivalent circuit for the operation of the integrator is shown in Fig. 16.40(d). At the beginning of this mode, the output voltage is $-V_{\text{th}}$, and the initial voltage on the capacitor is $v_C(t = 0) = v_1 - v_O = V_{\text{ref}} + V_{\text{th}}$. If we redefine the time origin $t = 0$ as the beginning of this mode, the instantaneous voltage across the capacitor is given by

$$V_{\text{ref}} - v_O(t) = \frac{1}{C} \int i_C \, dt + v_C(t = 0) = \frac{1}{C} \int \frac{-V_{\text{sat}} - V_{\text{ref}}}{R} \, dt + V_{\text{ref}} + V_{\text{th}}$$

which gives the instantaneous output voltage as

$$v_O(t) = -V_{\text{th}} + \frac{V_{\text{sat}} + V_{\text{ref}}}{RC} \, t$$ \hspace{1cm} (16.43)

At the end of this mode (at $t = t_2$), the output voltage has changed to $V_{\text{th}}$, whose value can be found from Eq. (16.34); that is, $v_O(t = t_2) = V_{\text{th}}$. From Eq. (16.43),

$$V_{\text{th}} = -V_{\text{th}} + \frac{V_{\text{sat}} + V_{\text{ref}}}{RC} \, t_2$$

which gives the duration for mode 2 as

$$t_2 = \frac{2RCV_{\text{th}}}{V_{\text{sat}} + V_{\text{ref}}}$$ \hspace{1cm} (16.44)

The period of the sawtooth wave can be found from Eqs. (16.42) and (16.44):

$$T = t_1 + t_2 = \frac{2RCV_{\text{th}}}{V_{\text{sat}} - V_{\text{ref}}} + \frac{2RCV_{\text{th}}}{V_{\text{sat}} + V_{\text{ref}}} = \frac{4RCV_{\text{th}}V_{\text{sat}}}{V_{\text{sat}}^2 - V_{\text{ref}}^2}$$ \hspace{1cm} (16.45)

which gives the frequency of oscillation as

$$f_o = \frac{V_{\text{sat}}^2 - V_{\text{ref}}^2}{4RCV_{\text{th}}V_{\text{sat}}}$$ \hspace{1cm} (16.46)
Duty cycle $k$, which is defined as the ratio of $t_1$ to $T$, can be determined from Eqs. (16.42) and (16.45) as follows:

$$
k = \frac{t_1}{T} = \frac{2RCV_{th}}{V_{sat} - V_{ref}} \times \frac{V_{sat}^2 - V_{ref}^2}{4RCV_{th}V_{sat}} = \frac{V_{sat} + V_{ref}}{2V_{sat}}
$$

$$
= \frac{1}{2} \left[ 1 + \frac{V_{ref}}{V_{sat}} \right]
$$

NOTE  This circuit allows a designer more control over the shape of the output waveform because $k$, $T$, and $V_{th}$ can be varied.

EXAMPLE 16.10

Designing a sawtooth-wave generator  Design the sawtooth-wave generator shown in Fig. 16.40(a) so that $f_o = 4$ kHz, $V_{th} = 5$ V, and the circuit has a duty cycle of $k = 0.25$. Assume $V_{sat} = | -V_{sat} | = 14$ V.

SOLUTION

The steps used to design the sawtooth-wave generator are as follows:

Step 1. Find the value of $V_{ref}$ required to obtain the desired duty cycle $k$. From Eq. (16.47),

$$V_{ref} = (2k - 1)V_{sat}
$$

$$= (2 \times 0.25 - 1) \times 14 = -7 \text{ V}
$$

Step 2. Find the values of $R_1$ and $R_F$. Equation (16.34) gives

$$\frac{R_1}{R_F} = \frac{V_{th}}{V_{sat}} = \frac{5}{14} = 0.36
$$

Let $R_1 = 10 \text{ k}\Omega$; then

$$R_F = \frac{R_1}{0.36} = \frac{10 \text{ k}\Omega}{0.36} = 28 \text{ k}\Omega \quad \text{(use a 30-k}\Omega \text{ potentiometer)}
$$

Step 3. Choose a suitable value of $C$: Let $C = 0.01 \mu F$.

Step 4. Find the value of $R$. Equation (16.46) gives

$$R = \frac{V_{sat}^2 - V_{ref}^2}{4f_oCV_{th}V_{sat}}
$$

$$= \frac{14^2 V^2 - (-7)^2 V^2}{4 \times 4 \text{ kHz} \times 0.01 \mu F \times 5 \text{ V} \times 14 \text{ V}} = 13.1 \text{ k}\Omega \quad \text{(use a 15-k}\Omega \text{ potentiometer)}
$$

KEY POINT OF SECTION 16.8

- A sawtooth waveform has unequal rise and fall times. The rise time may be faster than the fall time or vice versa. A triangular-wave generator can be converted to a sawtooth-wave generator by adding a variable DC voltage $V_{ref}$ to the noninverting terminal of the op-amp.
16.9 Voltage-Controlled Oscillators

A voltage-controlled oscillator (VCO) is an oscillator circuit in which the oscillation frequency is controlled by an externally applied voltage [5, 6]. A linear relationship between the oscillation frequency $f_0$ and the control voltage $v_C$ is often required in a VCO. A VCO is also called a voltage-to-frequency (V/F) converter. VCOs are employed in many applications such as frequency modulation (FM), tone generation, and frequency-shift keying (FSK).

To convert a voltage to a frequency, a capacitor is normally charged and then discharged at a constant current whose value depends on an externally applied voltage. The charging begins when the capacitor voltage falls to a lower threshold voltage $V_L$. Similarly, the discharging begins when the capacitor voltage rises to an upper threshold voltage $V_H$.

Figure 16.41(a) shows a simplified block diagram of the operation of a VCO. The current sources are used to charge and discharge capacitor $C_1$. The input to the Schmitt trigger is the capacitor voltage; the output of the Schmitt trigger has two threshold voltage switching levels $V_L$ and $V_H$, which control the closing and opening of the current switch. Thus, depending on the capacitor voltage $v_C(t)$, the current switch connects the capacitor either to the top current source for charging or to the bottom current source for discharging. The waveform of the capacitor voltage is shown in Fig. 16.41(b); it has two modes, charging and discharging.

16.9.1 Charging Mode

During charging mode, the capacitor is charged by the top current source $I_Q$ from the lower threshold voltage $V_L$ to the upper trigger level $V_H$. The time required to charge the capacitor from $V_L$ to $V_H$ is given by

$$\Delta t_1 = \frac{C_1}{I_Q} \Delta v_C = \frac{C_1}{I_Q} (V_H - V_L) \quad (16.50)$$

![Block diagram and waveform of capacitor voltage](image)
16.9.2 Discharging Mode

The discharging mode begins when the capacitor is charged to the upper trigger level $V_H$. At this point the current switch disconnects the top current source from $C_1$ and connects the bottom current source to $C_1$. The capacitor is then discharged by the bottom current source down to the lower trigger level $V_L$. The time required to discharge the capacitor from $V_H$ to $V_L$ is given by

$$\Delta t_2 = -\frac{C_1}{I_Q} \Delta v_C = -\frac{C_1}{I_Q} (V_L - V_H) = \frac{C_1}{I_Q} (V_H - V_L)$$  \hspace{1cm} (16.51)

As long as the charging and discharging currents are maintained at the same magnitude of $I_Q$, $\Delta t_1 = \Delta t_2$. The period of oscillation $T$ is given by

$$T = \Delta t_1 + \Delta t_2 = \frac{2C_1(V_H - V_L)}{I_Q}$$  \hspace{1cm} (16.52)

which gives the frequency of oscillation $f_o$ as

$$f_o = \frac{1}{T} = \frac{I_Q}{2C_1(V_H - V_L)}$$  \hspace{1cm} (16.53)

Let us assume that the voltage-controlled current sources have a linear voltage-to-current transfer relationship; that is,

$$I_Q = G_m(v_{CN} + V_{CO})$$  \hspace{1cm} (16.54)

where $G_m =$ transconductance of the current source, in A/V

$v_{CN} =$ applied control voltage, in V

$V_{CO} =$ constant voltage, in V

Therefore, the oscillation frequency will be a linear function of the control voltage $v_{CN}$; that is,

$$f_o = \frac{I_Q}{2C_1(V_H - V_L)} = \frac{G_m(v_{CN} + V_{CO})}{2C_1(V_H - V_L)}$$  \hspace{1cm} (16.55)

which gives the voltage-to-frequency transfer coefficient $K_{vf}$ as

$$K_{vf} = \frac{df_o}{dv_{CN}} = \frac{G_m}{2C_1(V_H - V_L)}$$  \hspace{1cm} (16.56)

16.9.3 Circuit Implementation

A circuit implementation for the charging and discharging of the capacitor is shown in Fig. 16.42. It can be divided into two parts: a voltage-controlled current source and a current switch.

Voltage-Controlled Current Source

The voltage-controlled current source consists of an $n$-$p$-$n$ transistor $Q_1$ and a $p$-$n$-$p$ transistor $Q_2$. The voltage at the emitter of $Q_2$ is given by

$$V_{E2} = v_{CN} - V_{BE1} + V_{BE2}$$
which gives the current source $I_Q$ flowing through R as

$$I_Q = \frac{V_{CC} - V_{E2}}{R} = \frac{V_{CC} - v_{CN} + V_{BE1} - V_{EB2}}{R} \quad (16.57)$$

Since $V_{BE1}$ (for an npn transistor) $\approx V_{EB2}$ (for a pnp transistor) within the range from 10 mV to 50 mV, $I_Q$ in Eq. (16.57) can be approximated by

$$I_Q \approx \frac{V_{CC} - v_{CN}}{R} \quad (16.58)$$

which gives a linear relationship between the current source $I_Q$ and the control voltage $v_{CN}$.

**Current Switch**

The current switch consists of diodes $D_1$ and $D_2$ and transistors $Q_3$, $Q_4$, and $Q_5$. Transistor $Q_5$ is controlled by the Schmitt trigger and is operated as an on or off switch. When $Q_5$ is off, capacitor $C$ is charged by the current source $I_Q$ via diode $D_1$. Diode $D_2$ and transistors $Q_3$ and $Q_4$ are off. When $Q_5$ is turned on (in saturation) by the Schmitt trigger, $D_2$, $Q_3$, and $Q_4$ are also turned on. As a result, the current source $I_Q$ flows through $D_2$ and $Q_4$ instead of via diode $D_1$. The voltage at the anode of diode $D_2$ becomes

$$V_x = V_{D2(\text{anode})} = V_{D2} + V_{BE4} + V_{CES(\text{sat})} = 0.6 + 0.6 + 0.2 = 1.4 \text{ V}$$

However, the voltage at the cathode of diode $D_1$ is the capacitor voltage $v_C(t)$, which will be larger than 1.4 V. Thus, diode $D_1$ will be reverse biased (i.e., off). Capacitor $C$ will discharge through $Q_3$ at a rate of $I_Q$. The current through $Q_3$ is equal to the current through $Q_4$. Thus, transistor $Q_5$ will carry a current of $2I_Q$. 

**FIGURE 16.42** Circuit implementation
A typical example of a VCO-integrated circuit is the NE/SE-566 VCO, whose pin diagram is shown in Fig. 16.43(a) and whose internal block diagram is shown in Fig. 16.43(b). This VCO produces simultaneous square-wave and triangular-wave outputs at frequencies of up to 1 MHz. Both outputs are buffered so that the output impedance of each is $50 \, \Omega$. The typical amplitude of the square wave is 5.4 V pp (peak to peak), and that of the triangular wave is 2.4 V pp. A typical connection diagram is shown in Fig. 16.43(c), and the typical outputs are shown in Fig. 16.43(d).

The output frequency is determined by an external resistor $R_1$, capacitor $C_1$, and the voltage $v_{CN}$ applied to control terminal 5. The nominal DC value $V_{CN}$ of $v_{CN}$ is set by the voltage divider formed by $R_2$ and $R_3$ and must be in the range of

$$V_{CN} = \frac{R_3}{R_2 + R_3} V_{CC} \tag{16.59}$$

**FIGURE 16.43** Voltage-controlled oscillator NE/SE-566 (Courtesy of Philips Semiconductors)
where \( V_{CC} \) is the DC supply voltage. \( V_{CN} \) must satisfy the following constraint:

\[
\frac{3}{4} V_{CC} \leq V_{CN} \leq V_{CC} \tag{16.60}
\]

The modulating signal \( v_{cn} \) is AC-coupled with capacitor \( C_3 \), and its value must be \( <3 \) V pp. Since \( v_{cn} \) is superimposed on \( V_{CN} \), the control voltage \( v_{CN} \) is the sum of \( V_{CN} \) and \( v_{cn} \); that is, \( v_{CN} = V_{CN} + v_{cn} \). If we substitute \( I_Q = (V_{CC} - v_{CN}) \frac{R_1}{R} \) in Eq. (16.53), the frequency of the output waveforms can be found approximately from

\[
f_o = \frac{V_{CC} - v_{CN}}{2R_1C_1(V_H - V_L)} \tag{16.61}
\]

which, if we assume \( V_H - V_L = V_{CC} / 4 \), can be approximated by

\[
f_o \approx \frac{2(V_{CC} - v_{CN})}{R_1C_1V_{CC}} \tag{16.62}
\]

where \( R_1 \) should be in the range of \( 2 \text{k} \Omega < R_1 < 20 \text{k} \Omega \). The frequency can be varied over a 10-to-1 range by choosing \( R_1 \) between 2 k\( \Omega \) and 20 k\( \Omega \) for a fixed \( v_{CN} \) and constant \( C_1 \) or by choosing the control voltage \( v_{CN} \) for a constant \( RC \) product. A small capacitor of \( C_2 = 0.001 \text{uF} \) should be connected between pins 5 and 6 to eliminate possible oscillations in the internal current control source.

**EXAMPLE 16.11**

**Designing a voltage-controlled oscillator (VCO)**

(a) Design a VCO as in Fig. 16.43(c) that has a nominal frequency of \( f_o = 20 \text{kHz} \). Assume \( V_{CC} = 12 \text{V} \).

(b) Calculate the modulation in the output frequencies if \( v_{CN} \) is varied by \( \pm 10\% \) because of the modulating signal \( v_{cn} \).

**SOLUTION**

(a) The steps used to design the VCO are as follows:

**Step 1.** Find the limiting values of \( V_{CN} \). From Eq. (16.60), \( 9 \text{V} \leq V_{CN} \leq 12 \text{V} \).

**Step 2.** Choose a suitable value of \( C_1 \): Let \( C_1 = 0.001 \text{uF} \).

**Step 3.** Choose a value of \( R_1 \) between 2 k\( \Omega \) and 20 k\( \Omega \): Let \( R_1 = 10 \text{k} \Omega \).

**Step 4.** Find the value of \( v_{CN} \). From Eq. (16.62), \( v_{CN} \) is given by

\[
v_{CN} = V_{CC} \left(1 - \frac{R_1C_1f_o}{2}\right) \tag{16.63}
\]

\[
= 12 \left(1 - \frac{10 \text{k} \Omega \times 0.001 \text{uF} \times 20 \text{kHz}}{2}\right) = 12 \times 0.9 = 10.8 \text{V}
\]

which falls within the range specified in step 1. If the calculated value falls outside of the specified range, choose a different value for \( R_1 \) and/or \( C_1 \) and recalculate \( v_{CN} \).
Step 5. Find the values of $R_2$ and $R_3$. Letting $V_{CN} = 10.8 \text{ V}$ from step 4, Eq. (16.59) gives

$$1 + \frac{R_2}{R_3} = \frac{V_{CC}}{V_{CN}}$$

$$= \frac{12}{10.8} = 1.11$$

so $R_2/R_3 = 1.11 - 1 = 0.11$. Let $R_3 = 100 \text{ k\Omega}$; then

$$R_2 = 0.11 \times R_3 = 11 \text{ k\Omega}$$

(b) For a 10% increase in $v_{CN}$,

$$v_{CN}^+ = V_{CN} + v_{cn} = 1.1 \times v_{CN} = 1.1 \times 10.8 = 11.88 \text{ V}$$

The corresponding value of output frequency can be calculated from Eq. (16.62):

$$f_{o1} \approx \frac{2 \times (12 - 11.88) \text{ V}}{10 \text{ k\Omega} \times 0.001 \mu \text{F} \times 12 \text{ V}} = 2 \text{ kHz}$$

For a 10% decrease in $v_{CN}$,

$$v_{CN}^- = V_{CN} - v_{cn} = 0.9 \times V_C = 0.9 \times 10.8 = 9.72 \text{ V}$$

The output frequency is

$$f_{o2} \approx \frac{2 \times (12 - 9.72) \text{ V}}{10 \text{ k\Omega} \times 0.001 \mu \text{F} \times 12 \text{ V}} = 38 \text{ kHz}$$

Thus, the change in the output frequency is

$$\Delta f_o = f_{o2} - f_{o1} = 38 \text{ kHz} - 2 \text{ kHz} = 36 \text{ kHz}$$

Using Eq. (16.62), we can find the $V \mu \text{F}$ transfer coefficient $K_{vf}$:

$$K_{vf} = \frac{df_o}{dv_{CN}} = -\frac{2}{R_1C_1V_{CC}}$$

$$= -\frac{2}{10 \text{ k\Omega} \times 0.001 \mu \text{F} \times 12 \text{ V}} = -16.67 \text{ kHz/V}$$

Thus, for $\Delta v_{CN} = 2v_{CN} = -2 \times 0.1 \times 10.8 = -2.16 \text{ V}$,

$$\Delta f_o = f_{o2} - f_{o1} = K_{vf} \Delta v_{CN} = -16.67 \text{ kHz/V} \times (-2.16 \text{ V}) = 36.01 \text{ kHz}$$

which is the same as the value obtained by calculating individual frequencies.

NOTE: If the modulating signal is a sine wave so that $v_{cn} = V_m \sin \omega t$, then the control voltage becomes $v_{CN} = V_{CN} + V_m \sin \omega t$. During the positive half-cycle of the modulating signal, the control voltage will increase and the frequency $f_o$ of the output voltage will decrease. However, during the negative half-cycle of the modulating signal, the control voltage will decrease and the frequency $f_o$ of the output voltage will increase.
16.10 The 555 Timer

The 555 timer, introduced by Signetics Corporation in early 1970, is one of the most versatile integrated circuits. The 555 is a monolithic timing circuit that can produce highly accurate and highly stable delays or oscillation. It is used in many applications, such as monostable and astable multivibrators, digital logic probes, analog frequency meters, tachometers, infrared transmitters, and burglar and toxic gas alarms. Several versions of the 555 timer are available from various manufacturers. In addition to looking briefly at its internal structure, we consider two common applications of the 555 timer: monostable multivibrator and astable multivibrator.

16.10.1 Functional Block Diagram

The pin diagram of a 555 timer is shown in Fig. 16.44(a); its functional block diagram is shown in Fig. 16.44(b). The timer consists of two comparators CM1 and CM2, an RS flip-flop, a discharge transistor Q1, and a resistive voltage divider string. The voltage divider sets the voltage at the inverting terminal of CM1 to $2V_{CC}/3$ and the voltage at the noninverting terminal of CM2 to $V_{CC}/3$.

The reset, threshold, and trigger inputs control the state of the flip-flop. If the reset input is low, the Q output of the flip-flop is low, and $\bar{Q}$ is high. With $\bar{Q}$ high, current flows through the base of transistor Q1, and the transistor is switched on (in saturation). This generally provides a path for an external capacitor to discharge its voltage.

The reset input has the highest priority in setting the state of the flip-flop. Thus, Q is low if the reset input is low, regardless of the inputs to the comparators. If the reset is not in use, then it is connected to the positive DC supply $V_{CC}$ so that it does not affect the state of the flip-flop.

If the trigger input becomes lower than the voltage at the noninverting input of CM2 (i.e., $V_{CC}/3$), the output of CM2 (i.e., the S input to the flip-flop) will be high. As a result, the Q output of the flip-flop will be set to high. Thus, $\bar{Q}$ will be low, and the discharge transistor Q1 will be off.

If the threshold input becomes higher than the voltage at the inverting input of CM1 (i.e., $2V_{CC}/3$), the output of CM1 will be high. As a result, the Q output of the flip-flop will be reset to low. Thus, $\bar{Q}$ will be high, and the discharge transistor Q1 will be on (in saturation), providing a discharge path.
16.10.2 Monostable Multivibrators

A monostable multivibrator is a one-shot pulse-generating circuit. Normally, its output is zero—that is, at logic low level in the stable state. This circuit has only one stable state at output low—hence the name monostable. The circuit configuration of the 555 timer for monostable operation is shown in Fig. 16.45(a). The discharge and threshold terminals are connected together. The external pulse $v_I$ is applied to the trigger terminal through coupling capacitor $C_2$.

If the external pulse $v_I$ is high, the $Q$ output of the flip-flop is low; that is, the output of the timer is low. At the negative edge of the trigger signal, the flip-flop will be set to high. Thus, the output will be switched to high ($V_{CC}$). It will remain high until the capacitor is charged to the threshold voltage of $2V_{CC}/3$, at which time the flip-flop will be reset and the output will return to zero.

The duration of the output pulse ($t_p$) is determined by the $RC$ network connected externally to the 555 timer. At the end of the timing interval $t_p$, the output automatically reverts to its stable state of logic low. The output remains low until another negative-going trigger pulse is applied. Then the cycle repeats. The waveforms for the trigger input voltage $v_I(t)$, output voltage $v_O(t)$, and capacitor voltage $v_C(t)$ are shown in Fig. 16.45(b).

The width of the triggering pulse must be smaller than the expected pulse width of the output waveform. Also, the trigger pulse must be a negative-going signal and have an amplitude larger than $V_{CC}$. The time during which the output remains high is given by

$$t_p = 1.1RC$$

(16.66)
It is important to note that once the monostable multivibrator is triggered and the output is in the high state, another trigger pulse will have no effect until after an interval of $t_p$. That is, the multivibrator cannot be retriggered during the timing interval $t_p$.

A decoupling capacitor $C_3$, typically of $10 \mu F$, is normally connected between $V_C C$ (pin 8) and ground (pin 1) to eliminate unwanted voltage spikes in the output waveform. A waveshaping circuit consisting of $R_2$, $C_2$, and diode $D_1$ is often connected between the trigger input (pin 2) and $V_C C$ (pin 8) to prevent any possible mistriggering by positive pulse edges. This circuit is shown in Fig. 16.45(a) within the shaded area. The values of $R_2$ and $C_2$ should be selected so that the output pulse width $t_p$ is much larger than the time constant $R_2C_2$. This condition is generally met by maintaining the following relationship:

$$ t_p = 10R_2C_2 $$  \hspace{1cm} (16.67)

**EXAMPLE 16.12**

Designing a monostable multivibrator  
Design a monostable multivibrator as in Fig. 16.45(a) so that $t_p = 5$ ms. Assume $V_C C = 12$ V.

**SOLUTION**

The steps used to design a monostable multivibrator are as follows:

**Step 1.** Choose a suitable value of $C$: Let $C = 0.1 \mu F$.

**Step 2.** Find the value of $R$. From Eq. (16.66),

$$ R = \frac{t_p}{1.1C} = \frac{5 \text{ ms}}{1.1 \times 0.1 \mu F} = 45.5 \text{ k}\Omega \quad \text{(use a 50-k}\Omega \text{ potentiometer)} $$

**Step 3.** Choose a suitable value of $C_2$: Let $C_2 = 0.01 \mu F$.

**Step 4.** Find the value of $R_2$. From Eq. (16.67),

$$ R_2 = \frac{t_p}{10C_2} = \frac{5 \text{ ms}}{10 \times 0.01 \mu F} = 50 \text{ k}\Omega $$
16.10.3 Applications of Monostable Multivibrators

Monostable multivibrators can be used in many applications such as frequency dividers, missing-pulse detectors, and pulse stretchers.

**Frequency Divider**

If the frequency of the input signal is known, adjusting the length of the timing cycle \( t_p \) will permit a monostable multivibrator to be used as a frequency divider. The circuit configuration for a frequency divider is shown in Fig. 16.46(a). This application makes use of the fact that the monostable multivibrator cannot be retrigged during the timing interval.

For a monostable multivibrator to be used as a divide-by-2 circuit, the timing interval must be slightly larger (say, by 20%) than the period of the trigger signal \( T \), as shown in Fig. 16.46(b); that is, \( t_p = 1.2T \). At the first falling edge, the output is set to high. However, at the second falling edge, the capacitor is still charging and has not yet reached the threshold value of \( 2 \frac{V_{CC}}{3} \). As a result, the second triggering signal has no effect on the output, and the output is set by the alternate trigger pulses.

For a monostable multivibrator to be used as a divide-by-3 circuit, \( t_p \) must be slightly larger than twice the period of the trigger signal. Thus, for a divide-by-\( n \) circuit, \( t_p \) must be slightly larger than \( (n - 1)T \); that is, \( t_p = (0.2 + (n - 1))T \). For example, for a divide-by-2 circuit, if \( f_o = 5 \text{ kHz} \) and \( T = 1 f_o = 200 \mu s \), then \( t_p = 1.2T = 240 \mu s \).

**Missing-Pulse Detector**

In some applications, a train of regular pulses is needed for the normal operation of a circuit or system. Any missing pulse may cause malfunction. The configuration of a monostable multivibrator that can detect any missing pulse is shown in Fig. 16.47(a).

The timing cycle is continuously reset by the input pulse train, but the time duration is not enough to complete the timing cycle. When the trigger pulse becomes low, transistor \( Q_1 \) is turned on, and it provides a discharge path for capacitor \( C \). As a result, the capacitor voltage cannot reach the threshold voltage \( 2V_{CC} \). A change in frequency, or a missing pulse, allows completion of the timing cycle so that \( v_C \)
reaches $2V_{CC}$, causing a change in the output level due to the discharge of $C$ through the internal transistor $Q_1$ in Fig. 16.44(b). The time delay $t_p$ should be slightly longer than the normal time between input pulses. The waveforms are shown in Fig. 16.47(b).

**Pulse Widener**

A narrow pulse is not desirable for driving an LED display, as the flashing of the LED will not be visible to the eyes if the on time is infinitesimally small compared to the off time. A narrow pulse can be widened by a monostable multivibrator. This application is possible because of the fact that the timing interval $t_p$ is longer than the negative pulse width of the trigger input.

The circuit configuration for a pulse widener is shown in Fig. 16.48(a). At the falling trigger edge, the output will be high, and it will remain high until the capacitor voltage reaches $2V_{CC}$ after time $t_p$. The waveforms of the input and output voltages are shown in Fig. 16.48(b). Since the output pulse can be viewed as the stretched version of the narrow input signal, this configuration is also known as a *pulse stretcher*.
16.10.4 Astable Multivibrators

An astable multivibrator is a rectangular-wave-generating circuit. Because this circuit does not require an external trigger to change the state of the output, it is often called a free-running multivibrator. A 555 timer connected as an astable multivibrator is shown in Fig. 16.49(a). The duration of the high or low output is determined by resistors \( R_A \) and \( R_B \) and capacitor \( C \). When the output is high, capacitor \( C \) starts charging toward \( V_{CC} \) through \( R_A \) and \( R_B \). As soon as the capacitor voltage equals \( \frac{2}{3} V_{CC} \), the output switches to low, and capacitor \( C \) discharges through \( R_B \) and the internal circuit of the timer. When the capacitor voltage equals \( \frac{1}{3} V_{CC} \), the output goes high, and the capacitor charges through \( R_A \) and \( R_B \). Then the cycle repeats. The waveforms for the output voltage and the voltage across the capacitor are shown in Fig. 16.49(b).

The capacitor is periodically charged and discharged between \( \frac{2}{3} V_{CC} \) and \( \frac{1}{3} V_{CC} \). Assuming the initial capacitor voltage is \( V_{CO} = \frac{1}{3} V_{CC} \), the equivalent circuit during the charging period is as shown in Fig. 16.49(c). The charging current \( i_C(t) \) and the capacitor voltage \( v_C(t) \) are given by

\[
i_C(t) = \frac{2V_{CC}}{3(R_A + R_B)} e^{-\frac{t}{(R_A+R_B)C}}
\]

\[
v_C(t) = V_{CC} - \frac{2V_{CC}}{3} e^{-\frac{t}{(R_A+R_B)C}}
\]  

At \( t = t_c \), \( v_C(t = t_c) = \frac{2}{3} V_{CC} \), and Eq. (16.69) yields

\[
\frac{2V_{CC}}{3} = V_{CC} - \frac{2V_{CC}}{3} e^{-\frac{t_c}{(R_A+R_B)C}}
\]

which gives the charging time \( t_c \) as

\[
t_c = C(R_A + R_B) \ln(2) = 0.69C(R_A + R_B)
\]
During time $t_d$, capacitor C discharges from $2V_{CC}$ to $V_{CC}/3$ through $R_B$. Assuming the initial capacitor voltage is $V_{CO} = 2V_{CC}/3$, the equivalent circuit during the discharging period is as shown in Fig. 16.49(d). The current $i_C(t)$ and capacitor voltage $v_C(t)$ are given by

$$i_C(t) = \frac{2V_{CC}}{3R_B} e^{-t/R_B}$$  \hspace{1cm} (16.71) \\
$$v_C(t) = \frac{2V_{CC}}{3} e^{-t/R_B}$$  \hspace{1cm} (16.72)

At $t = t_d$, $v_C(t = t_d) = V_{CC}/3$, and Eq. (16.72) yields

$$v_C(t = t_d) = \frac{2V_{CC}}{3} e^{-t_d/R_B}$$

which gives the discharging time $t_d$ as

$$t_d = CR_B \ln(2) = 0.69CR_B$$  \hspace{1cm} (16.73)

Thus, the period of the output waveform is given by

$$T = t_c + t_d = 0.69C(R_A + R_B) + 0.69CR_B = 0.69C(R_A + 2R_B)$$  \hspace{1cm} (16.74)

and the frequency of the output voltage is therefore given by

$$f_o = \frac{1}{T} = \frac{1}{0.69C(R_A + 2R_B)} = \frac{1.45}{C(R_A + 2R_B)}$$  \hspace{1cm} (16.75)

Duty cycle $k$, which is the ratio of charging time $t_c$ to period $T$, can be found from Eqs. (16.70) and (16.74):

$$k = \frac{t_c}{T} = \frac{R_A + R_B}{R_A + 2R_B}$$  \hspace{1cm} (16.76)

Thus, the duty cycle $k$ can be set by selecting $R_A$ or $R_B$. 

**EXAMPLE 16.13**

**Designing an astable multivibrator**  Design an astable multivibrator as in Fig. 16.49(a) so that $k = 75\%$ and $f_o = 2.5$ kHz. Assume $V_{CC} = 12$ V. Use PSpice to verify by simulating, EX16-13.SCH.

**SOLUTION**

$k = 75\% = 0.75$, and $T = 1/f_o = 1$ 2.5 kHz = 400 $\mu$s. The steps used to design an astable multivibrator are as follows:

**Step 1.** Find the charging time $t_c$ and the discharging time $t_d$:

$$t_c = kT$$  \hspace{1cm} (16.77) \\
$$= 0.75 \times 400 \mu s = 300 \mu s$$

$$t_d = (1 - k)T$$  \hspace{1cm} (16.78) \\
$$= (1 - 0.75) \times 400 \mu s = 100 \mu s$$
Step 2. Choose a suitable value of $C$: Let $C = 0.1 \ \mu F$.

Step 3. Find the value of $R_B$. From Eq. (16.73),

$$R_B = \frac{t_d}{0.69C}$$

$$= \frac{100 \ \mu s}{0.69 \times 0.1 \ \mu F} = 1449 \ \Omega$$

Step 4. Find the value of $R_A$. From Eq. (16.70),

$$R_A = \frac{t_c}{0.69C} - R_B$$

$$= \frac{300 \ \mu s}{0.69 \times 0.1 \ \mu F} - 1449 = 2899 \ \Omega$$

EXAMPLE 16.14

**PSpice/SPICE simulation of an astable multivibrator** An astable multivibrator can be used as a voltage-controlled oscillator (VCO) if an external control voltage is applied to terminal 5. This arrangement is shown in Fig. 16.50 for $v_C = 6 + 4 \sin (2000\pi t)$. Use PSpice/SPICE to plot the output voltage $v_O(t)$ from 0 to 2 ms with an increment of 10 ns.

**FIGURE 16.50** Astable multivibrator as a VCO for PSpice simulation
16.10.5 Applications of Astable Multivibrators

As examples of applications of an astable multivibrator, we consider a square-wave generator, a ramp generator, and an FSK modulator.

Square-Wave Generator

The astable multivibrator in Fig. 16.49(a) can be modified to produce a square wave. A diode is connected across resistor $R_B$, as shown in Fig. 16.52. For a finite value of $R_A$ in Eq. (16.70), $t_c > t_d$, and the duty cycle is $k > 50\%$. However, to obtain a square wave, the duty cycle $k$ must be 50\%; that is, the...
value of $R_A$ should be set to zero. With $R_A = 0$, terminal 7 is connected directly to $V_{CC}$. During the discharging time, capacitor $C$ discharges through the internal circuit of the timer, and an external current is applied by $V_{CC}$ to the same internal path. The current flowing through the internal path may be large enough to damage the timer. This situation can be avoided by connecting a diode across $R_B$ so that $R_B$ is bypassed during the charging time. In that case, Eqs. (16.74) and (16.75) are reduced, respectively, to

$$T = 0.69C(R_A + R_B) \quad \text{(for } k = 0.5)$$

$$f_o = \frac{1}{T} = \frac{1.45}{C(R_A + R_B)} \quad \text{(for } k = 0.5)$$

**NOTE** We need both $R_A$ and $D_1$. Making $R_A = 0$, the circuit will not work.

**EXAMPLE 16.15**

**Designing a square-wave generator** Design a square-wave generator as in Fig. 16.52 so that $k = 50\%$ and $f_o = 2.5 \text{ kHz}$. Assume $V_{CC} = 12 \text{ V}$. Use PSpice to verify by simulating, EX16-15.SCH.

**SOLUTION**

$k = 50\% = 0.5$, and $T = 1 f_o = 1 \ 2.5 \text{ kHz} = 400 \text{ } \mu\text{s}$. The steps used to design the square-wave generator are as follows:

**Step 1.** Find the charging time $t_c$ and the discharging time $t_d$:

$$t_c = t_d = kT$$

$$= 0.5 \times 400 \text{ } \mu\text{s} = 200 \text{ } \mu\text{s}$$

**Step 2.** Choose a suitable value of $C$: Let $C = 0.1 \text{ } \mu\text{F}$.

**Step 3.** Find the value of $R_B$. From Eq. (16.73),

$$R_B = \frac{t_d}{0.69C}$$

$$= \frac{200 \text{ } \mu\text{s}}{0.69 \times 0.1 \text{ } \mu\text{F}} = 2899 \text{ } \Omega$$

**Step 4.** Find the value of $R_A$. From Eq. (16.82),

$$R_A = \frac{1.45}{Cf_o} - R_B$$

$$= \frac{1.45}{0.1 \text{ } \mu\text{F} \times 2.5 \text{ kHz}} - 2899 \text{ } \Omega = 291 \text{ } \Omega$$

**Ramp Generator**

The astable multivibrator in Fig. 16.49(a) can be used as a free-running ramp generator. This is accomplished by charging the capacitor through a constant current source and discharging through the internal circuit of the timer. That is, resistors $R_A$ and $R_B$ are replaced by a current source, as shown in Fig. 16.53(a). The waveforms for the output voltage and the capacitor voltage are shown in Fig. 16.53(b).
The collector current, which is the charging current, is given by

\[ I_C = I_E - I_B \]

Assuming that the voltage drop of diode D1 is approximately equal to the base–emitter voltage \( V_{BE} \) of the transistor, the diode current \( I_D \approx I_E \). Thus,

\[ I_C = I_D - I_B = I_2 = \frac{V_B}{R} = \frac{V_{CC} - V_{BE}}{R} \]  \hspace{1cm} (16.86)

The capacitor charges from \( V_{CC}/3 \) to \( 2V_{CC}/3 \) at a constant current of \( I_C \). For a charging time of \( t_c \), the change in the capacitor voltage \( \Delta V_C \) is given by

\[ \Delta V_C = 2V_{CC}/3 - V_{CC}/3 = \frac{1}{C} \int_0^{t_c} I_C dt = \frac{1}{C} I_C t_c \]

which gives the charging time \( t_c \) as

\[ t_c = \frac{CV_{CC}}{3I_C} \]  \hspace{1cm} (16.87)

The charging time is related to the duty cycle \( k \) and period \( T \) by

\[ kT = t_c = \frac{CV_{CC}}{3I_C} \]  \hspace{1cm} (16.88)

Therefore, the free-running frequency of the ramp generator is given by

\[ f_o = \frac{1}{T} = \frac{3kI_C}{CV_{CC}} \]

\[ = \frac{3k(V_{CC} - V_{BE})}{CRV_{CC}} \]  \hspace{1cm} (16.90)
If the discharging time \( t_d \) of the capacitor is negligible compared to its charging time \( t_c \), then \( k = 1 \) and the free-running frequency becomes
\[
f_o = \frac{3(V_{CC} - V_{BE})}{CRV_{CC}}
\] (16.91)

**EXAMPLE 16.16**

**Designing a ramp generator** Design a ramp generator using the circuit in Fig. 16.53(a) so that \( k = 50\% \) and \( f_o = 2.5 \text{ kHz} \). Assume \( V_{CC} = 12 \text{ V} \), \( V_{BE} = 0.7 \text{ V} \), and a transistor of \( \beta_F = 150 \).

**SOLUTION**

\( k = 50\% = 0.5 \), and \( T = 1 f_o = 1 \text{ 2.5 kHz} = 400 \mu\text{s} \). The steps used to design the ramp generator are as follows:

**Step 1.** Find the charging time \( t_c \) and the discharging time \( t_d \):
\[
t_c = t_d = kT = 0.5 \times 400 \mu\text{s} = 200 \mu\text{s}
\] (16.92)

**Step 2.** Choose a suitable value of \( C \): Let \( C = 0.1 \mu\text{F} \).

**Step 3.** Find the value of \( R \). From Eq. (16.90),
\[
R = \frac{3k(V_{CC} - V_{BE})}{V_{CC}f_o} = \frac{3 \times 0.5 \times (12 - 0.7) \text{ V}}{12 \times 0.1 \mu\text{F} \times 2.5 \text{ kHz}} = 5.65 \text{ k}\Omega
\] (16.93)

**Step 4.** Find the collector current \( I_C \) of the transistor. From Eq. (16.86),
\[
I_C = \frac{V_{CC} - V_{BE}}{R} = \frac{(12 - 0.7) \text{ V}}{5.65 \text{ k}\Omega} = 2 \text{ mA}
\]

**Step 5.** Find the current \( I_D \) through the diode:
\[
I_D = I_E = I_C + I_B = I_C \frac{1 + \beta_F}{\beta_F} = 2 \text{ mA} \times \frac{1 + 150}{150} \approx 2.01 \text{ mA}
\]

**FSK Modulator**

In computer peripheral and radio (wireless) communication, the binary data or code is transmitted by means of a carrier frequency that shifts between two preset frequencies. This technique for data transmission is called frequency-shift keying (FSK). The frequency shift is usually accomplished by driving a VCO with the binary data signal so that the 0 to 1 states (commonly called space and mark) of the binary data signal produce two frequencies, known as the space and mark frequencies. For example, when teletypewriter information is transmitted using a modulator/demodulator (modem, for short), a 1070-Hz (for mark) and 1270-Hz (for space) pair will represent the original signal, whereas a 2025-Hz (for mark) and 2250-Hz (for space) pair will represent the answer signal.
FSK modulators are often used in AM/FM transmitters, as shown in Fig. 16.54(a). The 555 astable multivibrator can be used as an FSK generator; the connection is shown in Fig. 16.54(b). The on or off condition of transistor Q1 will depend on the input signal. Thus, the output frequency depends on the logic state of the digital input signal. A signal frequency of 150 Hz is commonly used for data transmission.

When the input signal is 1, transistor Q1 is off, and the 555 operates in its normal mode as an astable multivibrator. Thus, the output frequency corresponding to logic 1 can be found from Eq. (16.75) as

\[ f_{o(\text{mark})} = \frac{1.45}{C(R_A + 2R_B)} \]

The values for \( C, R_A, \) and \( R_B \) can be selected so as to give 1070 Hz.

When the input signal is 0, transistor Q1 is turned on (in saturation). As a result, \( R_C \) is effectively connected in parallel with \( R_B \). This reduces the charging time of capacitor \( C \) and increases the output frequency. Thus, the output frequency corresponding to logic 0 can be found from

\[ f_{o(\text{space})} = \frac{1.45}{C(R_A \parallel R_C + 2R_B)} \]

The value for \( R_C \) can be selected so as to give 1270 Hz.

Thus, with properly selected values of \( C, R_A, R_B, \) and \( R_C \), the 555 astable multivibrator can produce frequencies of 1070 Hz and 1270 Hz corresponding to 1 and 0, respectively. The difference between the FSK signals of 1270 Hz and 1070 Hz (i.e., 200 Hz) is called the frequency shift.
16.11 Phase-Lock Loops

The phase-lock loop (PLL) is one of the fundamental building blocks of electronic circuits used in such applications as motor-speed controllers, FM stereo decoders, tracking filters, frequency-synthesized transmitters and receivers, and FSK decoders. A block diagram of a phase-lock loop is shown in Fig. 16.55(a). The loop consists of a phase detector, a low-pass filter, and a VCO.

The phase detector (or comparator) compares the phase of the input voltage with that of the VCO output voltage and produces a DC or low-frequency voltage proportional to their phase difference. The output of the phase detector, which is called the error voltage, is applied to a low-pass filter. The filter removes any high-frequency components and produces a smooth DC voltage. This DC voltage is then applied to the control input of the VCO, whose output frequency is proportional to the DC value. If the frequency of the input voltage shifts slightly, the phase difference between the input signal and the VCO output voltage will begin to increase with time. This will change the control voltage to the VCO in such a way as to bring the VCO frequency back to the same value as the input voltage. The VCO frequency is continuously adjusted until it is equal to the input frequency.

The operation of a PLL involves three modes: a free-running mode, a capture mode, and a phase-lock mode. During the free-running mode, there is no input frequency (or voltage), and the VCO runs at a fixed frequency corresponding to zero-applied input voltage. This frequency is called the center, or free-running, frequency $f_o$. Once an input frequency is applied, the VCO frequency starts to change, and the PLL is said...
to be in the capture mode. The VCO frequency changes continuously to match the input frequency. When
the input frequency becomes equal to the output frequency, the PLL is said to be in the phase-lock mode.
The feedback loop maintains the lock when the input signal frequency changes.

The center frequency \( f_o \) is the free-running frequency of the VCO. The lock range \( f_L \) is defined as the
range of input frequencies around the center frequency for which the loop can maintain lock. The capture
range \( f_c \) is defined as the range of input frequencies around the center frequency for which the loop will
become locked from an unlocked condition. The relations among \( f_o, f_L, \) and \( f_c \) are shown in Fig. 16.55(b).

### 16.11.1 Phase Detector

A phase detector takes two input voltages and produces a DC voltage proportional to their phase difference.
To understand the principle of operation, consider two voltages \( v_{I1} \) and \( v_{I2} \), as shown in Fig. 16.56(a), with a
phase difference of \( \phi \). An output voltage is obtained when they differ in phase—that is, when only one input
is high. A phase detector can be implemented using either an exclusive-OR gate, as shown in Fig. 16.56(b),
or an analog multiplier [7]. Integrating the output voltage will give an average output voltage, which will be a
linear function of the phase difference \( \phi \), as shown in Fig. 16.56(c). The average output voltage \( V_{O(DC)} \) can be
expressed as

\[
V_{O(DC)} = \begin{cases} 
\frac{V_C}{\pi} \phi & \text{(for } 0 \leq \phi \leq \pi) \\
\frac{V_C}{\pi} (2\pi - \phi) & \text{(for } \pi \leq \phi \leq 2\pi) 
\end{cases} \tag{16.94}
\]

The phase difference can also be detected by using an edge-triggered RS flip-flop. Two input signals
are shown in Fig. 16.57(a). If these signals are passed through an edge-triggered RS flip-flop as shown in
Fig. 16.57(b), the output voltage will be as shown in Fig. 16.57(a). Integrating the output voltage will give an
average output voltage, as shown in Fig. 16.57(c). The average output voltage \( V_{O(DC)} \) is given by

\[
V_{O(DC)} = \frac{V_C}{2\pi} \phi \quad \text{(for } 0 \leq \phi \leq 2\pi) \tag{16.95}
\]

Phase detectors can be broadly divided into two types: digital detectors and analog detectors. The digital
detectors are simple to implement with digital devices. However, they are sensitive to the harmonic
content of the input signal and changes in the duty cycles of the input signal and the VCO output voltage. Analog detectors are monolithic types such as CMOS MC4344/4044. They respond only to transitions in the input signals. Thus, sensitivity to harmonic content and duty cycle is not a problem. The output voltage is independent of variations in the amplitude and duty cycle of the input waveform. Analog detectors are generally preferred over digital detectors, especially in applications in which accuracy is a critical factor.

16.11.2 Integrated Circuit PLL

The NE/SE-565 PLL is one of the most commonly used IC devices. The elements of the PLL in Fig. 16.55(a) are built into the 565 IC. The internal block diagram of the 565 is shown in Fig. 16.58(a), and the pin configuration is shown in Fig. 16.58(b). A typical connection diagram for the NE/SE-565 PLL is shown in Fig. 16.58(c). A small capacitor $C_3$, typically of 0.001 μF, is connected between pins 7 and 8 to eliminate possible oscillations. The center frequency of the PLL is given approximately by

$$f_o \approx \frac{1.2}{4R_1C_1}$$

(16.96)

where $R_1$ and $C_1$ are an external resistance and a capacitance connected to pins 8 and 9, respectively. $C_1$ can have any value, but $R_1$ must have a value between 2 kΩ and 20 kΩ. A capacitor $C_2$ is connected between pins 7 and 10 to form a first-order low-pass filter with an internal resistance of 3.6 kΩ. The filter capacitor $C_2$ should be large enough to eliminate variations in the demodulated output voltage at pin 7 in order to stabilize the VCO frequency.

The 565 PLL can typically lock to and track an input signal over a bandwidth of ±60% of the center frequency $f_o$. The lock range $f_L$ is given by

$$f_L = \frac{8f_o}{V_{CC} - V_{EE}}$$

(16.97)

where $V_{CC}$ and $-V_{EE}$ are the positive and negative power supplies, in volts, respectively. The capture range $f_c$ is given by

$$f_c = \left[ \frac{f_L}{2\pi \times 3.6 \times 10^3 C_2} \right]^{1/2}$$

(16.98)
FIGURE 16.58  NE/SE-565 PLL connection diagram
EXAMPLE 16.17

**Designing a PLL** Design a PLL as shown in Fig. 16.58(c) so that $f_o = 2.5$ kHz and $f_c = 50$ Hz. Assume $V_{CC} = -V_{EE} = 12$ V.

**SOLUTION**

The steps used to design the 565 PLL are as follows:

**Step 1.** Choose a suitable value of $C_1$: Let $C_1 = 0.01$ μF.

**Step 2.** Find the value of $R_1$. From Eq. (16.96),

$$R_1 = \frac{1.2}{4C_1f_o} = \frac{1.2}{4 \times 0.01 \, \text{μF} \times 2.5 \, \text{kHz}} = 12 \, \text{kΩ}$$

**Step 3.** Find the lock range $f_L$. From Eq. (16.97),

$$f_L = \frac{8 \times 2.5 \, \text{kHz}}{12 - (-12)} = 833 \, \text{Hz}$$

**Step 4.** Find the value of $C_2$. From Eq. (16.98),

$$C_2 = \frac{f_L}{2\pi \times 3.6 \times 10^3 f_c^2} = \frac{833}{2\pi \times 3.6 \times 10^3 \times 50^2} = 14.17 \, \text{μF}$$

Choose $C_2 = 14$ μF.

### 16.11.3 Applications of the 565 PLL

As examples of applications of the 565 PLL, we consider a frequency multiplier, an FSK demodulator, and an SCA (subsidiary carrier authorization) decoder [8].

**Frequency Multiplier**

A block diagram of a frequency multiplier using the 565 PLL is shown in Fig. 16.59(a). A frequency divider is inserted between the VCO and the phase detector. Since the output frequency of the divider is locked to the input frequency, the VCO will actually be running at a multiple of the input frequency; that is, $f_o = Nf_{in}$, where $N$ is an integer. The amount of multiplication desired can be obtained by selecting the appropriate divide-by network. A typical connection of the 565 PLL to give an output frequency of $f_o = 5f_{in}$ is shown in Fig. 16.59(b).

To set up the circuit, you must know the frequency limits of the input signal. The free-running frequency of the VCO then can be adjusted by means of $R_1$ and $C_1$ so that the output frequency of the divider is midway between the input frequency limits. That is, for $f_{in} = 400$ Hz to 4 kHz, the output frequency would be $f_o = 2$ kHz to 20 kHz, with a midway frequency of $f_{o(mid)} = 11$ kHz. The filter capacitance $C_2$ should be large enough (typically 10 μF) to eliminate variations in the demodulated output voltage (at pin 7) in order to stabilize the VCO frequency. The output of the VCO will be a square wave whose frequency will be the multiple of the input frequency as long as the loop is in lock. The input and output waveforms are shown in Fig. 16.59(c).
FIGURE 16.59 565 PLL as a frequency multiplier
FSK Demodulator

An FSK demodulator is often used in AM/FM receivers, as shown in Fig. 16.60(a). One very useful application of the 565 PLL is as an FSK demodulator to receive FSK signals of 1070 Hz and 1270 Hz; the configuration is shown in Fig. 16.60(b). As the signal appears at the input, the PLL locks to the input frequency and tracks it between the two frequencies, with a corresponding DC shift at the output. The input signal is connected through a coupling capacitor $C_i$ to block the DC level from the FSK receiver. Both input terminals are connected to the ground through identical resistors $R_2$ and $R_3$.

The loop filter capacitor $C_2$ determines the dynamic characteristics of the demodulator, and its value should be smaller than usual to eliminate overshoot on the output pulse. A three-stage $RC$-ladder low-pass filter is used to remove the carrier component from the output. The high cutoff frequency of the ladder filter—that is, $f_H = 1/2\pi RC$—should be approximately halfway between the maximum keying rate (150 Hz) and twice the input frequency ($2 \times 1070$ Hz is approximately 2200 Hz).

The output signal of 150 Hz can be made logic compatible by connecting a voltage comparator between the output and pin 6 of the 565 PLL. $R_2$ and $C_1$ determine the free-running frequency of the VCO. The free-running frequency is adjusted with $R_1$ so as to produce a slightly positive voltage with $f_o = 1070$ Hz.

SCA (Background Music) Decoder

Some FM stations are authorized by the FCC to broadcast uninterrupted background music for commercial use, using a frequency-modulated subcarrier of 67 kHz. This frequency was chosen so as not to interfere

FIGURE 16.60 565 PLL as an FSK demodulator
with the frequency spectrum of normal stereo or monaural FM program material, which is substantially lower. In addition, the level of the subcarrier is only 10% of the amplitude of the combined signal.

A PLL may be used to recover the SCA (subsidiary carrier authorization, or storecast music) signal from the combined signal of many commercial FM broadcast stations. This application involves demodulation of a frequency-modulated subcarrier of the main channel. The SCA signal can be filtered out and demodulated by the 565 PLL without the use of any resonant circuits. A connection diagram is shown in Fig. 16.61. The PLL is tuned to 67 kHz with a 5-kΩ potentiometer; only approximate tuning is required, since the loop will seek the signal.

The demodulated output (pin 7) is passed through a three-stage low-pass filter to provide deemphasis and attenuate the high-frequency noise that often accompanies SCA transmissions. Since no capacitor is provided directly at pin 7, the circuit operates as a first-order loop. The demodulated output signal is on the order of 50 mV, and the frequency response extends to 7 kHz. By connecting the circuit of Fig. 16.61 to a point between the FM discriminator and the deemphasis filter of a commercial-band (home) FM receiver and tuning the receiver to a station that broadcasts an SCA signal, one can obtain hours of commercial-free background music.

**FIGURE 16.61  565 PLL as an SCA (background music) decoder**

KEY POINTS OF SECTION 16.11

- A phase-lock loop (PLL) consists of a phase detector, a low-pass filter, and a voltage-controlled oscillator. PLLs find applications as frequency multipliers, FSK demodulators, and SCA (background music) decoders.
- The operation of a PLL involves three modes: a free-running mode, a capture mode, and a phase-lock mode. The VCO frequency is continuously adjusted until it is equal to the input frequency. When the input frequency becomes equal to the output frequency, the PLL is said to be in the phase-lock mode.
- The 565 PLL can typically lock to and track an input signal over a bandwidth of ±60% of the center frequency.
16.12 Voltage-to-Frequency and Frequency-to-Voltage Converters

The NE/SE-566 VCO discussed in Sec. 16.9 can be used as a voltage-to-frequency converter. In many applications, it is also necessary to convert frequency to voltage. The TelCom 9400 series converters can be used as either voltage-to-frequency (V/F) or frequency-to-voltage (F/V) converters, and they can produce pulse and square-wave outputs with a frequency range of 1 Hz to 100 kHz. For V/F conversion, the device accepts an analog input signal and generates an output pulse train whose frequency is linearly proportional to the input voltage. For F/V conversion, the device accepts any input frequency waveform and provides a linearly proportional voltage output. The complete V/F or F/V conversion requires only the addition of two capacitors, three resistors, and a reference voltage. The 9400 series consists of CMOS devices and bipolar devices that can operate on single or dual supply voltages.

16.12.1 V/F Converter

The 9400 V/F converter operates on the principle of charge balancing. The functional block diagram is shown in Fig. 16.62(a). The input voltage \( v_I \) is converted to a current \( I_{in} \) by the input resistor \( R_{in} \). This current \( I_{in} = \frac{v_I}{R_{in}} \) is then converted to a charge by the internal integrating capacitor \( C_{int} \) and gives a linearly decreasing voltage \( v_{O3} \) at the output of the op-amp integrator; that is,

\[
\frac{v_{O3}}{C_{int}} = -I_{in}t = -\frac{v_I}{R_{in}C_{int}}t \tag{16.99}
\]

As soon as voltage \( v_{O3} \) falls below the threshold level of the threshold detector, the switch closes and causes reference voltage \( V_{ref} \) to be applied to reference capacitor \( C_{ref} \) for a time long enough to charge the capacitor to reference voltage \( V_{ref} \). This action also reduces the charge on the integrating capacitor by a fixed amount \( q = C_{ref}V_{ref} \), causing the integrator output to step up by a certain amount. At the end of the charging period, \( C_{ref} \) is shorted out, dissipating the charge stored on the reference capacitor so that the system is ready to repeat the cycle when the output passes again through the zero axis.

The continued charging of the integrating capacitor \( C_{int} \) by the input voltage is balanced out by fixed charges from the reference voltage. As the input voltage is increased, the number of reference pulses required to maintain balance increases, causing the output frequency to increase also. Since each charge increment is fixed (i.e., \( q = C_{ref}V_{ref} \)), the increase in frequency with voltage is linear. The output frequency \( f_o \) is related to the input voltage by

\[
f_o = \frac{v_I}{|V_{ref}|R_{in}C_{ref}} \tag{16.100}
\]

where

- \( v_I \) = input voltage
- \( |V_{ref}| \) = reference voltage
- \( C_{ref} \) = reference capacitance

The pin diagram of the 9400 V/F converter is shown in Fig. 16.62(b), and its internal block diagram is shown in Fig. 16.62(c). The threshold detector senses the output of the integrator. The output of the detector triggers a 3-\( \mu \)s network when its input voltage passes through the threshold. The nominal threshold of
FIGURE 16.62  9400 V/F converter (Courtesy of TelCom Semiconductor, Inc.)
the detector is halfway between the power supplies, or \((V_{DD} + V_{SS})/2 \pm 400\,\text{mV}\). The output of the 3-\(\mu\)s network is applied to the output transistor \(M_1\), the divide-by-2 network, and the \(C_{\text{ref}}\) charge/discharge control circuit.

The self-start circuit ensures that the V/F converter operates properly when the power is first applied. If the integrator output is below the threshold voltage (i.e., 0 V) of the threshold detector and \(C_{\text{ref}}\) is already charged, then a positive voltage step will not occur when the power is turned on. The integrator output will continue to decrease until it crosses the \(-3.0\,-\text{V}\) threshold of the self-start comparator. When this happens, an internal resistor of 20 k\(\Omega\) is connected to the op-amp integrator input, thereby forcing the output to become positive. As soon as the op-amp output becomes positive, the self-start circuit is disabled, and the 9400 operates in its normal mode.

Pulse \(f_{\text{out}}\) (\(=f_o\)) output is an open-drain \(n\)-channel FET that provides a pulse waveform whose frequency is proportional to the input voltage \(v_i\). Pulse \(f_{\text{out}}/2\) (\(=f_o/2\)) output is an open-drain \(n\)-channel FET that provides a square wave whose frequency is one-half that of the pulse waveform. This output will change state on the rising edge of \(f_o\). Both \(f_o\) and \(f_o/2\) outputs require a pull-up resistor and interface directly with MOS, CMOS, and TTL logic circuits.

The waveforms of the V/F converter are shown in Fig. 16.63. Three microseconds after the output \(V_d\) of the detector switches to low, the output \(V_{G1}\) of the delay circuit switches from low to high. When \(V_{G1}\)
is low, transistor M₁ is off and \( f_{\text{out}} \) is high (i.e., 5 V). The divide-by-2 network is a negative-edge-triggered flip-flop whose output \( V_{G2} \) is a complement (inversion) of \( V_{G1} \). Thus, transistor M₂ will be on, and \( f_{\text{out}}/2 \) will be low. With \( V_{G1} \) low, the charge/discharge control is disabled and capacitor \( C_{\text{ref}} \) remains discharged (i.e., shorted out).

### EXAMPLE 16.18

**Designing a V/F converter** Using the 9400 as shown in Fig. 16.64, design a V/F converter so that \( f_{o} = 5 \text{ kHz} \) at \( v_{I} = 5 \text{ V} \). The input voltage \( v_{I} \) can vary between 10 mV and 10 V. Assume \( V_{DD} = -V_{SS} = 5 \text{ V} \).

**SOLUTION**

The steps used to design the V/F converter are as follows:

**Step 1.** Choose \( V_{DD} \) and \( V_{SS} \) such that

\[
4 \text{ V} \leq V_{DD} \leq 7.5 \text{ V} \quad \text{and} \quad -7.5 \text{ V} \leq V_{SS} \leq -4 \text{ V}
\]

Choose \( V_{DD} = 5 \text{ V} \) and \( V_{SS} = -5 \text{ V} \).

**Step 2.** Choose the capacitors such that \( C_{3} = C_{4} = 0.1 \mu\text{F} \). These capacitors should be close to pins 4 and 14, respectively.

**Step 3.** Choose the reference voltage \( V_{\text{ref}} = V_{SS} = -5 \text{ V} \).

**Step 4.** Choose \( R_{\text{in}} = 1 \text{ M}\Omega \).

**Step 5.** Choose \( R_{\text{bias}} = 100 \text{ k}\Omega \).

**FIGURE 16.64** TelCom 9400 converter connected as a V/F converter

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
When used as an F/V converter, the 9400 generates an output voltage that is linearly proportional to the input frequency $f_i$. The internal block diagram of the 9400 F/V converter is shown in Fig. 16.65(a). The input signal is differentiated by an $RC$ network whose output is then applied to the $(/H11001)$ input of the threshold detector (i.e., at pin 11). The threshold detector has about $200 \text{ mV}$ of hysteresis. Each time the input to the detector at pin 11 crosses zero in the negative direction, its output goes to low. Three microseconds later, the charge/discharge circuit is enabled, instantaneously connecting the reference capacitor $C_{\text{ref}}$, which remains discharged, to the reference voltage $V_{\text{ref}}$. This causes a precise amount of charge ($q = C_{\text{ref}}V_{\text{ref}}$) to be dispensed into the op-amp’s summing junction. This charge in turn flows through feedback resistor $R_{\text{int}}$ and generates a voltage pulse at the output of the op-amp. Capacitor $C_{\text{int}}$ across $R_{\text{int}}$ averages these pulses into a DC signal that is linearly proportional to the input frequency. The waveforms of the F/V converter are shown in Fig. 16.65(b). For charge $q$ dispensed to capacitor $C_{\text{int}}$ in time period $T$, we get

$$q = iT = \frac{V_O}{R_{\text{int}}} T$$

### Step 6. Choose the pull-up resistance $R_L = 10 \text{ k}\Omega$.

### Step 7. Choose $C_{\text{ref}}$ such that $C_{\text{ref}} < 500 \text{ pF}$. From Eq. (16.100),

$$C_{\text{ref}} = \frac{v_I}{|V_{\text{ref}}| R_{\text{in}} f_o} = \frac{5 \text{ V}}{5 \text{ V} \times 1 \text{ M}\Omega \times 5 \text{ kHz}} = 200 \text{ pF}$$

$C_{\text{ref}}$ should be located as close as possible to pins 3 and 5. Glass-film capacitors are recommended for high accuracy.

### Step 8. Choose $C_{\text{int}}$ such that $4C_{\text{ref}} \leq C_{\text{int}} \leq 10C_{\text{ref}}$. Assume

$$C_{\text{int}} = 5 \times C_{\text{ref}} = 5 \times 200 \text{ pF} = 1 \text{ nF}$$

$C_{\text{int}}$ should be located as close as possible to pins 3 and 12.

### Step 9. Determine the values of the offset resistors. Since $R_c = R_a \leq R_b$, assume $R_a$ is a 50-k$\Omega$ potentiometer, $R_b = 450 \text{ k}\Omega$, and $R_c = 10 \text{ k}\Omega$.

### Step 10. Determine the minimum frequency corresponding to the minimum input voltage $v_I = 10 \text{ mV}$. From Eq. (16.100),

$$f_{o(\text{min})} = \frac{10 \text{ mV}}{5 \text{ V} \times 1 \text{ M}\Omega \times 200 \text{ pF}} = 10 \text{ Hz}$$

### Step 11. Set the input voltage to the minimum value $v_I = 10 \text{ mV}$, and adjust potentiometer $R_a$ to obtain the corresponding minimum output frequency $f_{o(\text{min})} = 10 \text{ Hz}$.

### Step 12. Determine the maximum frequency corresponding to the maximum input voltage $v_I = 10 \text{ V}$. From Eq. (16.100),

$$f_{o(\text{max})} = \frac{10 \text{ V}}{5 \text{ V} \times 1 \text{ M}\Omega \times 200 \text{ pF}} = 10 \text{ kHz}$$

### Step 13. Set the input voltage to the maximum value $v_I = 10 \text{ V}$, and adjust $R_{\text{in}}$, $V_{\text{ref}}$, or $C_{\text{ref}}$ to obtain the corresponding maximum output frequency $f_{o(\text{max})} = 10 \text{ kHz}$.

### 16.12.2 F/V Converter

When used as an F/V converter, the 9400 generates an output voltage that is linearly proportional to the input frequency $f_{\text{in}}$. The internal block diagram of the 9400 F/V converter is shown in Fig. 16.65(a). The input signal is differentiated by an $RC$ network whose output is then applied to the (+) input of the threshold detector (i.e., at pin 11). The threshold detector has about $\pm 200 \text{ mV}$ of hysteresis. Each time the input to the detector at pin 11 crosses zero in the negative direction, its output goes to low. Three microseconds later, the charge/discharge circuit is enabled, instantaneously connecting the reference capacitor $C_{\text{ref}}$, which remains discharged, to the reference voltage $V_{\text{ref}}$. This causes a precise amount of charge ($q = C_{\text{ref}}V_{\text{ref}}$) to be dispensed into the op-amp’s summing junction. This charge in turn flows through feedback resistor $R_{\text{int}}$ and generates a voltage pulse at the output of the op-amp. Capacitor $C_{\text{int}}$ across $R_{\text{int}}$ averages these pulses into a DC signal that is linearly proportional to the input frequency. The waveforms of the F/V converter are shown in Fig. 16.65(b). For charge $q$ dispensed to capacitor $C_{\text{int}}$ in time period $T$, we get

$$q = iT = \frac{V_O}{R_{\text{int}}} T$$
which, for \( q = C_{\text{ref}} V_{\text{ref}} \), relates average output voltage \( V_O \) to input frequency \( f_{\text{in}} \) as follows:

\[
V_O = |V_{\text{ref}}| R_{\text{int}} C_{\text{ref}} f_{\text{in}}
\]

where

\( f_{\text{in}} \) = input frequency, in hertz

\( |V_{\text{ref}}| \) = reference voltage, in volts

\( R_{\text{int}} \) = internal integrating resistance, in ohms

\( C_{\text{ref}} \) = reference capacitance, in farads
The F/V converter will accept any input wave shape. However, the positive pulse width of the detector input (pin 11) must be at least 5 μs, and the negative pulse width must be greater than 0.5 μs. When the input frequency is less than 1 kHz, the duty cycle should be greater than 20% to ensure that $C_{\text{ref}}$ is fully charged and discharged.

The output voltage $V_O$ will have a certain amount of ripple, which is inversely proportional to $C_{\text{int}}$ and the input frequency $f_{\text{in}}$. Therefore, for low frequencies, $C_{\text{int}}$ can be increased in the range from 1 μF to 100 μF to reduce the ripple. To eliminate the ripple on $V_O$, an op-amp circuit in the common-mode configuration may be connected at the output of the F/V converter. This arrangement is shown in Fig. 16.66. Since the AC ripple content appears at both the (+) and the (−) terminals of the op-amp, the AC ripple will be canceled, and the output will have only DC voltage.

**EXAMPLE 16.19**

**Designing an F/V converter**  Using the 9400 as shown in Fig. 16.67, design an F/V converter so that $V_O = 2.5$ V at $f_{\text{in}} = 5$ kHz. The input frequency $f_{\text{in}}$ can vary between 0 and 10 kHz. Assume $V_{DD} = 5$ V, $-V_{SS} = 0$.

**SOLUTION**

The steps used to design the F/V converter are as follows.

**Step 1.** Choose $V_{DD}$ and $V_{SS}$ such that

$$4 \text{ V} \leq V_{DD} \leq 7.5 \text{ V}$$

Choose $V_{DD} = 5$ V.

**Step 2.** Choose the capacitors such that $C_4 = 0.1$ μF. These capacitors should be close to pins 4 and 14, respectively.
Step 3. Choose the reference voltage $V_{\text{ref}} = 0$ V.

Step 4. Choose $R_{\text{int}} = 1 \, \text{M}\Omega$.

Step 5. Choose $R_{\text{bias}} = 100 \, \text{k}\Omega$.

Step 6. Choose the pull-up resistance $R_L = 10 \, \text{k}\Omega$.

Step 7. Choose $C_{\text{ref}}$. From Eq. (16.101),

$$C_{\text{ref}} = \frac{V_O}{|V_{\text{ref}}| R_{\text{int}} f_{\text{in}}} = \frac{2.5 \, \text{V}}{5 \times 1 \, \text{M}\Omega \times 5 \, \text{kHz}} = 100 \, \text{pF}$$

$C_{\text{ref}}$ should be located as close as possible to pins 3 and 5. Glass-film capacitors are recommended for high accuracy.

Step 8. Choose $C_{\text{int}}$. Let

$$C_{\text{int}} = 10 \times C_{\text{ref}} = 10 \times 100 \, \text{pF} = 1 \, \text{nF}$$

$C_{\text{int}}$ should be located as close as possible to pins 3 and 12. Since the amount of ripple on the output voltage is inversely proportional to $C_{\text{int}}$ and the input frequency, $C_{\text{int}}$ can be increased to lower the ripple. Acceptable values of $C_{\text{int}}$ for low frequencies are 1 $\mu$F to 100 $\mu$F.

Step 9. Determine the values of the offset resistors. Since $R_c \leq R_a \leq R_b$, assume $R_a$ is a 50-k$\Omega$ potentiometer, $R_b = 450 \, \text{k}\Omega$, and $R_c = 10 \, \text{k}\Omega$.

Step 10. With no input signal applied ($f_{\text{in}} = 0$), adjust the potentiometer $R_a$ to obtain the minimum output voltage $V_{O(\text{min})} = 0$.

Step 11. Determine the maximum output voltage corresponding to the maximum input frequency $f_{\text{in}} = 10 \, \text{kHz}$. From Eq. (16.101),

$$V_{O(\text{max})} = 5 \times 1 \, \text{M}\Omega \times 100 \, \text{pF} \times 10 \, \text{kHz} = 5 \, \text{V}$$

Step 12. Set the input frequency to the maximum value $f_{\text{in}} = 10 \, \text{kHz}$, and adjust $C_{\text{ref}}$ so that $V_O$ is approximately 2.5 V.

**FIGURE 16.67** TelCom 9400 converter connected as an F/V converter
Sample-and-Hold (SAH) circuits are used as an interface between an analog signal and a digital circuit in a wide variety of applications such as data acquisition, analog-to-digital (A/D) conversion, and synchronous data demodulation. An SAH circuit is used to sample an analog signal at a particular instant of time and hold the value of the sample as long as required. The sampling instants and hold duration are determined by a logic control signal. The hold duration depends on the type of application. For example, in A/D conversion, samples must be held long enough for the conversion to be completed.

The principle of operation of a sample-and-hold circuit can be explained with Fig. 16.68(a), which shows a circuit consisting of capacitor C, switch S1, and internal resistor Rs. The capacitor is used to hold the sample. The switch provides a means of rapidly charging the capacitor to the sample voltage and then removing the input so that the capacitor can retain the desired voltage. When the control signal \( v_{CN} \) is high, the switch is closed. If the time constant \( R_s C \) is very small, the output voltage \( v_O \) will be very close to the...
input voltage $v_I$ and will be equal to it at the instant the control signal becomes low and the switch is opened. The idealized waveforms for output voltage $v_O$, input voltage $v_I$, and control voltage $v_{CN}$ are shown in Fig. 16.68(b).

In practice, the capacitor can neither charge instantly nor hold a constant voltage. Also, the switch cannot open and close instantaneously. As a result, the practical output waveform will differ from the ideal one. Among the important specifications given by the manufacturers of SAH circuits are aperture time, acquisition time, settling time, and drop.

Aperture time $t_{AP}$, shown in Fig. 16.69(a), is the maximum time required for the SAH circuit to open. It is the delay between application of the control signal to open the switch and the instant when the switch actually does open. This time depends on the type of switch, but typically ranges from 4 $\mu$s to 20 $\mu$s. The $t_{AP}$ of FET switches is in the range of 50 ns to 100 ns. The aperture time should be much less than the sampling period (i.e., the reciprocal of the sampling rate). Since the input signal is changing continuously, the hold voltage will change slightly during the aperture time, causing an error in the hold voltage.

Once the switch is closed for sampling, it takes a finite amount of time for the output voltage to become identical to the input signal because the input was changing during the holding interval. Acquisition time $t_{AQ}$, shown in Fig. 16.69(b), is the minimum time, after application of the sample signal, required for the output voltage to reach the input voltage (with the necessary degree of accuracy).

Settling time $t_s$ is the delay between the opening of the switch and the instant when the output reaches within a specified percentage of its final value (usually 0.99% of full-scale output). If the SAH circuit is followed by an A/D converter, conversion should not begin until the signal has settled; otherwise, the wrong signal will be converted.

Drop, or output decay rate, is the voltage drop across capacitor C during the hold time. It is inversely proportional to the capacitance, since $dv_O / dt = I / C$, where $I$ is the capacitor leakage current. This leakage current can arise as a result of biasing current in an op-amp, leakage current through the switch, or internal leakage in the capacitor.

The speed with which the output follows the input depends on the characteristics of the input signal $v_I$. $v_O$ will follow $v_I$ exponentially with time constant $R_s C$. For $v_O$ to be within 0.01% of the output, its time period should be approximately $9R_s C$. In addition, the signal source must be able to supply the charging current required by capacitor C. Usually, the analog signal is buffered from the switch by a unity-gain op-amp follower in order to ensure a low value of $R_s$. 

*FIGURE 16.69* Aperture time and acquisition time
16.13.1 SAH Op-Amp Circuits

An SAH circuit can be implemented using an op-amp and a switch, as shown in Fig. 16.70(a). When switch S1 is closed, the circuit operates as an RC filter. For a step input voltage of $V_I$, the output voltage $v_O(t)$ can be found from

$$v_O(t) = -\frac{R_F}{R_1} V_I (1 - e^{-t/R_F C}) \quad (16.102)$$

For $v_O$ to reach $V_I$ in the shortest time, the time constant $R_F C$ must be shorter than the sample interval so that the output can track the input. When switch S1 is opened, the capacitor will hold its voltage of $-V_I$. To minimize the output voltage drop, the op-amp should have a low input biasing current (as does an op-amp with a FET input stage, for example). Also, a high-quality capacitor with a low leakage current should be used.

Switch S1 can be replaced by a transistor $M_1$ (i.e., a $p$-type depletion MOSFET), as shown in Fig. 16.70(b). If the control voltage $v_{CN}$ is low (say, 0 V), the FET $M_1$ will be on (i.e., the switch will be closed), and the capacitor will be in sample mode, charging to $V_I$. If the control voltage $v_{CN}$ is high (say, +5 V), the FET will be off (i.e., the switch will be open), and the capacitor will be in hold mode.

Diode $D_1$ clamps the voltage at node A to 0.7 V. When $M_1$ is on, the diode effectively becomes connected across the FET (i.e., between its drain and source). Since the voltage drop across the FET is low, the voltage across the diode will also be small—much less than 0.7 V. Thus, the diode will have no effect during the sampling time.

16.13.2 SAH Integrated Circuits

SAH integrated circuits such as the LF198 use BiFET technology to obtain ultrahigh DC accuracy (within 0.01%) with fast signal acquisition (4 μs) and a low drop (3 mV/s). The functional block diagram of the LF198 is shown in Fig. 16.71(a), and its connection diagram is shown in Fig. 16.71(b). The manufacturers give curves showing the variation in acquisition time $t_{AQ}$ with hold capacitance $C_h$. For example, $t_{AQ} = 4 \mu s$ for hold capacitance $C_h = 1000$ pF, and $t_{AQ} = 20 \mu s$ for $C_h = 0.01$ μF.

**FIGURE 16.70** Inverting SAH op-amp circuit
16.14 Digital-to-Analog Converters

Digital systems are used in a wide variety of applications because of their efficiency, reliability, and economical operation. Applications include process and industrial control, measurement and testing, graphics and displays, data telemetry, voice and video communication, and arithmetic operations. Data processing, which has become an integral part of various systems, involves the transfer of data to and from digital devices such as microprocessors via input and output devices.

The output of digital systems is in binary form: 1s and 0s. After processing is accomplished using digital methods, the processed signal is converted back to analog form. The circuit that performs this conversion is called a digital-to-analog (D/A) converter. A D/A system normally contains four separate parts: a reference quantity; a set of binary switches to simulate binary coefficients \( B_0, \ldots, B_N \); a resistive network; and an output summing means.
16.14.1 Weighted-Resistor D/A Converter

A simple D/A converter is shown in Fig. 16.72(a). This converter can convert a 4-bit parallel digital word \((B_0B_1B_2B_3)\) to an analog voltage that is proportional to the binary number corresponding to the digital word. Four switches are used to simulate the binary inputs. (In practice, a 4-bit binary counter may be used instead.) The logic voltages, which represent the individual bits \(B_0, B_1, B_2,\) and \(B_3,\) are used to operate switches \(S_0, S_1, S_2,\) and \(S_3,\) respectively. When a \(B\) is a 1, the corresponding switch is connected to reference voltage \(V_{\text{ref}},\) when a \(B\) is a 0, the corresponding switch is grounded.

The inverting terminal of the op-amp is at virtual ground (i.e., \(V_{\text{d}} = 0\)), so the total current \(I_S\) is given by

\[
I_S = V_{\text{ref}} \left( \frac{B_3}{R_3} + \frac{B_2}{R_2} + \frac{B_1}{R_1} + \frac{B_0}{R_0} \right)
\]

Since the current flowing into the op-amp is negligible, \(I_S = I_F.\) Thus, the analog output voltage is given by

\[
V_O = -R_F I_F = -R_F V_{\text{ref}} \left( \frac{B_3}{R_3} + \frac{B_2}{R_2} + \frac{B_1}{R_1} + \frac{B_0}{R_0} \right)
\]

Resistors are weighted so that successive resistor values are related by a factor of 2 and the value of each individual resistor is inversely proportional to the numerical significance of the appropriate binary digit. That is,

\[
\text{LSB (least significant bit)} \rightarrow R_0 = \frac{R}{2^0} = R
\]

\[
R_1 = \frac{R}{2^1} = \frac{R}{2}
\]

\[
R_2 = \frac{R}{2^2} = \frac{R}{4}
\]

\[
\text{MSB (most significant bit)} \rightarrow R_3 = \frac{R}{2^3} = \frac{R}{8}
\]

**FIGURE 16.72** Weighted-resistor D/A converter
Substituting these weighted resistor values into Eq. (16.103) gives the analog output voltage $V_O$ as

$$V_O = -\frac{V_{\text{ref}}R_F}{R} \left( 2^3B_3 + 2^2B_2 + 2^1B_1 + 2^0B_0 \right)$$

(16.104)

where $B_i = 1$ if switch $S_i$ is connected to $V_{\text{ref}}$ and $B_i = 0$ if switch $S_i$ is grounded. For an input of $B_3B_2B_1B_0 = 1111$, $V_O = -15V_{\text{ref}}R_F/R$; for $B_3B_2B_1B_0 = 0110$, $V_O = -6V_{\text{ref}}R_F/R$; and for $B_3B_2B_1B_0 = 0001$, $V_O = 0$. Thus, the output $V_O$ is directly proportional to the numerical value of the binary number $B_3B_2B_1B_0$. Since there are 16 (i.e., $2^4$) combinations of the binary inputs $B_3, B_2, B_1, B_0$, the analog output will have 16 possible corresponding values. For $V_{\text{ref}} = 5\text{ V}$ and $R = 10R_F$, Eq. (16.104) gives $V_O$ as

$$V_O = -0.5 \times (2^3B_3 + 2^2B_2 + 2^1B_1 + 2^0B_0)$$

The plot is shown in Fig. 16.72(b). The major disadvantage of this D/A converter is the wide variety of resistor values required to weight the network. If the resistor values change in response to temperature changes, it will be difficult to obtain identical tracking characteristics. As a result, the accuracy and the stability of the D/A will be degraded.

### 16.14.2 R-2R Ladder Network D/A Converter

The $R-2R$ D/A ladder converter, shown in Fig. 16.73(a), has only two resistor values $R$ and $2R$, rather than a wide range of resistor values. The plot of the output (which is known as a “resistance ladder”) is shown in Fig. 16.73(b). Figure 16.73(c) exhibits the property that the equivalent resistance, looking into any of the terminals $X, Y, S_3, S_2, S_1, S_0$ with the remainder of the terminals grounded, is $3R$.

Consider the circuit with LSB $= 1$ only; that is, switch $S_0$ is closed. The equivalent circuit for LSB $= 1$ only is shown in Fig. 16.74(a). Successive Thevenin’s conversions lead, through the circuits shown in Fig. 16.74[(b) and (c)], to the final circuit shown in Fig. 16.74(d), which gives the output due to LSB $= 1$ as

$$V_O = -\frac{V_{\text{ref}}R_F}{3R} \left( \frac{B_0}{2^4} \right) \quad \text{(for LSB }= 1 \text{ only)}$$

Now consider the circuit with MSB $= 1$ only; that is, switch $S_3$ is closed. The equivalent circuit for MSB $= 1$ only is shown in Fig. 16.74(e), which can be simplified by applying the series and parallel rule for $R_s$. The simplified circuit shown gives the output due to MSB $= 1$ as

$$V_O = -\frac{V_{\text{ref}}R_F}{3R} \left( \frac{B_3}{2^4} \right) \quad \text{(for MSB }= 1 \text{ only)}$$

Thus, the output voltage is scaled up by the numerical value of the binary digit. Applying the superposition theorem, we can find the output voltage when all switches are on (i.e., switch $S_i$ is connected) as

$$V_O = -\frac{V_{\text{ref}}R_F}{3R} \left( \frac{B_3}{2^4} + \frac{B_2}{2^3} + \frac{B_1}{2^2} + \frac{B_0}{2^1} \right)$$

(16.105)

which can be simplified to

$$V_O = -\frac{V_{\text{ref}}R_F}{48R} \left( 2^3B_3 + 2^2B_2 + 2^1B_1 + 2^0B_0 \right)$$

(16.106)
where $B_i = 1$ if switch $S_i$ is connected to $V_{\text{ref}}$ and $B_i = 0$ if switch $S_i$ is grounded. For an input of $B_3B_2B_1B_0 = 1111$, $V_O = -15V_{\text{ref}}R_F/48R$; for $B_3B_2B_1B_0 = 0110$, $V_O = -6V_{\text{ref}}R_F/48R$; and for $B_3B_2B_1B_0 = 0001$, $V_O = -V_{\text{ref}}R_F/48R$. For $V_{\text{ref}} = 5 \text{ V}$ and $6R = R_F$, Eq. (16.106) gives $V_O$ as

$$V_O = \left(-\frac{5}{48}\right) \times (2^3B_3 + 2^2B_2 + 2^1B_1 + 2^0B_0)$$

whose plot is shown in Fig. 16.73(b) for $V_{O(\text{max})} = 10 \text{ V}$.

FIGURE 16.73  R-2R ladder D/A converter
Switches in IC D/A converters are made either of BJTs or of MOSFETs. They are generally one of two types: voltage driven or current driven. Voltage-driven converters, which use BJTs or MOSFETs as on or off switches, are generally used for relatively low-speed low-resolution applications. In a current-driven converter, switching is accomplished using emitter-coupled logic (ECL) current switches, which do not saturate but are driven from the active region to cutoff. This type of converter is capable of much faster operation than the voltage-driven type. IC D/A converters of 8, 10, 12, 14, and 16 bits are commercially available with either a current output, a voltage output, or both a current and a voltage output.

**FIGURE 16.74** Equivalent R-2R ladder for LSB = 1 only and MSB = 1 only

### 16.14.3 Integrated Circuit D/A Converters

Switches in IC D/A converters are made either of BJTs or of MOSFETs. They are generally one of two types: voltage driven or current driven. Voltage-driven converters, which use BJTs or MOSFETs as on or off switches, are generally used for relatively low-speed low-resolution applications. In a current-driven converter, switching is accomplished using emitter-coupled logic (ECL) current switches, which do not saturate but are driven from the active region to cutoff. This type of converter is capable of much faster operation than the voltage-driven type. IC D/A converters of 8, 10, 12, 14, and 16 bits are commercially available with either a current output, a voltage output, or both a current and a voltage output.
The MC1408 is an example of a D/A converter with current output. It is a low-cost, high-speed converter designed for use in applications where the output current is a linear product of an 8-bit digital word and an analog reference voltage. Its internal block diagram, shown in Fig. 16.75(a), consists of four parts: current switches, an R-2R ladder, a biasing current network, and a reference current amplifier. The connection diagram is shown in Fig. 16.75(b). The output current is converted to a voltage by an instantaneous current-to-voltage (I/V) op-amp converter.

The NE/SE-5018 is an example of a D/A converter with voltage output. It gives an output voltage that is a linear product of an 8-bit digital word and an analog reference voltage. Its internal block diagram is shown in Fig. 16.76(a). A typical configuration of the 5018 is shown in Fig. 16.76(b).

The manufacturer’s specifications for a D/A converter normally include the following parameters. Resolution is determined by the number of input bits of the D/A converter. An 8-bit converter has $2^8$ possible output levels, so its resolution is $1 \ 2^8 = 1 \ 256 = 0.39\%$. For a 4-bit converter, the resolution is $1 \ 2^4 = 1 \ 16 = 6.25\%$. Thus, resolution is the value of the LSB. Accuracy is defined in terms of the maximum deviation of the D/A output from an ideal straight line drawn from zero to full-scale output. Nonlinearity, or linearity error, is the difference between the actual output of the D/A converter and its ideal straight-line output. The error is normally expressed as a percentage of the full-scale range. Gain error is any error in gain, usually caused by deviations in the feedback resistor on the I/V converter. Offset error is any error caused by the fact that the output of the D/A converter is not zero when the binary inputs are all zero. This error stems from input offsets (in voltages and currents) of the op-amp as well as the D/A converter. Settling time is the time required for the output of the D/A converter to reach within $\pm 1 \ 2$ LSB of the final value for a given digital input—that is, go from zero to full-scale output. Stability is a measure of the independence of the converter parameters from variations in external conditions such as temperature and supply voltage.
FIGURE 16.76  NE/SE-5018 D/A converter with voltage output (Courtesy of Philips Semiconductors)
16.15 Analog-to-Digital Converters

A large number of physical devices generate output signals that are analog or continuous variables; examples include temperature and pressure gauges and flow transducers. For digital processing, the input signal must be converted into a binary form of 1s and 0s. The circuit that performs this conversion is called an analog-to-digital (A/D) converter. There are many types of A/D converters, depending on the type of conversion technique used, such as counting, tracking (up-down), successive approximation, single-ramp integrating, or dual-ramp integrating. The successive-approximation technique is the one most commonly used, mainly because it offers excellent trade-offs in resolution, speed, accuracy, and cost.

16.15.1 Successive-Approximation A/D Converter

A successive-approximation A/D converter operates by successively dividing in half the voltage range of the converter. The simplified block diagram of a 4-bit A/D converter is shown in Fig. 16.77(a). The converter consists of five parts: an analog comparator, a 4-bit register that has independent set and reset capability for each stage, a 4-bit D/A converter, a ring counter, and a logic control. The ring counter provides a timing (or clock) signal to control the operation of the converter. The logic control synchronizes the operation of the converter with the clock. The combination of the logic control, 4-bit register, and ring counter is often known as the successive-approximation register (SAR).

The comparator converts analog voltages to digital signals. It has two inputs, \( V_a \) and \( V_b \), and gives a binary output. If \( V_a > V_b \), the output is high (logic 1); if \( V_a < V_b \), the output is low (logic 0). Thus, the comparator output \( V_{\text{com}} \) is

\[
V_{\text{com}} = \text{sgn} \left( V_a - V_b \right) = \begin{cases} 1 & \text{for } V_a > V_b \\ 0 & \text{for } V_a < V_b \end{cases}
\]

An SAH circuit is commonly used to hold the input voltage constant during the conversion process. There is no need for an SAH circuit if the input signal varies slowly enough and has a low enough noise level that the input will not change during the conversion.

The algorithm for the operation of a successive-approximation A/D converter can be best described by an example. The steps in converting an analog voltage of, say, 10 V are as follows:

**Step 1.** The first pulse from the ring counter sets the D/A converter, 4-bit register, and ring counter so that MSB = 1 and all others are 0; that is, \( B_3 = 1 \), and \( B_2 = B_1 = B_0 = 0 \). Thus, for \( B_3B_2B_1B_0 = 1000 \), the output \( V_b \) of the D/A is 8 V, which is compared by the comparator. If \( V_a \geq 8 \) V, the MSB in the register (\( B_3 \)) is maintained at 1; otherwise, it is set to 0. At the end of step 1, \( B_3 = 1 \) for \( V_a = 10 \) V.
FIGURE 16.77 Successive-approximation A/D converter
Step 2. The second pulse from the ring counter sets \( B_2 = 1 \). \( B_1 \) and \( B_0 \) remain at 0, and \( B_3 \) remains at either 1 or 0, depending on the condition in step 1; that is, \( B_3 = B_2 = 1 \), and \( B_1 = B_0 = 0 \). Thus, for \( B_3B_2B_1B_0 = 1100 \), the output \( V_b \) of the D/A is 12 V, which is compared by the comparator. If \( V_a \geq 12 \) V, the \( B_3 \) in the register is maintained at 1; otherwise, it is set to 0. At the end of step 2, \( B_2 = 0 \) for \( V_a = 10 \) V.

Step 3. The third pulse from the ring counter sets \( B_1 = 1 \). \( B_0 \) remains at 0. \( B_2 \) and \( B_3 \) remain as they were at the end of step 2. That is, \( B_3 = 1 \), \( B_2 = 0 \), \( B_1 = 1 \), and \( B_0 = 0 \). Thus, for \( B_3B_2B_1B_0 = 1010 \), the output \( V_b \) of the D/A is 10 V, which is compared by the comparator. If \( V_a \geq 10 \) V, the \( B_1 \) in the register is maintained at 1; otherwise, it is set to 0. At the end of step 3, \( B_1 = 1 \) for \( V_a = 10 \) V.

Step 4. The fourth pulse from the ring counter sets \( B_0 = 1 \). \( B_3 \), \( B_2 \), and \( B_1 \) remain as they were at the end of step 3. That is, \( B_3 = 1 \), \( B_2 = 0 \), \( B_1 = 1 \), and \( B_0 = 1 \). Thus, for \( B_3B_2B_1B_0 = 1011 \), the output \( V_b \) of the D/A is 11 V, which is compared by the comparator. If \( V_a \geq 11 \) V, the \( B_0 \) in the register is maintained at 1; otherwise, it is set to 0. That is, \( B_0 = 0 \) for \( V_a = 10 \) V.

At the end of the fourth step, the desired number, which is in the counter, will give Read output. The results of the conversion steps are shown in Table 16.2. For an \( N \)-bit A/D converter, the conversion process will take \( N \) clock periods. That is, for an 8-bit A/D converter and a 10-MHz clock, the conversion will take \( 8 \times 10^9 \) = \( 8 \times 10^{-7} \) = 800 ns.

The successive-approximation technique can be extended to the higher-bit converter shown in Fig. 16.77(b), in which the SAR also performs the functions of logic control and ring counter. The conversion complete (CC) signal enables the latch. Digital data appear at the output of the latch and are also available serially as the SAR determines each bit. The cycle of the conversion process is normally repeated continuously, and the CC signal is connected to the Start-Conversion input.

### 16.15.2 Integrated Circuit A/D Converters

There are many types of IC A/D converters, such as the integrating A/D converter, the integrating A/D converter with three-stage outputs, and the tracking A/D converter with latched output. Also, the output can be in straight binary, binary-coded decimal (BCD), complementary binary (1s or 2s), or sign-magnitude binary form.

The NE5034 is an example of an IC A/D converter. Its internal block diagram is shown in Fig. 16.78(a). It is a high-speed microprocessor-compatible 8-bit A/D converter that uses the successive-approximation technique. It includes a comparator, a reference D/A converter, an SAR, an internal clock, and three-stage buffers all on the same chip. The connection diagram for the NE5034 is shown in Fig. 16.78(b). Upon receipt of the Start pulse, successive bits are applied to the input of the internal 8-bit current D/A converter by the

<table>
<thead>
<tr>
<th>Step</th>
<th>( V_b )</th>
<th>( B_3 )</th>
<th>( B_2 )</th>
<th>( B_1 )</th>
<th>( B_0 )</th>
<th>Comparisons</th>
<th>Answer</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>8 V</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>( V_a \approx 8 ) ( V ) ?</td>
<td>Yes</td>
</tr>
<tr>
<td>2</td>
<td>12 V</td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>( V_a \approx 12 ) ( V ) ?</td>
<td>No</td>
</tr>
<tr>
<td>3</td>
<td>10 V</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>( V_a \approx 10 ) ( V ) ?</td>
<td>Yes</td>
</tr>
<tr>
<td>4</td>
<td>11 V</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>1</td>
<td>( V_a \approx 11 ) ( V ) ?</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td>10 V</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>Read output</td>
<td></td>
</tr>
</tbody>
</table>

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
$I^2L$ SAR, beginning with the MSB (DB7). During the successive approximations, the sequence Data-Ready (DR) remains at 1. When the Output-Enable (OE) input is at logic 1, the data outputs assume a high-impedance status. When OE is at logic 0, the data are placed on the outputs. External capacitor $C_L$ sets the internal clock frequency, as shown in Fig. 16.78(c). For $C_L = 100 \text{ pF}$, for example, $f_{\text{clock}} = 120 \text{ kHz}$.

The manufacturer’s specifications for an A/D converter normally include the following parameters: *Input signal* is the maximum allowable analog input voltage range and may be unipolar or bipolar.
Conversion speed is the speed at which the A/D converter can make repetitive data conversions. The conversion time for successive-approximation converters ranges from \( \frac{1}{1000} \) s to \( \frac{1}{100} \) s, whereas for an ultrafast parallel converter the time is in the range of 10 ns to 60 ns. Quantizing error is the error inherent in the conversion process because of the finite resolution of the discrete output. It is usually \( \pm 1 \) 2 LSB. For a 10-bit converter with an analog input range of 0 to 10 V, the quantizing error will be \( 2^{10} \times 10 \, \text{V} \approx 10 \, \text{mV} \). Accuracy is the deviation of the actual bit transition value from the ideal transition value at any level over the range of the A/D converter. Accuracy includes errors from both the analog and the digital parts. With a digital error of 10 mV and a quantizing error of 10 mV, the overall error becomes 20 mV. With this amount of error, the converter will operate as a 9-bit A/D converter because a 9-bit converter has a quantizing error of \( 2^9 \times 10 \, \text{V} \approx 20 \, \text{mV} \).

### KEY POINTS OF SECTION 16.15

- An A/D converter can convert an analog signal to a digital word that is proportional to the analog signal. Although there are many conversion techniques, the successive-approximation technique is the one most commonly used, mainly because of its excellent trade-offs in resolution, speed, accuracy, and cost.
- The specifications for an A/D converter include input signal range, conversion speed, quantizing error, and accuracy.

### 16.16 Circuit Design Using Analog Integrated Circuits

There are many analog ICs for general- and special-purpose applications. They include operational amplifiers, voltage comparators, instrument amplifiers, timers, buffers, interfacing circuits, voltage/frequency converters, data conversion circuits, power conversion and control circuits, and voltage regulators. The circuit design for an application using an IC is very simple and requires the selection of external components only. The steps involved are as follows:

- **Step 1.** Identify the function(s) to be performed and the specifications, including available power supplies, range of input and output signals, and operating frequency range.
- **Step 2.** Find a suitable IC that can perform the desired function(s) and look for application examples and/or guidelines for that IC. Usually the manufacturer provides application examples and guidelines.
- **Step 3.** Determine the values of external components (usually capacitors and resistors). Generally, the manufacturer provides selection charts or curves. Unless otherwise specified, use standard values of components, with tolerances of, say, 5%.
- **Step 4.** Simulate the circuit with a simulator such as PSpice/SPICE or Electronics Workbench, if the IC is supported by the simulator.
- **Step 5.** Build and test the circuit, if possible.
Op-amp circuits with diodes can generate nonlinear functions for applications such as peak signal detectors, precision rectifiers, and comparator circuits. Op-amp circuits with diodes can reduce the effects of diode drop for low-voltage signals and are used for precision signal processing. The diode is placed in the feedback path of the op-amp and behaves as a superdiode with a negligible voltage drop on the order of microvolts.

Various waveforms are often required in electronic and control circuits. There are many integrated circuits that can be used to generate these waveforms. The design of wave generators is very simple and requires the selection of the external circuit components only. ICs such as op-amps, comparators, the NE/SE-566 VCO, the 555 timer, the NE/SE-565 PLL, and the 9400 series converters can be used to generate various waveforms.

References


Review Questions

1. What is a voltage limiter?
2. What is a soft limiter?
3. What is a clamper?
4. What is a comparator?
5. What is a limiting comparator?
6. What are the advantages of precision rectifiers?
7. What is a superdiode?
8. What are the differences between a comparator and an op-amp?
9. What is the principle of operation of a zero-crossing detector?
10. What is a Schmitt trigger?
11. What circuit parameters determine the upper and lower threshold voltages of a Schmitt trigger?
12. What are the effects of hysteresis on output voltage?
13. What is the principle of operation of a square-wave generator?
14. Why is a square-wave generator called an astable multivibrator?
15. What is saturation of an op-amp?
16. What is the purpose of series resistors in the input terminals of an op-amp in a square-wave generator?
17. What circuit parameters determine the frequency of a square wave?
18. What is the principle of operation of a triangular-wave generator?
19. What circuit parameters determine the frequency of a triangular wave?
20. What is the principle of operation of a sawtooth-wave generator?
21. What circuit parameters determine the frequency of a sawtooth wave?
22. What is the duty cycle of a sawtooth wave?
23. What is a voltage-controlled oscillator (VCO)?
24. What is the principle of operation of a VCO?
25. What circuit parameters determine the output frequency of a VCO?
26. What is the 555 timer?
27. What is a monostable multivibrator?
28. What advantages does the 555 timer connected as an astable multivibrator have over an op-amp astable multivibrator?
29. What circuit parameters determine the output frequency of the 555 timer connected as ramp generator?
30. What is a phase-lock loop (PLL)?
31. What are the main components of a PLL?
32. What is the principle of operation of a PLL?
33. What is the free-running mode of a PLL?
34. What is the capture mode of a PLL?
35. What is the phase-lock mode of a PLL?
36. What is the capture range of a PLL?
37. What is the lock range of a PLL?
38. What are the relationships among the free-running frequency, capture frequency, and lock frequency of a PLL?
39. What are some applications of the TelCom 9400 series converter?
40. What is a sample-and-hold circuit?
41. What are the main parts of a sample-and-hold circuit?
42. What is a digital-to-analog (D/A) converter?
43. What are the main parts of a D/A converter?
44. What is an analog-to-digital (A/D) converter?
45. What are the main parts of an A/D converter?

Problems

The symbol D indicates that a problem is a design problem. The symbol P indicates that you can check the solution to a problem using PSpice/SPICE or Electronics Workbench.
16.2 Circuits with Op-Amps and Diodes

16.1 a. The precision full-wave rectifier in Fig. 16.6(a) has \( R_1 = R_2 = R_F = R = 10 \, \text{k}\Omega, \ R_3 = 40 \, \text{k}\Omega, \) and \( R_4 = 40 \, \text{k}\Omega. \) The input voltage is \( v_S = 2 \sin (377\pi t) \). Plot the transfer characteristic and draw the waveform of the output voltage \( v_O \).

b. Use PSpice/SPICE to plot the transfer characteristic and the output voltage.

d. Design a precision full-wave rectifier as shown in Fig. 16.6(a) to provide a voltage gain of \( A_f = v_O\, v_S = 50. \)

The input voltage is \( v_S = 0.01 \sin (200\pi t) \).

b. Use PSpice/SPICE to check your design by plotting the output voltage.

16.2 a. Design a negative voltage limiter using the circuit in Fig. 16.10(a) by determining the values of \( V_{CC}, R_1, R_2, R_3, \) and \( R_F. \) The supply DC voltage is \( V_A = 15 \, \text{V}. \) The circuit should limit the negative output voltage to \( V_{O(min)} = -8 \, \text{V}. \) The voltage gain without limiting is \( A_f = -5. \) The diode is fully turned on at a forward current of \( i_D = 0.1 \, \text{mA}, \) and its corresponding forward voltage drop is \( V_D = 0.7 \, \text{V}. \) The slope after the break point is to be limited to \( S_1 = -1 \, \text{mV/V}. \)

b. Use PSpice/SPICE to plot the transfer characteristic.

16.3 a. A hard limiter as shown in Fig. 16.15(a) by determining the values of \( V_{CC}, R_1, R_2, \) and \( R_F. \) The supply DC voltage is \( V_A = 15 \, \text{V}. \) The on-state diode current is \( D = 0.1 \, \text{mA}, \) and its corresponding forward voltage drop is \( V_D = 0.7 \, \text{V}. \) The magnitude of the slopes after the break points should be less than or equal to \( 1 \, \text{V/mV}. \)

b. Use PSpice/SPICE to plot the transfer characteristic.

16.4 The adjustable limiter in Fig. 16.11(a) has \( R_1 = 12 \, \text{k}\Omega, R_2 = 70 \, \text{k}\Omega, \ R_2 = 8 \, \text{k}\Omega, \ R_3 = 1 \, \text{k}\Omega, \) and \( R_4 = 6 \, \text{k}\Omega. \) Assume \( V_A = 15 \, \text{V}. \) The diode drop is \( v_D = 0.7 \, \text{V}. \) Determine \( V_{CC}, R_1, R_2, \) and \( R_F. \) The adjustable limiter in Fig. 16.11(a) to satisfy the following specifications:

- The positive clamping voltage \( V_{O(max)} = 5 \, \text{V}, \) and the corresponding input voltage \( V_{S(max)} = 3 \, \text{V}. \)
- The negative clamping voltage \( V_{O(min)} = 5 \, \text{V}, \) and the corresponding input voltage \( V_{S(min)} = 0 \, \text{V}. \)
- The output voltage when the input voltage is \( v_S = 5 \, \text{V}. \)

b. Use PSpice/SPICE to plot the transfer characteristic for part (a).

16.5 a. Design a negative voltage limiter as shown in Fig. 16.11(a) to satisfy the following specifications:

- The positive clamping voltage \( V_{O(max)} = 5 \, \text{V}, \) and the corresponding input voltage \( V_{S(max)} = 3 \, \text{V}. \)
- The negative clamping voltage \( V_{O(min)} = 5 \, \text{V}, \) and the corresponding input voltage \( V_{S(min)} = 0 \, \text{V}. \)
- The output voltage when the input voltage is \( v_S = 5 \, \text{V}. \)

b. Use PSpice/SPICE to plot the transfer characteristic. Assume \( V_{CC} = 12 \, \text{V} \) and \( V_{EE} = -12 \, \text{V}. \) Use the PSpice/SPICE op-amp macromodel.

16.6 Design an output voltage–clamping circuit as shown in Fig. 16.14(a) so that the slope of the transfer characteristic is \( S = v_O, \) \( v_S = 20, \) \( V_{O(max)} = 6.7 \, \text{V}, \) and \( V_{O(min)} = -8.7 \, \text{V}. \) Determine the zener voltages \( V_{Z1} \) and \( V_{Z2}. \) Assume \( V_D = 0.7 \, \text{V}. \)

16.7 a. Design a hard limiter as shown in Fig. 16.15(a) by determining the values of \( R_1, R_2, R_3, R_4, \) and \( R_5. \) The circuit should limit the negative output voltage to \( V_{O(min)} = -6 \, \text{V}, \) and the positive voltage to \( V_{O(max)} = 6 \, \text{V}. \) The magnitude of the slopes after the break points should be less than or equal to \( 1 \, \text{V/mV}. \)

b. Use PSpice/SPICE to plot the transfer characteristic. Assume \( V_{CC} = 15 \, \text{V}, -V_{EE} = -15 \, \text{V}, \) and \( v_S = -5 \, \text{V} \) to \( 5 \, \text{V}. \) Use the PSpice/SPICE op-amp macromodel.

16.8 a. Design a hard limiter as shown in Fig. 16.15(a) by determining the values of \( R_1, R_2, R_3, R_4, \) and \( R_5. \) The circuit should limit the negative output voltage to \( V_{O(min)} = -5 \, \text{V}, \) and the positive voltage to \( V_{O(max)} = 5 \, \text{V}. \) The magnitude of the slopes after the break points should be less than or equal to \( 1 \, \text{V/mV}. \)

b. Use PSpice/SPICE to plot the transfer characteristic. Assume \( V_{CC} = 15 \, \text{V}, -V_{EE} = -15 \, \text{V}, \) and \( v_S = -5 \, \text{V} \) to \( 5 \, \text{V}. \) Use the PSpice/SPICE op-amp macromodel.

16.9 a. Design a hard limiter as shown in Fig. 16.15(a) by determining the values of \( R_1, R_2, R_3, R_4, \) and \( R_5. \) The circuit should limit the negative output voltage to \( V_{O(min)} = -7 \, \text{V}, \) and the positive voltage to \( V_{O(max)} = 9 \, \text{V}. \) The magnitude of the slopes after the break points should be less than or equal to \( 1 \, \text{V/mV}. \)

b. Use PSpice/SPICE to plot the transfer characteristic. Assume \( V_{CC} = 15 \, \text{V}, -V_{EE} = -15 \, \text{V}, \) and \( v_S = -5 \, \text{V} \) to \( 5 \, \text{V}. \) Use the PSpice/SPICE op-amp macromodel.
16.3 & 16.5 Comparators and Schmitt Triggers

For Probs. 16.10 through 16.14, use comparator LM111 and \( v_S = 10 \sin(2000\pi t) \) to plot the hysteresis characteristic using PSpice/SPICE.

16.10 Design a Schmitt trigger as in Fig. 16.28(a) so that \( V_{th} = |+V_{th}| = |-V_{th}| = 5 \text{ V}. \) Assume \( V_{sat} = |-V_{sat}| = 12 \text{ V}. \)

16.11 The parameters of the Schmitt trigger in Fig. 16.25(a) are \( R_1 = 100 \Omega \) and \( R_P = 47 \text{ k}\Omega. \) Calculate the threshold voltages \( +V_{th} \) and \( -V_{th}. \) Assume \( V_{sat} = |-V_{sat}| = 12 \text{ V}. \)

16.12 Design an inverting Schmitt trigger with the reference voltage of Fig. 16.25(a) so that \( V_{HT} = -8 \text{ V} \) and \( V_{LT} = -4 \text{ V}. \) Assume \( V_{sat} = |-V_{sat}| = 12 \text{ V}. \)

16.13 Design a noninverting Schmitt trigger as in Fig. 16.28(a) so that \( V_{th} = |+V_{th}| = |-V_{th}| = 5 \text{ V}. \) Assume \( V_{sat} = |-V_{sat}| = 12 \text{ V}. \)

16.14 Design a noninverting Schmitt trigger with the reference voltage of Fig. 16.29(a) so that \( V_{HT} = 8 \text{ V} \) and \( V_{LT} = 4 \text{ V}. \) Assume \( V_{sat} = |-V_{sat}| = 12 \text{ V}. \)

16.6–16.8 Square-, Triangular-, and Sawtooth-Wave Generators

For Probs. 16.15 through 16.19, use op-amp LF411 to plot the output using PSpice/SPICE.

16.15 Design the square-wave generator shown in Fig. 16.34(a) so that \( f_0 = 2 \text{ kHz}. \) Assume \( V_{sat} = |-V_{sat}| = 10 \text{ V}. \)

16.16 The parameters of the square-wave generator of Fig. 16.34(a) are \( R_1 = 10 \text{ k}\Omega, R_F = 15 \text{ k}\Omega, R = 10 \text{ k}\Omega, \) and \( C = 0.047 \text{ \mu F}. \) Calculate the output frequency \( f_0. \)

16.17 Design the triangular-wave generator shown in Fig. 16.37(a) so that \( f_0 = 2 \text{ kHz} \) and \( V_{th} = 5 \text{ V}. \) Assume \( V_{sat} = |-V_{sat}| = 12 \text{ V}. \)

16.18 The parameters of the triangular-wave generator of Fig. 16.37(a) are \( R_1 = 10 \text{ k}\Omega, R_F = 40 \text{ k}\Omega, R = 10 \text{ k}\Omega, \) and \( C = 0.047 \text{ \mu F}. \) Calculate the output frequency \( f_0. \)

16.19 Design the sawtooth-wave generator shown in Fig. 16.40(a) so that \( f_0 = 5 \text{ kHz}, V_{th} = 5 \text{ V}, \) and the circuit has a duty cycle of \( k = t_1 \) \( T = 0.4. \) Assume \( V_{sat} = |-V_{sat}| = 12 \text{ V}. \)

16.9 Voltage-Controlled Oscillators

16.20 a. Design a VCO as shown in Fig. 16.43(c) that has a nominal frequency of \( f_0 = 10 \text{ kHz}. \) Assume \( V_{CC} = 15 \text{ V}. \)

b. Calculate the modulation in the output frequencies if \( v_{CN} \) is varied by \( \pm 10\%. \)

16.21 The parameters of the VCO in Fig. 16.43(c) are \( R_A = 2.5 \text{ k}\Omega, R_1 = R_B = 10 \text{ k}\Omega, \) and \( C = 0.01 \text{ \mu F}. \)

a. Calculate the nominal frequency of the output waveform \( f_0. \)

b. Calculate the modulation in the output frequencies if \( v_{CN} \) is varied by \( \pm 10\%. \) Assume \( V_{CC} = 12 \text{ V}. \)

16.10 The 555 Timer

For Probs. 16.22 through 16.29, use the 555 timer to plot the output by PSpice/SPICE.
16.22 Design a monostable multivibrator as in Fig. 16.45(a) so that $t_p = 2$ ms. Assume $V_{CC} = 15$ V.

16.23 Design an astable multivibrator as in Fig. 16.49(a) so that $k = 80\%$ and $f_0 = 5$ kHz. Assume $V_{CC} = 15$ V.

16.24 The parameters of the astable multivibrator in Fig. 16.49(a) are $R_A = 2.2$ k$\Omega$, $R_B = 3.9$ k$\Omega$, and $C = 0.1$ $\mu$F. Determine (a) the charging time $t_c$, (b) the discharging time $t_d$, and (c) the free-running frequency $f_0$.

16.25 Design a square-wave generator as in Fig. 16.52 so that $k = 50\%$ and $f_0 = 5$ kHz. Assume $V_{CC} = 15$ V.

16.26 The parameters of the square-wave generator in Fig. 16.52 are $R_A = 2.7$ k$\Omega$, $R_B = 4.7$ k$\Omega$, and $C = 1$ $\mu$F. Determine (a) the charging time $t_c$, (b) the discharging time $t_d$, and (c) the free-running frequency $f_0$.

16.27 Design a ramp generator as in Fig. 16.53(a) so that $k = 50\%$ and $f_0 = 5$ kHz. Assume $V_{CC} = 15$ V, $V_{BE} = 0.7$ V, and a transistor of $\beta_F = 100$.

16.28 The parameters of the ramp generator in Fig. 16.53(a) are $R = 10$ k$\Omega$, $V_{CC} = 15$ V, $V_{BE} = 0.7$ V, and a transistor of $\beta_F = 100$. Determine the free-running frequency $f_0$.

16.29 Design the FSK modulator shown in Fig. 16.54(a) to produce frequencies of 1270 Hz and 1570 Hz corresponding to 1 (mark) and 0 (space), respectively.

16.11 Phase-Lock Loops

16.30 Design a PLL as shown in Fig. 16.58(c) so that $f_0 = 5$ kHz and $f_c = \pm 50$ Hz. Assume $V_{CC} = -V_{EE} = 15$ V.

16.31 The parameters of the PLL in Fig. 16.58(c) are $R_1 = 12$ k$\Omega$, $C_1 = 0.01$ $\mu$F, $C_2 = 10$ $\mu$F, and $V_{CC} = -V_{EE} = 15$ V. Determine (a) the free-running frequency $f_0$, (b) the lock frequency $f_L$, and (c) the capture range $f_c$.

16.12 Voltage-to-Frequency and Frequency-to-Voltage Converters

16.32 Design a V/F converter as shown in Fig. 16.64 so that $f_0 = 2.5$ kHz at $v_1 = 5$ V. The input voltage $v_1$ can vary between 10 mV and 10 V. Assume $V_{DD} = -V_{SS} = 5$ V.

16.33 Design an F/V converter as shown in Fig. 16.66 so that $V_O = 2.5$ V at $f_{in} = 10$ kHz. The input frequency $f_{in}$ can vary between 0 and 20 kHz. Assume $V_{DD} = -V_{SS} = 5$ V.

16.13 Sample-and-Hold Circuits

16.34 Design the SAH circuit shown in Fig. 16.71(a) so that the drop is within 0.5%. The leakage current in the hold mode is 1 nA, and the hold voltage is $V_h = 5$ V. The internal hold time is $t_h = 100$ $\mu$s. Find the holding capacitance $C_h$.

16.35 Design the SAH circuit shown in Fig. 16.71(a) so that the output tracks the input within 0.5%. The internal hold time is $t_h = 100$ ns. The input biasing current of the op-amp is $I_B = 10$ nA. Assume $R_F = R_1 = 20$ k$\Omega$ and $v_1 = 5$ V.

16.36 Design the SAH circuit shown in Fig. 16.71(a) so that the drop is within 0.5%. The internal hold time is $t_h = 0.1$ ms, and the hold voltage is $V_h = 5$ V. The input biasing current of the op-amp is $I_B = 200$ nA. Assume $R_F = R_1 = 20$ k$\Omega$. 
16.14 Digital-to-Analog Converters

16.37 A 10-bit D/A converter of type $2R$ ladder, as shown in Fig. 16.73(a), has an input of 00 1001 1001, and the reference voltage is 5 V. Find the analog output voltage $V_O$.

16.38 A D/A converter is to have a full-scale output of 5 V and a resolution of less than 20 mV. What is the bit size?

16.39 The 8-bit D/A converter shown in Fig. 16.73(a) is to have a full-scale output of 10 V with a reference voltage of 5 V. Find the values of $R_F$ and $R$.

16.15 Analog-to-Digital Converters

16.40 An 8-bit A/D converter has a reference voltage of 10 V. Find (a) the analog input corresponding to the binary outputs 1010 1010 and 0101 0101, (b) the binary output if $V_a = 3$ V, and (c) the resolution of the converter.

16.41 Construct a table similar to Table 16.2 for a 4-bit A/D converter if $V_a = 5$ V with a reference voltage of 16 V.

16.42 Construct a table similar to Table 16.2 for an 8-bit A/D converter if $V_a = 4$ V with a reference voltage of 10 V.
A.1 Introduction

This appendix provides an introduction to the electronic circuit simulation software OrCAD [1, 2]. The objectives of this appendix are the following:

- Becoming familiar with the electronic circuit simulation software OrCAD Capture
- Learning how to draw electrical circuits by OrCAD Capture

After installation instructions presented in Sec. A.2, Sec. A.3 provides an overview of the software package. Then Secs. A.4 to A.8 cover the basic steps in the circuit analysis process:

- Drawing the circuit
- Selecting the type of analysis
- Simulating the circuit
- Displaying the results of the simulation

The remaining sections, Secs. A.9 to A.13, deal with specific operations:

- Copying and capturing schematics
- Varying parameters
- Performing frequency response analysis
- Modeling devices and elements
- Creating netlists
- Adding library files

A.2 Installing the Software

The instructions for installing the OrCAD software are printed on the OrCAD installation disk or the CD-ROM. The software can be downloaded from the Cadence website at http://www.cadence.com/products/orcad/Pages/downloads.aspx

The following steps, which pertain to OrCAD Version 9.2, are applicable to other versions as well:

1. Place the schematics CD-ROM in the CD drive.
2. From Windows, enter the File Manager and click the left mouse button on the CD drive.
3. Click on setup.exe, File, Run, and OK.
4. Click on OK to select the products to install Capture and PSpice, as shown in Fig. A.1(a).
5. Click on OK to select the default C:OrcadLite.
6. Click on Yes to choose Program Folder: Orcad Family Release 9.2 Lite Edition, as shown in Fig. A.1(b).
7. Create the OrCAD Capture Lite and PSpice A/D icons from the Start menu; then go to Programs, Orcad Family, and Capture Lite menu. These icons are shown in Fig. A.2.
8. Click the left mouse button on the Capture Lite icon once, and the window of Capture Lite will open.

**NOTE** Although there are later versions of OrCAD, the 9.2 version has proven to be bug free and works well.
Appendix A  Introduction to OrCAD

(a) Selecting products

(b) Selecting the program folder

**FIGURE A.1**  Installation setup for OrCAD Capture
The OrCAD Capture software package has three major interactive programs: Capture, PSpice A/D, and Probe. Capture is a powerful program that lets us build circuits by drawing them within a window on the monitor. PSpice A/D lets us analyze the circuit created by Capture and generate voltage and current solutions. Probe is a graphic postprocessor and lets us display plots of voltages, currents, impedance, and power.

The general layout of Capture is shown in Fig. A.3. The top menu shows 10 main choices. The right-side menu shows the schematic “drawing” menu for selecting and placing parts. File, Edit, View, Place, and PSpice are most frequently used. For any help, click on the Help menu.
The mouse follows an object–action sequence. First select an object and then perform an action, as in the following:

1. Click the mouse on a menu title so that it stays open. Then click on the command that you want.
2. A single click on the left mouse button selects an item.
3. A double click on the left mouse button performs an action such as to end a mode or edit a selection.
4. To drag a selected item, click the left button, hold it down, and move the mouse. Release the left button when placed.
5. To end a mode or edit a selection, click the right mouse button once.
6. To repeat an action, double click the right button.

These mouse operations are summarized in Table A.1.

### Table A.1  PSpice mouse operations

<table>
<thead>
<tr>
<th>Button</th>
<th>Action</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Left</td>
<td>Single click</td>
<td>Select an item</td>
</tr>
<tr>
<td></td>
<td>Double click</td>
<td>End a mode</td>
</tr>
<tr>
<td></td>
<td>Double click on selected object</td>
<td>Edit a selection</td>
</tr>
<tr>
<td></td>
<td>Single click on selected object and hold</td>
<td>Drag a selection</td>
</tr>
<tr>
<td></td>
<td>Shift + single click</td>
<td>Extend a selection</td>
</tr>
<tr>
<td>Right</td>
<td>Single click</td>
<td>Abort the mode</td>
</tr>
<tr>
<td></td>
<td>Double click</td>
<td>Repeat an action</td>
</tr>
</tbody>
</table>

The mouse follows an object–action sequence. First select an object and then perform an action, as in the following:

1. Click the mouse on a menu title so that it stays open. Then click on the command that you want.
2. A single click on the left mouse button selects an item.
3. A double click on the left mouse button performs an action such as to end a mode or edit a selection.
4. To drag a selected item, click the left button, hold it down, and move the mouse. Release the left button when placed.
5. To end a mode or edit a selection, click the right mouse button once.
6. To repeat an action, double click the right button.

These mouse operations are summarized in Table A.1.

### A.4 The Circuit Analysis Process

As an example, let us draw and analyze the pulse response of an $RLC$ circuit as shown in Fig. A.4. The steps to draw and analyze a circuit are as follows:

1. Draw the circuit under Capture.
2. Select the mode of analysis under PSpice.
3. Simulate the circuit under PSpice A/D.
4. Display the results under Probe.

![FIGURE A.4  An RLC circuit](image.png)
A.5 Drawing the Circuit

The steps in drawing a circuit are as follows:

**Step 1.** Begin a new project.
**Step 2.** Get components from the Get New Part menu and place them on the drawing board.
**Step 3.** Rotate components as desired.
**Step 4.** Wire the components together.
**Step 5.** Label the components and add text as desired.
**Step 6.** Set attributes of the components.
**Step 7.** View the schematic.
**Step 8.** Save the circuit.

A.5.1 Beginning a New Project

Let us begin a new project to draw and analyze; call it Example B-3. Open the File menu, choose **New**, and then select **Project** as shown in Fig. A.5.

The New Project menu opens as shown in Fig. A.6. Give the file name of the new project (e.g., Example B-3), select **Analog or Mixed A/D**, and give the location of this new file (e.g., C:\Rashid\PT3). Next select **Create a blank project**, as shown in Fig. A.7.

![File menu for a new project](image-url)
A.5.2 Getting and Placing Components

Let us start by placing a pulse source, a resistor, an inductor, a capacitor, and a ground on the drawing board. From the capture editor, you can place parts from the component libraries onto your schematic.

1. Use the Part command in the Place menu shown in Fig. A.8 or from the Place part menu on the right side. Alternately, choose the **Place part** command from the Draw menu on the right side. Choose **Browse** to browse the list of libraries or use the search command to enter the name of a known part, resistor R, as shown in Fig. A.9.

2. Choose **OK** or double click after selecting the part. The chosen part becomes the “current part” and is ready to be placed on your schematic.
Appendix A  
Introduction to OrCAD

**FIGURE A.8** Place menu

**FIGURE A.9** Place part menu
3. The cursor is replaced by the shape of the part.
4. Click left to place an instance of the part; double click to place the part and end the mode or click right to end the mode without placing the part. To drag, point to an object, press and hold the left mouse button, and move the mouse. When the object is where you want it, release the mouse button.
5. To move a component, point to it, press and hold the left mouse button, and drag it to a new location.
6. To remove a component, select it, and choose Delete from the Edit menu.
7. Place a pulse source (VPULSE) from the source.slb library, a resistor (R), an inductor (L), and a capacitor (C) from the analog.slb library and a ground symbol (GND) from the Place menu or Place Ground (zero 0) menu on the right-side menu as shown in Fig. A.10. Arrange them as shown in Fig. A.11.

A.5.3 Rotating Components

Now rotate the capacitor so that it can be wired neatly into the circuit. Each time that you rotate a component, it turns clockwise 90°. The Flip command flips a selected object(s) to produce a mirror image of the object.

1. To rotate the capacitor (or other component), select it and choose Rotate from the Edit menu as shown in Fig. A.12. If you select an area of your schematic, the area is rotated around the center of the selection box. This is done as shown in Fig. A.13.
2. To flip a component, select it and choose Flip from the Edit menu.
If the Rotate command is dimmed, the capacitor is not selected. Try again by pointing to it so that the pointer becomes a hand and then click the left mouse button.

3. To deselect the selected capacitor (or other selected component), click it with the right mouse button or click an empty spot with the left mouse button.

4. To drag or rotate two or more components at once, first select them by drawing a rectangle around the components and then drag or rotate the rectangle.

To draw a rectangle around components, point above and beside one of the components that you want to select. Press and hold the left mouse button and drag diagonally until the rest of the components are in the rectangle that appears.

5. To deselect one of the selected components, click it with the right mouse button. To deselect everything on the marked rectangle, click an empty spot with the left mouse button.
A.5.4 Wiring Components

Once components are placed on the drawing board, you need to connect them. You can use the schematic editor to draw wires on your schematic and/or make vertices.

1. To draw a wire, choose Wire from the Place menu or Wire menu on the right side.
2. The cursor changes to a “wire” mode and is displayed as crosshairs. Click left to start drawing.
3. Move the mouse in any direction to extend the wire.
4. Click left again to end the wiring of a part. Click left once to start and click left again to stop. Continue until wiring of all parts is completed.
5. Press the ESC key or double click left or click right to end the wiring mode (the cursor will change from crosshairs to normal mode).

Now wire the circuit as shown in Fig. A.14.

**TIP** To bring back the last command used (for wire), double click the right mouse button. In the lower right-hand corner, you will see the Wire command after the word Cmd.

A.5.5 Labeling Components and Adding Text

We can place a label on selected wires, bus segments, or ports. Wire and bus segments or ports may display multiple labels; however, all labels for a segment will contain the same text. Each component in a circuit can be labeled. We will assign levels R, L, C, $V_S$, and $V_O$.

1. To edit a label, select a wire, bus segment, or port for labeling.
2. Double click on the label to bring up the dialog box for Display Properties.
3. Enter the text for the label.
4. Click OK.
5. Change all levels R, L, C, $V_S$, and $V_O$ as shown in Fig. A.15.

**TIP** To move the level, left click on it and move to a desired location.

6. Next click on Text in the Place menu (see Fig. A.8).

You can place text anywhere on your schematic and size it to suit your needs.

1. To add text to your schematic, choose Text from the Place menu or Place text on the right-side menu.
2. In the dialog box, type in the desired text.
3. To change the font size, modify the Font Size shown in the dialog box to suit your needs.
4. Click OK.
5. Move the text to the desired location on the schematic and click left to place the text. Click right to end the mode.

Now place the output voltage, $V_O$ as shown in Fig. A.16.
A.5.6 Setting Attributes

An attribute of a schematic item consists of a name and value pair.

1. To edit the properties of a selected object(s), select the object(s) to edit.
2. Choose Properties from the Edit menu or double click on the attribute text to bring up the Display Properties dialog box directly as shown in Fig. A.17.
3. Select an individual attribute: A dialog box appears in which you can enter a new value for the attribute. You can also select an entire part: A dialog box appears showing all attributes that can be edited for that part—for example, Value Only.
4. Set $R = 2 \Omega$, $L = 50 \mu H$, and $C = 10 \mu F$.

**TIP** A quick way to set a component’s value is to double click it.

**HELP** Changes made to attributes in the schematic editor occur only on the particular part instance. Changes do not affect the underlying symbol in the library.

5. To change the value of an attribute, select it. The name and value should appear in the properties of the Edit menu.

   a. Change the value in the **Value** edit field and click **OK**.
   b. To delete an attribute, select it from the list and press **Delete**.
   c. To change whether the attribute name and/or value is shown on the schematic, select it and click **OK**.
   d. To add a new attribute, type the name and value in the **Edit** fields and press **OK**. Set the pulse source $V_S$ as shown in Fig. A.18. Click on the attributes and then type $V_1 = 0$, $V_2 = 1$ V, $TD = 0$, $TR = 1$ ns, $TF = 1$ ns, $PW = 0.5$ ms, $PER = 1$ ms.

6. Click **OK** to incorporate the change.

**TIP** Protected attributes, marked with an asterisk (*), cannot be changed using the Schematic Editor.
A.5.7 Viewing the Schematic

You can change the viewing scale on your schematic via the View menu, shown in Fig. A.19. Table A.2 explains the functions of commands in the View menu.

A.5.8 Saving the Circuit File

Your schematic files are automatically saved into the current directory, unless otherwise specified.

- To save your schematic file, choose **Save** from the File menu, as shown in Fig. A.20, to save changes to the current file. You do not need to type a file name extension.

A circuit name must be a valid DOS file name. All schematic files are automatically given the file name extension OPJ. For example, the file name FIGB_3 will be saved as FIG B_3.OPJ.

**TABLE A.2** View menu commands

<table>
<thead>
<tr>
<th>Command</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fit</td>
<td>Resets the viewing scale so that all parts, wires, and text can be seen on the screen.</td>
</tr>
<tr>
<td>In</td>
<td>Allows you to view an area on the schematic at closer range (i.e., magnify). After you select this command, a crosshair appears on the screen. Move the crosshair to the area you want to zoom in on.</td>
</tr>
<tr>
<td>Out</td>
<td>Changes the viewing scale so that you can view the schematic from a greater distance (i.e., view more of the schematic on the screen). After you select this command, a crosshair appears on the screen. Move the crosshair to define the center of the viewing area.</td>
</tr>
<tr>
<td>Area</td>
<td>Allows you to select a rectangular area on the schematic to be expanded to fill the screen. If you already have a selection box on your screen when you choose <strong>Area</strong>, the contents of the selection box will be expanded. If not, drag the mouse to form a selection box around the portion of the schematic you want to expand. The items within the selection box will be expanded to fill the screen.</td>
</tr>
<tr>
<td>Entire Page</td>
<td>Allows you to view the entire schematic page at once.</td>
</tr>
</tbody>
</table>
PSpice allows DC sweep analysis, AC (frequency response) sweep analysis, and transient analysis. The Setup command specifies which types of simulation analyses are enabled and allows the user to set up the parameters for selected analyses (described next).

When a signal is first applied to a circuit, there is a short-lived transient state before the circuit settles down to its usual responses. For the sample RLC circuit, we will conduct a transient analysis to examine the charging and discharging voltage of the capacitor.

1. Choose New Simulation Profile for a new simulation file or Edit Simulation Profile for an existing file from the PSpice menu, shown in Fig. A.21, and the Analysis Setup dialog box will open.
2. Enable transient analysis by clicking once in the Enabled space so that a check appears in the space, as shown in Fig. A.22.
3. Click on **Transient** to open the Transient specifications dialog box.
4. Type the print particulars, such as a **Print values** of 10 ns and a **Run to time** of 0.5 ms, as shown in Fig. A.23.
5. Click **OK**.
A.7 Simulation with PSpice

Now we are ready to simulate the circuit with PSpice. To begin the simulation, left click on **Simulate** in the Analysis menu.

During the circuit simulation process, PSpice creates and accesses a number of files. The first file is the *Schematics* file (*SCH*), generated when a circuit drawn on the screen is saved. When the Schematics file is analyzed, three new files are generated: the *Circuit* file (*CIR*), the *Netlist* file (*NET*), and the *Alias* file (*ALS*). The Circuit file (the master file) contains the *simulation directives* and references to the Netlist, Alias, and Model files. The Netlist file contains a Kirchhoff-like set of equations that lists parts and how they are connected; the equations relate the voltages and currents to the circuit elements through node numbers. The Alias file lists alternative names for circuit nodes. The Model file lists the characteristics and model statement of each component.

**Tip** If there is any problem such as a missing attribute name or value, PSpice will indicate the error, and the simulation will be aborted. You can find the error message in the output file. For example, a message about an error in the file FIG1.1 will appear in FIG1_1.OUT.

When PSpice is run, each simulation directive in the Circuit (master) file specifies the information to be sent to the Output and Data files.

- The *Output* file (*OUT*) is an ASCII file that holds the “audit trail” for the simulation. It contains a wide variety of information, including the original netlist, all output variables, and various tables.
- The *Data* file (*DAT*) is sent to Probe, which uses the binary information to generate plots and graphs within the Probe window.

While a simulation is running, you can check the status of the simulation, as shown in Fig. A.24. When the simulation is completed, PSpice will display the message “Transient Analysis finished.”

**Tip** *Time step* is the internal simulation step for convergence to give a specified accuracy, whereas *Print step* (which was specified in the Transient specifications dialog box) is for printing or plotting the output variables.

![FIGURE A.24](image-url) PSpice analysis status box
A.8 Displaying the Results of a Simulation

Probe is a graphics postprocessor that allows the simulation results to be displayed in graphical form. After the calculations are completed, assuming no errors are found, PSpice sends the database it generated (FIG1_1.DAT) to Probe, which displays a graph.

1. To use Probe, choose Run Probe from the selecting Analysis type in Fig. A.22 menu (or use the F12 function key). As shown in Fig. A.25, Probe opens with an initial (default) graph in which the x-axis is automatically set to the transient variable, time.

   To run Probe automatically after a simulation, choose Automatically Run Probe After Simulation from Probe Setup in the Analysis menu.

2. Choose Trace on the Probe menu. Within the large box at the top of the dialog box that appears, you will see the list of default trace variables that you can choose from, as shown in Fig. A.26. Before moving to the third step, specify the plot variables(s). For this example, we choose the following variables:
   a. V(R:2): V specifies the variable type (voltage, which is referenced to the ground), R specifies a component (resistor), and 2 specifies one end (node 2) of the component. A 1 indicates what was the left-hand side and a 2 what was the right-hand side when the component was initially placed horizontally. After one counterclockwise rotation, node 2 would be at the top.
   b. V(Vs : +): This is the voltage at the positive terminal of the voltage source Vs.
   c. I(R): This is the current flowing through resistance R from terminal 1 (left) to terminal 2 (right).

FIGURE A.25  Probe menu
3. After specifying the plot variables, type \( V(C:2) \) in the trace command box. This instructs Probe to display a graph of the voltage at terminal 2 of capacitor C, which is our desired output voltage. Click **OK** and you will see the plot of the output voltage, \( V(C:2) \), shown in Fig. A.27.

4. While copying to the clipboard and pasting to another document, the background and foreground colors can be changed as shown in the dialog box in Fig. A.28.
A.9 Copying and Capturing Schematics

You can use the Copy to Clipboard command from the Edit menu to copy one or more items from your schematic into another Windows program. With the mouse, select a rectangular area on the schematic to copy. Choose **Copy to Clipboard** from the Edit menu. Open the Windows program into which you want to paste the item(s). Use the Paste command from the newly opened Windows program to paste the item into the new file.

The Cut command deletes items from your schematic and places them in the Paste buffer. The Copy command copies items from your schematic to the Paste buffer. The Paste command places the contents of the Paste buffer on the schematic.

To cut or copy one or more items on your schematic, first select the item(s) on your schematic to be cut or copied. Then choose **Cut** or **Copy** from the Edit menu. To paste a cut or copied item onto your schematic, choose **Paste** from the Edit menu. Then place the cursor on the schematic where you’d like the cut or copied item to appear, and left click to place the item; right click to end the mode.

A.10 Varying Parameters

PSpice allows variation of component values or device parameters. We will plot the output voltage of our sample **RLC** circuit for three values: \( R = 1 \, \Omega \), 2 \( \Omega \), and 10 \( \Omega \).

1. To begin, get the part **PARAM** from the special.slb library file.
2. To change the value of \( R \) to a variable name such as **RVAL**, double click on the value of \( R \) and type \{RVAL\}.
3. To change the attribute of the part **PARAM**, double click on **PARAM** to open the Attributes dialog box, shown in Fig. A.29.
4. Choose **NAME1=**RVAL and **VALUE1=**2 and click each time on **Save Attr**.
5. Choose **Setup** from the Analysis menu; then click once in the enabled space to enable parametric analysis.

6. Click on **Parametric** to open the box shown in Fig. A.30.

7. Enter or enable the Parametric specifications as follows: For Sweep variable, choose **Global parameter**; for Parameter name, type in **RVAL**; for Sweep Type, choose **Value list**; and for the values, type in 1 2 10.

▶ **TIP** You can define up to three variables in the same PARAM.

---

**FIGURE A.29** Dialog box for PARAM

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>RV AL</td>
<td>10 k</td>
</tr>
<tr>
<td>CV AL</td>
<td>10 µF</td>
</tr>
<tr>
<td>LV AL</td>
<td>50 µH</td>
</tr>
</tbody>
</table>

(a) Symbol  
(b) Parameters

**FIGURE A.30** Parametric specifications
8. Run the simulation for the complete circuit, shown in Fig. A.31. Use Probe to plot the output variable, \( V(C:2) \), as shown in Fig. A.32.

**TIP** The labels \( R_1 \), \( R_2 \), and \( R_{10} \) in Fig. A.32 were typed by choosing (in sequence) Tools, Label, and Text from the Probe menu. You can copy the Probe plot(s) to other Windows programs by choosing (in sequence) Tools and Copy to Clipboard from the Probe menu.

## A.11 Frequency Response Analysis

As an example of frequency response analysis, we will plot the output voltage and phase angle of the example RLC circuit for three values: \( R = 1 \, \Omega \), \( 2 \, \Omega \), and \( 10 \, \Omega \).

1. To begin, change the attributes of the source \( V_S \), shown in Fig. A.17, to 1 V AC by selecting the part and changing the attributes in the Part Name dialog box. The complete circuit is shown in Fig. A.33.

2. Choose Setup from the Analysis menu; then enable AC Sweep in the dialog box that appears (see Fig. A.22).

3. Click on AC Sweep to open the box shown in Fig. A.34.
4. Enter or enable the **AC Sweep** specifications as follows: For **AC Sweep Type**, choose **Decade**; for **Start Frequency**, type in 100; for **End Frequency**, type in 100k; for **Points/Decade**, type in 101.

5. Run the simulation for the complete circuit, shown in Fig. A.33. Use **Probe** to plot the output voltage, \( V(C:2) \), and the phase of the output voltage, \( VP(C:2) \), as shown in Fig. A.35.

**TIP** If you add DB, as in \( VDB(C:2) \), you will get the output voltage in decibels.

![Figure A.34](image.png)  
**FIGURE A.34** Dialog box for frequency response analysis

![Figure A.35](image.png)  
**FIGURE A.35** Frequency response for \( R = 1 \ \Omega, 2 \ \Omega, \) and \( 10 \ \Omega \)
A.12 Modeling Devices and Elements

A model that specifies a set of parameters for an element can be generated in PSpice using the .MODEL command. The same model can be used by one or more elements in the same circuit. The general form of the model statement [1] is as follows:

```
.MODEL MNAME TYPE (P1=A1 P2=A2 + P3=A3 ... PN=AN [<tolerance specifications>])
```

MNAME is the name of the model and must start with a letter. Although it is not required, it is advisable to use the symbol of the element as the first letter (e.g., R for resistor, L for inductor). P1, P2, . . . , PN are the element parameters, and A1, A2, . . . , AN are their values. TYPE is the type name of the element and must be one of the types shown in Table A.3. An element must have the correct type name; that is, a resistor must have the type name RES, not IND or CAP. However, a circuit with several model names can include more than one model of the same type.

Tolerance specifications are used with .MC or .WORSE analysis only. They may be appended to each parameter using the following format:

```
[DEV/<distribution name> <value in % from 0 to 9>]
[LOT/<distribution name> <value in % from 0 to 9>]
```

where <distribution name> is either UNIFORM (in which case uniformly distributed deviations are generated over the range of ±<value>) or GAUSS (in which case deviations with Gaussian distribution are generated over the range ±4 and <value> specifies the ±1 deviation).

Following are some sample model statements:

```
.MODEL RLOAD RES (R=1 TC1=0.02 TC2=0.005)
.MODEL RLOAD RES (R=1 DEV/GAUSS 0.5% LOT/UNIFORM 10%)
.MODEL CPASS CAP (C=1 VC1=0.01 VC2=0.002 TC1=0.02 TC2=0.005)
```

**TABLE A.3** Type names of elements

<table>
<thead>
<tr>
<th>Type Name</th>
<th>Element</th>
</tr>
</thead>
<tbody>
<tr>
<td>RES</td>
<td>Resistor</td>
</tr>
<tr>
<td>CAP</td>
<td>Capacitor</td>
</tr>
<tr>
<td>D</td>
<td>Diode</td>
</tr>
<tr>
<td>IND</td>
<td>Inductor</td>
</tr>
<tr>
<td>NPN</td>
<td>npn bipolar junction transistor</td>
</tr>
<tr>
<td>PNP</td>
<td>pnp bipolar junction transistor</td>
</tr>
<tr>
<td>NJF</td>
<td>n-channel junction FET</td>
</tr>
<tr>
<td>PJF</td>
<td>p-channel junction FET</td>
</tr>
<tr>
<td>NMOS</td>
<td>n-channel MOSFET</td>
</tr>
<tr>
<td>PMOS</td>
<td>p-channel MOSFET</td>
</tr>
<tr>
<td>GASFET</td>
<td>n-channel GaAs MESFET</td>
</tr>
<tr>
<td>VSWITCH</td>
<td>Voltage-controlled switch</td>
</tr>
<tr>
<td>ISWITCH</td>
<td>Current-controlled switch</td>
</tr>
<tr>
<td>CORE</td>
<td>Nonlinear magnetic core (transformer)</td>
</tr>
</tbody>
</table>
A.12.1 Resistors

The symbol for a resistor is R. The name of a resistor must start with R, and the model statement takes the following general form:

\[ R<\text{name}> \ N^+ \ N^- \ R\text{NAME} \ R\text{VALUE} \]

A resistor does not have a polarity, and so the order of the nodes does not matter. However, the current is assumed to flow from the node designated by \( N^+ \) as the positive node through the resistor to the node designated by \( N^- \) as the negative node. \( R\text{NAME} \) is the model name that defines the parameters of the resistor. \( R\text{VALUE} \) is the nominal value of the resistance.

**NOTE** Some versions of PSpice and SPICE do not make this assumption and thus do not allow you to refer to currents through a resistor. For example, such versions will not allow you to use a notation such as \( I(RL) \) to indicate the current through RL.

The model parameters for resistors are shown in Table A.4. If \( R\text{NAME} \) is omitted, \( R\text{VALUE} \) is the resistance in ohms; \( R\text{VALUE} \) can be positive or negative but must not be zero. If \( R\text{NAME} \) is included and \( TCE \) is not specified, the resistance as a function of temperature is calculated from

\[ R\text{ES} = R\text{VALUE} \times R \times [1 + TC1 \times (T - T0) + TC2 \times (T - T0)^2] \]

If \( R\text{NAME} \) is included and \( TCE \) is specified, the resistance as a function of temperature is calculated from

\[ R\text{ES} = R\text{VALUE} \times R \times 1.01^{TCE \times (T - T0)} \]

where \( T \) and \( T0 \) are the operating temperature and the room temperature, respectively, in degrees Celsius.

Following are some sample resistor statements:

\begin{verbatim}
RL 5 6 5K
RLOAD 10 13 ARES 1MEG
.MODEL RMOD RES (R=1 TC1=0.02 TC2=0.005)
RINPUT 13 17 RRES 2K
.MODEL ARESRES (R=1 TCE=1.5)
\end{verbatim}

**TABLE A.4** Model parameters for resistors

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
<th>Units</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>R</td>
<td>Resistance multiplier</td>
<td></td>
<td>1</td>
</tr>
<tr>
<td>TC1</td>
<td>Linear temperature coefficient</td>
<td>°C⁻¹</td>
<td>0</td>
</tr>
<tr>
<td>TC2</td>
<td>Quadratic temperature coefficient</td>
<td>°C⁻²</td>
<td>0</td>
</tr>
<tr>
<td>TCE</td>
<td>Exponential temperature coefficient</td>
<td>%/°C</td>
<td>0</td>
</tr>
</tbody>
</table>
A.12.2 Capacitors

The symbol for a capacitor is C. The name of a capacitor must start with C, and the model statement takes the following general form:

\[ C<\text{name}> \ N^+ \ N^- \ \text{CNAME} \ \text{CVALUE} \ \text{IC=VO} \]

\( N^+ \) is the positive node, and \( N^- \) is the negative node. The voltage of node \( N^+ \) is assumed to be positive with respect to node \( N^- \), and the current flows from node \( N^+ \) through the capacitor to node \( N^- \). \text{CNAME} is the model name, and \text{CVALUE} is the nominal value of the capacitor. \text{IC} defines the initial (time zero) voltage of the capacitor, VO.

The model parameters for capacitors are shown in Table A.5. If \text{CNAME} is omitted, \text{CVALUE} is the capacitance in farads; \text{CVALUE} can be positive or negative but must not be zero. If \text{CNAME} is included, the capacitance as a function of voltage and temperature is calculated from

\[ \text{CAP} = \text{CVALUE} \times C \times (1 + VC1 \times V + VC2 \times V^2) \times [1 + TC1 \times (T - T0) + TC2 \times (T - T0)^2] \]

where \( T \) and \( T0 \) are the operating temperature and the room temperature, respectively, in degrees Celsius.

Following are some sample capacitor statements:

- \text{C1} 2 6 0.01UF
- \text{LOAD} 10 13 10PF IC=1.5V
- \text{INPUT} 14 16 DCAP 5PF
- \text{CX} 10 25 DCAP 10NF IC=3.5V
- .MODEL DCAP CAP (C=1 VC1=0.01 VC2=0.002 TC1=0.02 TC2=0.005)

\[ \text{TIP} \quad \text{The initial conditions (if any) apply only if you specify the UIC (use initial condition) option under the .TRAN command.} \]

A.12.3 Inductors

The symbol for an inductor is L. The name of an inductor must start with L, and the model statement takes the following general form:

\[ L<\text{name}> \ N^+ \ N^- \ \text{LNAME} \ \text{LVALUE} \ \text{IC=IO} \]

\( N^+ \) is the positive node, and \( N^- \) is the negative node. The voltage of \( N^+ \) is assumed to be positive with respect to node \( N^- \), and the current flows from node \( N^+ \) through the inductor to node \( N^- \). \text{LNAME} is the model name, and \text{LVALUE} is the nominal value of the inductor. \text{IC} defines the initial (time zero) current of the inductor \text{IO}.

\[ \text{TIP} \quad \text{The initial conditions (if any) apply only if you specify the UIC (use initial condition) option under the .TRAN command.} \]
The model parameters for inductors are shown in Table A.6. If LNAME is omitted, LVALUE is the inductance in henries; LVALUE can be positive or negative but must not be zero. If LNAME is included, the inductance as a function of current and temperature is calculated from

\[
\text{IND} = \text{LVALUE} \times L \times (1 + \text{IL1} \times I + \text{IL2} \times I^2) \times [1 + \text{TC1} \times (T - T0) + \text{TC2} \times (T - T0)^2]
\]

where T and T0 are the operating temperature and the room temperature, respectively, in degrees Celsius.

Following are some sample inductor statements:

- LE 3 5 5MH
- LLOAD 10 14 2UH IC=0.1MA
- LLINE 12 14 LMOD 2MH
- LCHOKE 15 29 LMOD 5UH IC=0.4A
- .MODEL LMODIND (L=1 IL1=0.1 IL2=0.002 TC1=0.02 TC2=0.005)

**TIP** The initial conditions (if any) apply only if you specify the UIC (use initial condition) option using the .TRAN command.

### A.12.4 Diodes

The model statement for a diode has the following general form:

\[
\text{.MODEL DNAME D (P1=A1 P2=A2 P3=A3 ... PN=AN)}
\]

DNAME is the name of the model and can begin with any character, but its size is normally limited to eight characters. D is the type symbol for diodes. P1, P2, ..., PN are the model parameters, and A1, A2, ..., AN are their values. The model parameters are listed in Table A.7.

### A.12.5 Bipolar Transistors

The model statement for \textit{npn} transistors has the following general form:

\[
\text{.MODEL QNAME NPN (P1=A1 P2=A2 P3=A3 ... PN=AN)}
\]

The general form of the model statement for \textit{pnp} transistors is

\[
\text{.MODEL QNAME PNP (P1=A1 P2=A2 P3=A3 ... PN=AN)}
\]

QNAME is the name of the BJT model, and NPN and PNP are the type symbols for \textit{npn} and \textit{pnp} transistors, respectively. QNAME can begin with any character, but its size is normally limited to eight characters.
TABLE A.7 Model parameters for diodes

<table>
<thead>
<tr>
<th>Name</th>
<th>Model Parameter</th>
<th>Units</th>
<th>Default</th>
<th>Typical</th>
</tr>
</thead>
<tbody>
<tr>
<td>IS</td>
<td>Saturation current ($I_S$)</td>
<td>$A$</td>
<td>$1E-14$</td>
<td>$1E-14$</td>
</tr>
<tr>
<td>RS</td>
<td>Parasitic resistance ($R_S$)</td>
<td>$\Omega$</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>N</td>
<td>Emission coefficient ($n$)</td>
<td></td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>TT</td>
<td>Transit time</td>
<td>$s$</td>
<td>0</td>
<td>0.1 ns</td>
</tr>
<tr>
<td>CJO</td>
<td>Zero-bias $pn$ capacitance ($C_{JO}$)</td>
<td>$F$</td>
<td>0</td>
<td>2 pF</td>
</tr>
<tr>
<td>VJ</td>
<td>Junction potential ($V_J$)</td>
<td>$V$</td>
<td>1</td>
<td>0.6</td>
</tr>
<tr>
<td>M</td>
<td>Junction grading coefficient</td>
<td></td>
<td>0.5</td>
<td>0.5</td>
</tr>
<tr>
<td>EG</td>
<td>Activation energy</td>
<td>$eV$</td>
<td>1.11</td>
<td>1.11</td>
</tr>
<tr>
<td>XTI</td>
<td>IS temperature exponent</td>
<td></td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>KF</td>
<td>Flicker noise coefficient</td>
<td></td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>AF</td>
<td>Flicker noise exponent</td>
<td></td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>FC</td>
<td>Forward-bias depletion capacitance coefficient</td>
<td>0.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BV</td>
<td>Reverse breakdown voltage</td>
<td>$V$</td>
<td>$\infty$</td>
<td>50</td>
</tr>
<tr>
<td>IBV</td>
<td>Reverse breakdown current</td>
<td>$A$</td>
<td>$1E-10$</td>
<td></td>
</tr>
</tbody>
</table>

P1, P2, …, PN are the parameters, and A1, A2, …, AN are their values. Table A.8 shows the model parameters for BJTs.

### A.12.6 JFETs

The model statement for an $n$-channel JFET has the following general form:

```
.MODEL JNAME NJF (P1=A1 P2=A2 P3=A3 ... PN=AN)
```

The general form of the model statement for a $p$-channel JFET is

```
.MODEL JNAME PJF (P1=A1 P2=A2 P3=A3 ... PN=AN)
```

JNAME is the name of the model, and NJF and PJF are the type symbols for $n$-channel and $p$-channel JFETs, respectively. P1, P2, …, PN are the parameters, and A1, A2, …, AN are their values. Table A.9 lists the model parameters for JFETs.

### A.12.7 MOSFETs

The symbol for a metal oxide semiconductor field-effect transistor (MOSFET) is M. The name of a MOSFET must start with M, and the model statement takes the following general form:

```
M<name> ND NG NS NB MNAME + [L=<value>] [W=<value>]
```

where ND, NG, NS, and NB are the drain, gate, source, and bulk (or substrate) nodes, respectively. MNAME is the model name. The positive current is the current flowing into a terminal; that is, the current flows from the drain node through the device to the source node in an $n$-channel MOSFET.
### TABLE A.8  Model parameters for BJTs

<table>
<thead>
<tr>
<th>Name</th>
<th>Model Parameter</th>
<th>Units</th>
<th>Default</th>
<th>Typical</th>
</tr>
</thead>
<tbody>
<tr>
<td>IS</td>
<td>$pn$ saturation current</td>
<td>A</td>
<td>1E-16</td>
<td>1E-16</td>
</tr>
<tr>
<td>BF</td>
<td>Ideal maximum forward beta</td>
<td></td>
<td>100</td>
<td>100</td>
</tr>
<tr>
<td>NF</td>
<td>Forward current emission coefficient</td>
<td></td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>VAF(VA)</td>
<td>Forward Early voltage</td>
<td>V</td>
<td>$\approx$</td>
<td>100</td>
</tr>
<tr>
<td>IKF(IK)</td>
<td>Corner for forward beta high-current roll-off</td>
<td>A</td>
<td>$\approx$</td>
<td>10 MA</td>
</tr>
<tr>
<td>NE</td>
<td>Base–emitter leakage emission coefficient</td>
<td></td>
<td>1.5</td>
<td>2</td>
</tr>
<tr>
<td>BR</td>
<td>Ideal maximum reverse beta</td>
<td></td>
<td>1</td>
<td>0.1</td>
</tr>
<tr>
<td>NR</td>
<td>Reverse current emission coefficient</td>
<td></td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>IKR</td>
<td>Corner for reverse beta high-current roll-off</td>
<td>A</td>
<td>$\approx$</td>
<td>100 MA</td>
</tr>
<tr>
<td>RB</td>
<td>Zero-bias (maximum) base resistance (base spreading resistance)</td>
<td>$\Omega$</td>
<td>0</td>
<td>100</td>
</tr>
<tr>
<td>RE</td>
<td>Emitter ohmic resistance</td>
<td>$\Omega$</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>RC</td>
<td>Collector ohmic resistance (collector lead and bulk resistance)</td>
<td>$\Omega$</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>CJE</td>
<td>Base–emitter zero-bias $pn$ capacitance ($C_{je0}$)</td>
<td>F</td>
<td>0</td>
<td>2 pF</td>
</tr>
<tr>
<td>VJE(PE)</td>
<td>Base–emitter built-in potential ($V_{jbe}$)</td>
<td>V</td>
<td>0.75</td>
<td>0.7</td>
</tr>
<tr>
<td>MJE(ME)</td>
<td>Base–emitter $pn$ grading factor</td>
<td></td>
<td>0.33</td>
<td>0.33</td>
</tr>
<tr>
<td>CJC</td>
<td>Base–collector zero-bias $pn$ capacitance ($C_{jco}$)</td>
<td>F</td>
<td>0</td>
<td>1 pF</td>
</tr>
<tr>
<td>VJC(PC)</td>
<td>Base–collector built-in potential ($V_{jco}$)</td>
<td>V</td>
<td>0.75</td>
<td>0.5</td>
</tr>
<tr>
<td>MJC(MC)</td>
<td>Base–collector $pn$ grading factor</td>
<td></td>
<td>0.33</td>
<td>0.33</td>
</tr>
<tr>
<td>CIS(CCS)</td>
<td>Collector–substrate zero-bias $pn$ capacitance</td>
<td>F</td>
<td>0</td>
<td>2 pF</td>
</tr>
<tr>
<td>MJS(MS)</td>
<td>Collector–substrate $pn$ grading factor</td>
<td></td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>FC</td>
<td>Forward-bias depletion capacitor coefficient</td>
<td></td>
<td>0.5</td>
<td></td>
</tr>
<tr>
<td>TF</td>
<td>Ideal forward transit time</td>
<td>s</td>
<td>0</td>
<td>0.1 ns</td>
</tr>
<tr>
<td>TR</td>
<td>Ideal reverse transit time</td>
<td>s</td>
<td>0</td>
<td>10 ns</td>
</tr>
<tr>
<td>EG</td>
<td>Bandgap voltage (barrier height)</td>
<td>eV</td>
<td>1.11</td>
<td>1.11</td>
</tr>
<tr>
<td>XTI(PT)</td>
<td>IS temperature effect exponent (temperature coefficient for IS)</td>
<td></td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>KF</td>
<td>Flicker noise coefficient</td>
<td></td>
<td>0</td>
<td>6.6E-16</td>
</tr>
<tr>
<td>AF</td>
<td>Flicker noise exponent</td>
<td></td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

### TABLE A.9  Model parameters for JFETs

<table>
<thead>
<tr>
<th>Name</th>
<th>Model Parameter</th>
<th>Units</th>
<th>Default</th>
<th>Typical</th>
</tr>
</thead>
<tbody>
<tr>
<td>VTO</td>
<td>Threshold voltage</td>
<td>V</td>
<td>$-2$</td>
<td>$-2$</td>
</tr>
<tr>
<td>BETA</td>
<td>Transconductance coefficient</td>
<td>$A/V^2$</td>
<td>1E-4</td>
<td>1E-3</td>
</tr>
<tr>
<td>LAMBDA</td>
<td>Channel-length modulation</td>
<td>$V^{-1}$</td>
<td>0</td>
<td>1E-4</td>
</tr>
<tr>
<td>RD</td>
<td>Drain ohmic resistance</td>
<td>$\Omega$</td>
<td>0</td>
<td>100</td>
</tr>
<tr>
<td>RS</td>
<td>Source ohmic resistance</td>
<td>$\Omega$</td>
<td>0</td>
<td>100</td>
</tr>
<tr>
<td>IS</td>
<td>Gate $pn$ saturation current</td>
<td>A</td>
<td>1E-14</td>
<td>1E-14</td>
</tr>
<tr>
<td>PB</td>
<td>Gate $pn$ potential</td>
<td>V</td>
<td>0.6</td>
<td></td>
</tr>
<tr>
<td>CGD</td>
<td>Gate–drain zero-bias $pn$ capacitance ($C_{gdo}$)</td>
<td>F</td>
<td>5 pF</td>
<td></td>
</tr>
<tr>
<td>CGS</td>
<td>Gate–source zero-bias $pn$ capacitance ($C_{gso}$)</td>
<td>F</td>
<td>1 pF</td>
<td></td>
</tr>
<tr>
<td>FC</td>
<td>Forward-bias depletion capacitance coefficient</td>
<td>0.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>KF</td>
<td>Flicker noise coefficient</td>
<td>0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>AF</td>
<td>Flicker noise exponent</td>
<td>1</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
The model statement for an \( n \)-channel MOSFET has the following general form:

\[ \text{.MODEL } \text{MNAME } \text{NMOS} (P1=A1 \text{ P2=A2 } P3=A3 \ldots PN=AN) \]

The model statement for a \( p \)-channel MOSFET is

\[ \text{.MODEL } \text{MNAME } \text{PMOS} (P1=A1 \text{ P2=A2 } P3=A3 \ldots PN=AN) \]

NMOS and PMOS are the type symbols for \( n \)-channel and \( p \)-channel MOSFETs, respectively. MNAME can begin with any character, but its size is normally limited to eight characters. P1, P2, \ldots, PN are the parameters, and A1, A2, \ldots, AN are their values. Table A.10 lists the model parameters for MOSFETs.

## A.13 Creating Netlists

Once you have drawn a schematic, you can create a netlist to use with other PSpice/SPICE software applications. After drawing your schematic, choose **Create Netlist** from the Analysis menu, shown in Fig. A.36. The netlist for Figs. A.31 and A.33 follows. It can be found in the file EXA-25.NET.

### Table A.10 Model parameters for MOSFETs

<table>
<thead>
<tr>
<th>Name</th>
<th>Model Parameter</th>
<th>Units</th>
<th>Default</th>
<th>Typical</th>
</tr>
</thead>
<tbody>
<tr>
<td>LEVEL</td>
<td>Model type (Shichman-Hodges)</td>
<td></td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>L</td>
<td>Channel length</td>
<td>m</td>
<td>DEFL</td>
<td></td>
</tr>
<tr>
<td>W</td>
<td>Channel width</td>
<td>m</td>
<td>DEFW</td>
<td></td>
</tr>
<tr>
<td>VTO</td>
<td>Zero-bias threshold voltage</td>
<td>V</td>
<td>0</td>
<td>0.1</td>
</tr>
<tr>
<td>KP</td>
<td>Transconductance</td>
<td>A/V²</td>
<td>2E-5</td>
<td>2.5E-5</td>
</tr>
<tr>
<td>GAMMA</td>
<td>Bulk threshold parameter</td>
<td>V¹/²</td>
<td>0</td>
<td>0.35</td>
</tr>
<tr>
<td>LAMBDA</td>
<td>Channel-length modulation (LEVEL=1)</td>
<td>V⁻¹</td>
<td>0</td>
<td>0.02</td>
</tr>
<tr>
<td>RD</td>
<td>Drain ohmic resistance</td>
<td>Ω</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>RS</td>
<td>Source ohmic resistance</td>
<td>Ω</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>IS</td>
<td>Bulk ( pn ) saturation current</td>
<td>A</td>
<td>1E-14</td>
<td>1E-15</td>
</tr>
<tr>
<td>JS</td>
<td>Bulk ( pn ) saturation current/area</td>
<td>A/m²</td>
<td>0</td>
<td>1E-8</td>
</tr>
<tr>
<td>PB</td>
<td>Bulk ( pn ) potential</td>
<td>V</td>
<td>0.8</td>
<td>0.75</td>
</tr>
<tr>
<td>CBD</td>
<td>Bulk-drain zero-bias ( pn ) capacitance</td>
<td>F</td>
<td>0</td>
<td>5 pF</td>
</tr>
<tr>
<td>CBS</td>
<td>Bulk-source zero-bias ( pn ) capacitance</td>
<td>F</td>
<td>0</td>
<td>2 pF</td>
</tr>
<tr>
<td>CJ</td>
<td>Bulk ( pn ) zero-bias bottom capacitance/length</td>
<td>F/m²</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>MJ</td>
<td>Bulk ( pn ) bottom grading coefficient</td>
<td></td>
<td>0.5</td>
<td></td>
</tr>
<tr>
<td>FC</td>
<td>Bulk ( pn ) forward-bias capacitance coefficient</td>
<td>F</td>
<td>0.5</td>
<td></td>
</tr>
<tr>
<td>CGSO</td>
<td>Gate–source overlap capacitance/channel width</td>
<td>F/m</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>CGDO</td>
<td>Gate–drain overlap capacitance/channel width</td>
<td>F/m</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>CGBO</td>
<td>Gate-bulk overlap capacitance/channel length</td>
<td>F/m</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>NSUB</td>
<td>Substrate doping density</td>
<td>1/cm³</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>TOX</td>
<td>Oxide thickness</td>
<td>m</td>
<td>( \infty )</td>
<td>600</td>
</tr>
<tr>
<td>UO</td>
<td>Surface mobility</td>
<td>cm²/V · s</td>
<td></td>
<td>1E-26</td>
</tr>
<tr>
<td>KF</td>
<td>Flicker noise coefficient</td>
<td></td>
<td>0</td>
<td>1E-26</td>
</tr>
<tr>
<td>AF</td>
<td>Flicker noise exponent</td>
<td></td>
<td>1</td>
<td>1.2</td>
</tr>
</tbody>
</table>
* Schematics Netlist

C_C  0 $N_0001  10uF  ; C is connected between nodes 0 and $N_0001
L_L  $N_0002 $N_0001  50uH ; L is connected between nodes $N_0002 and $N_0001
R_R  $N_0003 $N_0002 {RVAL} ; R is connected between nodes $N_0003 and $N_0002
V_Vs $N_0003 0  AC  1V ; Vs is connected between nodes $N_0003 and 0
+PULSE 0 1V 0 1ns 1ns 0.5ms 1ms ; Pulse voltage specifications

**TIP**  When you open a schematic file (e.g., EX5-1.SCH) for the first time, PSpice may indicate an error with the message “Not Finding Netlist.” If this happens, first create the netlist by choosing Create Netlist from the Analysis menu, shown in Fig. A.36.

If the commands for transient analysis and AC analysis are included, the netlist for the PSpice circuit file becomes as follows:

```
Frequency Response ; The first line is the title line; PSpice always ignores this statement

* Circuit Description
C_C  0 $N_0001  10uF  ; C is connected between nodes 0 and $N_0001
L_L  $N_0002 $N_0001  50uH ; L is connected between nodes $N_0002 and $N_0001
R_R  $N_0003 $N_0002 {RVAL} ; R is connected between nodes $N_0003 and $N_0002

* Source Descriptions for both ac and pulse sources
V_Vs $N_0003 0  AC  1V ; Vs is connected between nodes $N_0003 and 0
+PULSE 0 1V 0 1ns 1ns 0.5ms 1ms ; Pulse voltage specifications

* Analysis Descriptions for both ac and pulse sources
.AC  DEC 101 100HZ 100KHZ ; ac analysis from 100 Hz to 100 kHz with decade increments of 101 points per decade
.TRAN 10ns 0.5ms ; Transient analysis from 0 to 0.5 ms with 10 ns printing/plotting interval
.PROBE ; Graphics post-processor
.END ; This is the last line and must always be included
```
A.14 Adding Library Files

The directory of the PSpice circuit simulator (default name C:\MSIMEV61) has a subdirectory LIB, which contains library files of schematics and device models. This library has only a limited number of transistor schematics and models. If you want to use devices or elements with different model parameters, use elements or devices from the symbol library BREAKOUT.SLB, as shown in Fig. A.37. Use Notepad to modify the model statement in the BREAKOUT.LIB file—for example,

\[\text{C:\OrcadLite\Capture\library\PSpice/Breakout.Lib}\]

You can either use the library file RASHID.LIB that comes with the book or create your own library file containing the model statement of a device with the same model name as in the PSpice schematic—for example,

\[\text{.MODEL Q2N2222 Q (BF=100)} ; \text{BJT Q2N2222}\]
\[\text{.MODEL D1N4148 D (IS=10E-15 BV=100)} ; \text{Diode D1N4148}\]
This way you keep the same model name and use the PSpice schematic symbols, but change the model parameters.

**TABLE A.12  PSpice/SPICE commands**

<table>
<thead>
<tr>
<th>Analysis or Function</th>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>Absolute value (operator)</td>
<td>ABS</td>
</tr>
<tr>
<td>AC/frequency analysis</td>
<td>.AC</td>
</tr>
<tr>
<td>DC operating point</td>
<td>.OP</td>
</tr>
<tr>
<td>DC sweep</td>
<td>.DC</td>
</tr>
<tr>
<td>Difference (operator)</td>
<td>DIF</td>
</tr>
<tr>
<td>End of subcircuit</td>
<td>.ENDS</td>
</tr>
<tr>
<td>Fourier analysis</td>
<td>.FOUR</td>
</tr>
<tr>
<td>Frequency response transfer function</td>
<td>FREQ</td>
</tr>
<tr>
<td>Function definition</td>
<td>.FUNC</td>
</tr>
<tr>
<td>Gain limit (operator)</td>
<td>GLIMIT</td>
</tr>
<tr>
<td>Global nodes</td>
<td>.GLOBAL</td>
</tr>
<tr>
<td>Graphics postprocessor</td>
<td>.PROBE</td>
</tr>
<tr>
<td>Include file</td>
<td>.INC</td>
</tr>
<tr>
<td>Initial conditions</td>
<td>.IC</td>
</tr>
<tr>
<td>Library file</td>
<td>.LIB</td>
</tr>
<tr>
<td>Model definition</td>
<td>.MODEL</td>
</tr>
<tr>
<td>Multiplier (operator)</td>
<td>MULTI</td>
</tr>
<tr>
<td>Node setting</td>
<td>.NODESET</td>
</tr>
<tr>
<td>Noise analysis</td>
<td>.NOISE</td>
</tr>
<tr>
<td>Options</td>
<td>.OPTIONS</td>
</tr>
<tr>
<td>Parameter definition</td>
<td>PARAM</td>
</tr>
<tr>
<td>Parameter variation</td>
<td>.PARAM</td>
</tr>
<tr>
<td>Parametric analysis</td>
<td>.STEP</td>
</tr>
<tr>
<td>Plot output</td>
<td>.PLOT</td>
</tr>
<tr>
<td>Print output</td>
<td>.PRINT</td>
</tr>
<tr>
<td>Probe</td>
<td>.PROBE</td>
</tr>
<tr>
<td>Sensitivity analysis</td>
<td>.SENS</td>
</tr>
<tr>
<td>Subcircuit call</td>
<td>X_Call</td>
</tr>
<tr>
<td>Subcircuit definition</td>
<td>.SUBCKT</td>
</tr>
<tr>
<td>Summation (operator)</td>
<td>SUM</td>
</tr>
<tr>
<td>Table (operator)</td>
<td>TABLE</td>
</tr>
<tr>
<td>Temperature</td>
<td>.TEMP</td>
</tr>
<tr>
<td>Transfer function</td>
<td>.TF</td>
</tr>
<tr>
<td>Transient analysis</td>
<td>.TRAN</td>
</tr>
<tr>
<td>Value</td>
<td>VALUE</td>
</tr>
<tr>
<td>Value of voltage-controlled voltage source</td>
<td>VALUE</td>
</tr>
<tr>
<td>Width</td>
<td>.WIDTH</td>
</tr>
</tbody>
</table>

**TIP** If you wish to change the model parameters of any device, use a word processing program in ASCII (DOS) text format (or DOS Editor or Notepad in PSpice) to edit the file EVAL.LIB and change the model parameters.
We can add the library file RASHID_MODEL.LIB while running the simulation. First, choose **Library and Include Files** from the Simulation Settings menu, shown in Fig. A.38. Then browse through the library files and select the file RASHID_MODEL.LIB. Click on **Add Library** and then on **OK**. PSpice will look for device models in this library file while running the simulation. The Include Files dialog box is shown in Fig. A.39.
FIGURE A.39 Dialog box for adding library files

References

APPENDIX B

REVIEW OF BASIC CIRCUITS
B.1 Introduction

Electronic devices that are used as parts of electronic circuits are normally modeled by equivalent circuits. Doing a performance evaluation and design of any electronic circuit requires a knowledge of circuit analysis. This appendix reviews the basic circuit theorems and analysis techniques that are commonly used for electronic circuits.

B.2 Kirchhoff’s Current Law

Kirchhoff’s current law (KCL) states that the sum of all currents at a node must be zero; that is,

\[ \sum I_n = 0 \]

where \( I_n \) is the current flowing into \( n \)th node and \( n = 1, 2, 3, \ldots, \infty \).

EXAMPLE B.1

Finding the currents in two parallel resistors For the circuit shown in Fig. B.1, find currents \( I_1 \) and \( I_2 \).

SOLUTION

Using KCL at node 1 gives

\[ I_S - I_1 - I_2 = 0 \quad \text{or} \quad I_S = I_1 + I_2 \quad \text{(B.1)} \]

Since the voltage \( V_S \) across \( R_1 \) is the same as that across \( R_2 \), we can write \( V_S = R_1 I_1 = R_2 I_2 \), which gives \( I_2 = \frac{R_1}{R_2} I_1 \). Substituting for \( I_2 \), we get

\[ I_S = I_1 + \frac{R_1}{R_2} I_1 = \frac{R_2 + R_1}{R_2} I_1 \]

FIGURE B.1 Current distribution in two resistors
which gives the current $I_1$ through $R_1$ as
\[ I_1 = \frac{R_2}{R_1 + R_2} I_S \]  
(B.2)
\[ = \frac{8 \text{ k}\Omega}{4 \text{ k}\Omega + 8 \text{ k}\Omega} \times 120 \text{ mA} = 80 \text{ mA} \]

Similarly, substituting $I_1 = R_2 I_2 / R_1$ in Eq. (B.1) and simplifying, we get the current $I_2$ through $R_2$ as
\[ I_2 = \frac{R_1}{R_1 + R_2} I_S \]  
(B.3)
\[ = \frac{4 \text{ k}\Omega}{4 \text{ k}\Omega + 8 \text{ k}\Omega} \times 120 \text{ mA} = 40 \text{ mA} \]

The branch currents $I_1$ and $I_2$ can be found by applying Ohm’s law as given by
\[ V_S = I_S (R_1 \parallel R_2) = \frac{R_1 R_2}{R_1 + R_2} I_S \]
which can be used to find $I_1 = V_S / R_1$ as in Eq. (B.2) and $I_2 = V_S / R_2$ as in Eq. (B.3).

**NOTE:** Equations (B.2) and (B.3) give the current distribution in two resistors, and the two equations together are often known as the *current divider rule.*

**EXAMPLE B.2**

**Finding the currents in three parallel resistors** For the circuit shown in Fig. B.2, find the currents $I_1$, $I_2$, and $I_3$.

**SOLUTION**

The current source $I_S$ is divided into $I_1$ through $R_1$, $I_2$ through $R_2$, and $I_3$ through $R_3$. Using KCL at node 1 gives
\[ I_S - I_1 - I_2 - I_3 = 0 \quad \text{or} \quad I_S = I_1 + I_2 + I_3 \]  
(B.4)

Since the voltage $V_S$ across $R_1$ is the same as that across $R_2$ and $R_3$, we can write $V_S = R_1 I_1 = R_2 I_2 = R_3 I_3$, which gives $I_2 = R_1 I_1$, $R_2$ and $I_3 = R_1 I_1$, $R_3$. Substituting for $I_2$ and $I_3$ in Eq. (B.4), we get
\[ I_S = I_1 + \frac{R_1}{R_2} I_1 + \frac{R_1}{R_3} I_1 = \frac{R_1 R_2 + R_2 R_3 + R_3 R_1}{R_2 R_3} I_1 \]

\[ I_S = \begin{cases} I_1 + \frac{R_1}{R_2} I_1 + \frac{R_1}{R_3} I_1 = \frac{R_1 R_2 + R_2 R_3 + R_3 R_1}{R_2 R_3} I_1 \end{cases} \]

**FIGURE B.2** Current distribution in three resistors
which gives the current $I_1$ through $R_1$ as

$$I_1 = \frac{1}{R_1} \left( \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} \right) I_S = \frac{R_2 R_3}{R_1 R_2 + R_2 R_3 + R_3 R_1} I_S$$  \hspace{1cm} (B.5)

$$= \frac{4 \text{ kΩ} \times 6 \text{ kΩ}}{2 \text{ kΩ} \times 4 \text{ kΩ} + 4 \text{ kΩ} \times 6 \text{ kΩ} + 6 \text{ kΩ} \times 2 \text{ kΩ}} \times 120 \text{ mA} = 65.45 \text{ mA}$$

Substituting $I_1 = R_2 I_2$ and $I_3 = R_2 I_3$ in Eq. (B.4) yields the current $I_2$ through $R_2$ as

$$I_2 = \frac{1}{R_2} \left( \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} \right) I_S = \frac{R_1 R_3}{R_1 R_2 + R_2 R_3 + R_3 R_1} I_S$$  \hspace{1cm} (B.6)

$$= \frac{2 \text{ kΩ} \times 6 \text{ kΩ}}{2 \text{ kΩ} \times 4 \text{ kΩ} + 4 \text{ kΩ} \times 6 \text{ kΩ} + 6 \text{ kΩ} \times 2 \text{ kΩ}} \times 120 \text{ mA} = 32.73 \text{ mA}$$

Similarly, substituting $I_1 = R_3 I_3$ and $I_2 = R_3 I_2$ in Eq. (B.4) yields the current $I_3$ through $R_3$ as

$$I_3 = \frac{1}{R_3} \left( \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} \right) I_S = \frac{R_1 R_2}{R_1 R_2 + R_2 R_3 + R_3 R_1} I_S$$  \hspace{1cm} (B.7)

$$= \frac{2 \text{ kΩ} \times 4 \text{ kΩ}}{2 \text{ kΩ} \times 4 \text{ kΩ} + 4 \text{ kΩ} \times 6 \text{ kΩ} + 6 \text{ kΩ} \times 2 \text{ kΩ}} \times 120 \text{ mA} = 21.82 \text{ mA}$$

The branch currents $I_1$, $I_2$, and $I_3$ can be found by applying Ohm’s law as given by

$$V_S = I_S (R_1 \parallel R_2 \parallel R_3) = \frac{1}{\left( \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} \right)} I_S$$

which can be used to find $I_1 = V_S / R_1$ as in Eq. (B.5), $I_2 = V_S / R_2$ as in Eq. (B.6), and $I_3 = V_S / R_3$ as in Eq. (B.7).

NOTE: If there are $n$ parallel resistances, the current $I_n$ through the $n$th resistance $R_n$ can be expressed in general form as

$$I_n = \frac{1}{R_n} \left( \frac{1}{R_1} + \frac{1}{R_2} + \cdots + \frac{1}{R_m} \right) I_S$$

### B.3 Kirchhoff’s Voltage Law

Kirchhoff’s voltage law (KVL) states that the sum of the voltages around any loop must be zero; that is,

$$\sum V_n = 0$$

where $V_n$ is the voltage across the $n$th segment of a loop and $n = 1, 2, 3, \ldots, \infty$.  

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
EXAMPLE B.3

Finding the voltage distribution in two resistors  In Fig. B.3, the voltage $V_S$ is divided into $V_1$ across $R_1$ and $V_2$ across $R_2$. Find the voltages $V_1$ and $V_2$.

**SOLUTION**

Using KVL around loop I gives

$$V_S - V_1 - V_2 = 0 \quad \text{or} \quad V_S = V_1 + V_2 \quad (B.8)$$

Since the current $I_S$ through $R_1$ is the same as that through $R_2$, we can write $V_1 = R_1 I_S$ and $V_2 = R_2 I_S$. Substituting for $V_1$ and $V_2$ in Eq. (B.8), we get

$$V_S = V_1 + V_2 = R_1 I_S + R_2 I_S = (R_1 + R_2) I_S$$

which gives the current $I_S$ as

$$I_S = \frac{V_S}{R_1 + R_2}$$
$$= \frac{24 \text{ V}}{4 \text{ k}\Omega + 8 \text{ k}\Omega} = 2 \text{ mA}$$

Therefore, the voltage $V_1$ across $R_1$ can be found from

$$V_1 = R_1 I_S = \frac{R_1}{R_1 + R_2} V_S \quad (B.9)$$
$$= \frac{4 \text{ k}\Omega}{4 \text{ k}\Omega + 8 \text{ k}\Omega} \times 24 \text{ V} = 8 \text{ V}$$

Similarly, the voltage $V_2$ across $R_2$ can be found from

$$V_2 = R_2 I_S = \frac{R_2}{R_1 + R_2} V_S \quad (B.10)$$
$$= \frac{8 \text{ k}\Omega}{4 \text{ k}\Omega + 8 \text{ k}\Omega} \times 24 \text{ V} = 16 \text{ V}$$

**NOTE:** The loop current $I_S$ can be found directly by applying Ohm’s law, $I_S = V_S/(R_1 + R_2)$. Equations (B.9) and (B.10) give the voltage distribution in two resistors only when the current through $R_1$ and $R_2$ is the same. This distribution of voltage is often known as the voltage (or potential) divider rule.

---

**FIGURE B.3** Voltage distribution in two resistors
Appendix B  Review of Basic Circuits

EXAMPLE B.4

Analyzing a circuit with a current-dependent current source For the circuit shown in Fig. B.4 with a current-dependent current source, find the currents \( I_B \), \( I_C \), and \( I_E \) and voltage \( V_C \). Assume \( R_{Th} = 15 \, \text{k}\Omega \), \( r_\pi = 1 \, \text{k}\Omega \), \( R_C = 2 \, \text{k}\Omega \), \( R_E = 500 \, \text{\Omega} \), \( \beta_F = 100 \), \( V_{CC} = 30 \, \text{V} \), and \( V_{Th} = 5 \, \text{V} \).

SOLUTION

Using KCL at node 1, we get

\[
I_E = I_B + I_C = I_B + \beta_F I_B = (1 + \beta_F)I_B \tag{B.11}
\]

Using KVL around loop I, we get

\[
V_{Th} = R_{Th}I_B + r_\pi I_B + R_E I_E = R_{Th}I_B + r_\pi I_B + R_E(1 + \beta_F)I_B
\]

which gives \( I_B \) as

\[
I_B = \frac{V_{Th}}{R_{Th} + r_\pi + R_E(1 + \beta_F)} = \frac{5 \, \text{V}}{15 \, \text{k}\Omega + 1 \, \text{k}\Omega + 500 \, \text{\Omega} \times (1 + 100)} = 75.19 \, \mu\text{A} \tag{B.12}
\]

Current \( I_C \), which is dependent only on \( I_B \), can be found from

\[
I_C = \beta_F I_B = \frac{\beta_F V_{Th}}{R_{Th} + R_\pi + R_E(1 + \beta_F)} = \frac{100 \times 5 \, \text{V}}{15 \, \text{k}\Omega + 1 \, \text{k}\Omega + 500 \, \text{\Omega} \times (1 + 100)} = 7519 \, \mu\text{A} \tag{B.13}
\]

Then

\[ I_E = I_B + I_C = 75.19 \, \mu\text{A} + 7519 \, \mu\text{A} = 7594 \, \mu\text{A} \]

\[ V_C = V_{CC} - I_C R_C = 30 \, \text{V} - 7519 \, \mu\text{A} \times 2 \, \text{k}\Omega = 14.96 \, \text{V} \]

Voltage \( V_E \) can be found from

\[
V_E = R_E I_E = R_E(I_C + I_B) = R_E(\beta_F I_B + I_B) = R_E I_B(1 + \beta_F)
\]
B.4 Superposition Theorem

The superposition theorem states that the current through or voltage across any element in a linear network is equal to the algebraic sum of the currents or voltages produced independently by each source. To calculate the effect of one source, the other independent sources are removed by short-circuiting the voltage sources and open-circuiting the current sources. Any internal resistance associated with the removed voltage sources must be considered, however.

**EXAMPLE B.5**

**Finding the output voltage using the superposition theorem**  
The circuit in Fig. B.6 has a DC source $V_{DC} = 10 \text{ V}$, an AC source $v_{ac} = 15 \sin (377t)$, $R_1 = 2 \text{ k\Omega}$, and $R_2 = 3 \text{ k\Omega}$. Use the superposition theorem to determine the instantaneous output voltage $v_O$.

**SOLUTION**

$V_{DC} = 10 \text{ V}, v_{ac} = 15 \sin (377t), R_1 = 2 \text{ k\Omega}, \text{ and } R_2 = 3 \text{ k\Omega}$. The DC equivalent circuit with source $V_{DC}$ only is shown in Fig. B.7(a); the output voltage due to $V_{DC}$ is

$$V_{01} = \frac{R_2}{R_1 + R_2} V_{DC} = \frac{3 \text{ k}}{2 \text{ k} + 3 \text{ k}} \times 10 = 6 \text{ V}$$
Appendix B  Review of Basic Circuits

Figure B.7(b) shows the AC equivalent circuit with source $v_{ac}$ only; the output voltage due to $v_{ac}$ is

$$v_{o2} = \frac{R_2}{R_1 + R_2} v_{ac} = \frac{3 \, k}{2 \, k + 3 \, k} \times 15 \sin (377 \pi t) = 9 \sin (377 \pi t)$$

Therefore, the resultant output voltage $v_O$ can be found by combining the output voltages due to individual sources; that is,

$$v_O = V_{O1} + v_{o2} = 6 + 9 \sin (377 \pi t) = 3 \times [2 + 3 \sin (377 \pi t)]$$

**EXAMPLE B.6**

**Finding the output voltage using the superposition theorem**  A circuit with three input voltages $V_{S1}$, $V_{S2}$, and $V_{S3}$ is shown in Fig. B.8. Use the superposition theorem to determine the output voltage $V_O$. Use $R_1 = 2 \, k\Omega$, $R_2 = 4 \, k\Omega$, $R_3 = 6 \, k\Omega$, $V_{S1} = 10 \, V$, $V_{S2} = 12 \, V$, and $V_{S3} = 15 \, V$.

**SOLUTION**

The equivalent circuit with source $V_{S1}$ only is shown in Fig. B.9(a). Applying the voltage divider rule, we can find the output voltage due to $V_{S1}$ as

$$V_{O1} = \frac{R_2 R_3}{R_1 + R_2 R_3} V_{S1} = \frac{4 \, k \times 6 \, k}{2 \, k + (4 \, k \times 6 \, k)} \times 10 = 5.45 \, V$$

**FIGURE B.8** Circuit for Example B.6
B.5 Thevenin’s Theorem

Thevenin’s theorem states that any two-terminal linear DC (or AC) network can be replaced by an equivalent circuit consisting of a voltage source and a series resistance (or impedance). This theorem is commonly used to find the voltage (or current) of a linear network with one or more sources. It allows us to concentrate on a specific portion of the network by replacing the remaining network by an equivalent circuit. In the case of sinusoidal AC circuits, the reactances are frequency dependent, and Thevenin’s equivalent circuit is valid for only one frequency.

Figure B.10(a) shows a general DC network; Thevenin’s equivalent circuit is shown in Fig. B.10(b). The steps in determining an equivalent voltage source $V_{Th}$ and an equivalent resistance $R_{Th}$ for Thevenin’s equivalent circuit are as follows:

**Step 1.** Decide on the part of the network for which you desire a Thevenin’s representation and mark the terminals, as shown in Fig. B.10(a).

**Step 2.** Remove that part of the network. In Fig. B.10(a), the load resistance $R_L$ is to be removed.

**Step 3.** Mark the open-circuit terminals of the remaining network—namely, a and b.
Step 4. Determine the open-circuit voltage $V_{Th}$ between terminals a and b.

Step 5. Set all independent sources to zero (voltage sources are replaced by short circuits and current sources by open circuits). Apply a test voltage $V_X$ across terminals a and b. The ratio of $V_X$ to its current $I_X$ gives Thevenin’s resistance $R_{Th}$.

EXAMPLE B.7

Finding Thevenin’s equivalent circuit Represent the network shown in Fig. B.11(a) by Thevenin’s equivalent, as shown in Fig. B.11(b). Assume $V_{CC} = 12 \, \text{V}$, $R_1 = 15 \, \text{k}\Omega$, and $R_2 = 7.5 \, \text{k}\Omega$.

SOLUTION

The open-circuit voltage, which is Thevenin’s voltage between terminals a and b, can be found from the voltage divider rule in Eq. (B.10); that is,

$$
V_{Th} = \frac{R_2}{R_1 + R_2} \frac{V_{CC}}{}
$$

$$
= \frac{7.5 \, \text{k}}{15 \, \text{k} + 7.5 \, \text{k}} \times 12 = 4 \, \text{V}
$$
If source $V_{CC}$ is set to zero and test voltage $V_X$ is applied across terminals a and b, the circuit for determining $R_{Th}$ is shown in Fig. B.12. $R_{Th}$ becomes the parallel combination of $R_1$ and $R_2$. That is,

$$R_{Th} = \frac{V_X}{I_X} = \frac{R_1 \parallel R_2}{15 \text{ k} \parallel 7.5 \text{ k}} = 5 \text{ kΩ}$$  (B.15)

**EXAMPLE B.8**

**Finding Thevenin’s equivalent circuit** Represent the network shown in Fig. B.13(a) by Thevenin’s equivalent, as shown in Fig. B.13(b). Assume $V_{CC} = 12 \text{ V}$, $V_A = 9 \text{ V}$, $R_1 = 15 \text{ kΩ}$, and $R_2 = 7.5 \text{ kΩ}$.

**SOLUTION**

Since there are two voltage sources $V_{CC}$ and $V_A$, we will apply the superposition theorem to find $V_{Th}$. The equivalent circuit with source $V_{CC}$ only is shown in Fig. B.14(a). Applying the voltage divider rule, we can find the output voltage due to $V_{CC}$ only as

$$V_{O1} = \frac{R_2}{R_1 + R_2} V_{CC} = \frac{7.5 \text{ k}}{15 \text{ k} + 7.5 \text{ k}} \times 12 = 4 \text{ V}$$

**FIGURE B.13** Network for Example B.8
Appendix B  Review of Basic Circuits

The circuit with source $V_A$ only is shown in Fig. B.14(b); the output voltage due to $V_A$ is

$$V_{O2} = \frac{R_1}{R_1 + R_2} V_A = \frac{15 \text{ k}}{15 \text{ k} + 7.5 \text{ k}} \times 9 = 6 \text{ V}$$

The resultant output voltage $V_O$, which equals $V_{Th}$, can be found by combining the output voltages due to individual sources; that is,

$$V_{Th} = V_O = V_{O1} + V_{O2} = \frac{R_2}{R_1 + R_2} V_{CC} + \frac{R_1}{R_1 + R_2} V_A$$

$$= 4 + 6 = 10 \text{ V}$$

If sources $V_A$ and $V_{CC}$ are set to zero and test voltage $V_X$ is applied across terminals a and b, the circuit for determining $R_{Th}$ is shown in Fig. B.14(c). $R_{Th}$ becomes the parallel combination of $R_1$ and $R_2$; that is,

$$R_{Th} = \frac{V_X}{I_X} = R_1 \parallel R_2$$

$$= 15 \text{ k} \parallel 7.5 \text{ k} = 5 \text{ k}\Omega$$

It is evident that Eqs. (B.15) and (B.17) are the same.

**EXAMPLE B.9**

Representing a network by Thevenin’s equivalent circuit  Represent the network shown in Fig. B.15 by Thevenin’s equivalent circuit. The circuit values are $R_i = 1.5 \text{ k}\Omega$, $R_C = 25 \text{ k}\Omega$, $\beta_F = 50$, $h_r = 3 \times 10^{-4}$, and $V_s = 5 \text{ mV}$.

(a) Calculate the parameters of Thevenin’s equivalent circuit.

(b) Use PSpice/SPICE to check your results.

**FIGURE B.15**  Network for Example B.9
Appendix B  Review of Basic Circuits

SOLUTION

(a) \( R_i = 1.5 \, \text{k}\Omega, \, R_C = 25 \, \text{k}\Omega, \, \beta_F = 50, \, h_r = 3 \times 10^{-4}, \) and \( V_s = 5 \, \text{mV}. \) The output voltage \( V_o \) between terminals a and b is

\[
V_o = -I_2R_C = -\beta_F I_R R_C \tag{B.18}
\]

The input current \( I_s \) can be found from loop I as

\[
I_s = \frac{V_s - h_r V_o}{R_i} \tag{B.19}
\]

Substituting \( I_s \) from Eq. (B.19) into Eq. (B.18) gives the output voltage \( V_o \) as

\[
V_{Th} = V_o = \frac{-\beta_F R_C}{R_i - \beta_F h_r R_C} V_s = \frac{-50 \times 25 \, \text{k} \times 5 \, \text{m}}{1.5 \, \text{k} - 50 \times 3 \times 10^{-4} \times 25 \, \text{k}} = -5.5556 \, \text{V} \tag{B.20}
\]

Thevenin’s resistance \( R_{Th} \) can be determined from the circuit in Fig. B.16, which is obtained by short-circuiting the independent source \( V_s. \) Considering \( V_x \) and \( I_x \) as the test voltage and current, respectively, we get

\[
I_s = -\frac{h_r V_x}{R_i} \tag{B.21}
\]

\[
I_x = \beta_F I_s + \frac{V_x}{R_C} \tag{B.22}
\]

Substituting \( I_s \) from Eq. (B.21) into Eq. (B.22) gives

\[
I_x = \frac{-\beta_F h_r V_s}{R_i} + \frac{V_x}{R_C} = \frac{R_i - \beta_F h_r R_C}{R_i R_C} V_x
\]

which gives Thevenin’s resistance \( R_{Th} \) as

\[
R_{Th} = \frac{V_x}{I_x} = \frac{R_i R_C}{R_i - \beta_F h_r R_C} = \frac{1.5 \, \text{k} \times 25 \, \text{k}}{1.5 \, \text{k} - 50 \times 3 \times 10^{-4} \times 25 \, \text{k}} = 33.33 \, \text{k}\Omega
\]
B.6 Norton’s Theorem

Norton’s theorem states that any two-terminal linear DC (or AC) network can be replaced by an equivalent circuit consisting of a current source and a parallel resistance (or impedance). Norton’s equivalent circuit can be determined from Thevenin’s equivalent circuit; the relationship between them is shown in Fig. B.18. Norton’s resistance \( R_N \) is identical to Thevenin’s resistance \( R_{Th} \), and Norton’s current equals the short-circuit current at the terminals of interest.

**FIGURE B.17** Circuit for PSpice simulation

(b) The circuit for PSpice simulation is shown in Fig. B.17, which is run by OrCAD PSpice A/D. The results of the PSpice simulation are shown below:

<table>
<thead>
<tr>
<th>NODE</th>
<th>VOLTAGE</th>
<th>NODE</th>
<th>VOLTAGE</th>
<th>NODE</th>
<th>VOLTAGE</th>
<th>NODE</th>
<th>VOLTAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1)</td>
<td>0.0050</td>
<td>(2)</td>
<td>-0.0017</td>
<td>(3)</td>
<td>0.0000</td>
<td>(4)</td>
<td>-5.5556</td>
</tr>
</tbody>
</table>

\[ V_{Th} = V_o = V(4) = -5.5556 \text{ V} \]

**SMALL-SIGNAL CHARACTERISTICS**

\[ V(4)/V_s = -1.111E+03 \]

GAIN \( A = V_o/V_s = -1111 \)

INPUT RESISTANCE AT \( V_s = 1.125E+03 \)

\( R_{in} = V_s/\Delta s = 1.125 \text{ k} \Omega \)

OUTPUT RESISTANCE AT \( V(4) = 3.333E+04 \)

\( R_{Th} = 33.33 \text{ k} \Omega \)

**FIGURE B.18** Thevenin’s and Norton’s equivalent circuits
B.7 Maximum Power Transfer Theorem

In electronic circuits, it is often necessary to transmit maximum power to the load. Let us consider the circuit in Fig. B.19, which could be Thevenin’s equivalent circuit of a network. The power $P_L$ delivered to load resistance $R_L$ can be found from

$$ P_L = I_L^2 R_L = \left( \frac{V_{Th}}{R_{Th} + R_L} \right)^2 R_L $$

$$ = \frac{V_{Th}^2}{R_{Th}} \times \frac{1}{(1 + R_L/R_{Th})^2} \times \frac{R_L}{R_{Th}} $$

For a given circuit, $V_{Th}$ and $R_{Th}$ will be fixed. Therefore, the load power $P_L$ depends on the load resistance $R_L$. If we set $R_L = uR_{Th}$, Eq. (B.24) becomes

$$ P_L = \frac{V_{Th}^2}{R_{Th}} \frac{u}{(1 + u)^2} $$

where $P = V_{Th}^2 R_{Th}$. Normalizing $P_L$ with respect to $P$, we get the normalized power $P_n$ as

$$ P_n = \frac{P_L}{P} = \frac{u}{(1 + u)^2} $$

Figure B.20 shows the variation of normalized power $P_n$ with $u$. The power $P_n$ becomes maximum at $u = 1$; that is, $R_{Th} = uR_L = R_L$. The value of $R_L$ for a maximum power transfer can also be determined from the condition $dP_L / dR_L = 0$. From Eq. (B.24),

$$ \frac{dP_L}{dR_L} = V_{Th}^2 \left[ \frac{(R_{Th} + R_L)^2 - 2R_L(R_{Th} + R_L)}{(R_{Th} + R_L)^4} \right] = 0 $$

or

$$(R_{Th} + R_L)^2 - 2R_L(R_{Th} + R_L) = 0$$

$$ R_L = \pm R_{Th}$$

Since $R_{Th}$ cannot be negative,

$$ R_L = R_{Th} $$

(B.26)
Thus, the maximum power transfer occurs when the load resistance \( R_L \) is equal to Thevenin’s resistance \( R_{Th} \) of the network. For Norton’s equivalent circuit of Fig. B.18(b), the maximum power will be delivered to the load when

\[
R_N = R_L
\]

(B.27)

Substituting \( R_L \) from Eq. (B.26) into Eq. (B.24) gives the maximum power \( P_{\text{max}} \) delivered to the load as

\[
P_{\text{max}} = \frac{V_{Th}^2 R_L}{4 R_L^2} = \frac{V_{Th}^2}{4 R_L}
\]

(B.28)

The input power \( P_{\text{in}} \) supplied by the source \( V_s \) is

\[
P_{\text{in}} = \frac{V_{Th}^2}{R_{Th} + R_L} = \frac{V_{Th}^2}{2 R_L}
\]

(B.29)

Thus, the efficiency \( \eta \) at the maximum power transfer condition is

\[
\eta = \frac{P_{\text{max}}}{P_{\text{in}}} \times 100\% = \frac{V_{Th}^2}{4 R_L} \times \frac{2 R_L}{V_{Th}^2} \times 100\% = 50\%
\]

Therefore, the efficiency is always 50% at the maximum power transfer condition. In electronic circuits, the amount of power being transferred is usually small, and efficiency is often not of primary concern. However, efficiency is of major concern in circuits involving high power—for example, in power systems.

### B.8 Transient Response of First-Order Circuits

The transient response gives the instantaneous value of an output voltage (or current) for a specified instantaneous input voltage (or current). The response due to a step-signal input is commonly used in evaluating electronic circuits because such a response allows us to predict the response due to other signals such as a pulse or square-wave input.
B.8.1 Step Response of Series RC Circuits

Consider the series RC circuit in Fig. B.21(a) with a step input voltage $V_S$. The output voltage $v_O$ is taken across capacitor C. For $t \geq 0$, the charging current $i$ of the capacitor can be found from

$$V_S = v_R + v_C = Ri + \frac{1}{C} \int i \, dt + v_C(t = 0)$$

with initial capacitor voltage $v_C(t = 0) = 0$.

Using the Laplace transformations in Table B.1, we can transform Eq. (B.30) into Laplace’s domain of $s$ as follows:

$$\frac{V_S}{s} = RI(s) + \frac{1}{Cs} I(s)$$

<table>
<thead>
<tr>
<th>$f(t)$</th>
<th>$F(s)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$1$</td>
<td>$\frac{1}{s}$</td>
</tr>
<tr>
<td>$t$</td>
<td>$\frac{1}{s^2}$</td>
</tr>
<tr>
<td>$e^{-\alpha t}$</td>
<td>$\frac{1}{s + \alpha}$</td>
</tr>
<tr>
<td>$\sin \alpha t$</td>
<td>$\frac{\alpha}{s^2 + \alpha^2}$</td>
</tr>
<tr>
<td>$\cos \alpha t$</td>
<td>$\frac{s}{s^2 + \alpha^2}$</td>
</tr>
<tr>
<td>$f'(t)$</td>
<td>$sF(s) - F(0)$</td>
</tr>
<tr>
<td>$f''(t)$</td>
<td>$s^2F(s) - sF(s) - F'(0)$</td>
</tr>
</tbody>
</table>
which, solved for the current $I(s)$, gives

$$I(s) = \frac{V_S}{sR + (1/C)} = \frac{V_S}{R(s + 1/\tau)} \quad (B.31)$$

where $\tau = RC$ is the *time constant* of the circuit.

The inverse transform of Eq. (B.31) in the time domain gives the charging current:

$$i(t) = \frac{V_S}{R} e^{-t/\tau} \quad (B.32)$$

The output voltage $v_O(t)$, which is the voltage across the capacitor, can be expressed as

$$v_O(t) = \frac{1}{C} \int_0^t i \, dt = \frac{1}{C} \int_0^t \frac{V_S}{R} e^{-\tau} \, dt = V_S(1 - e^{-t/\tau}) \quad (B.33)$$

In the steady state ($at t = \infty$), Eq. (B.32) gives

$$i(t = \infty) = 0$$

From Eq. (B.33),

$$v_O(t = \infty) = V_S \quad (B.34)$$

At $t = \tau$, Eq. (B.33) gives

$$v_O(t = \tau) = V_S(1 - e^{-1}) = 0.632V_S \quad (B.35)$$

The initial slope of the tangent to $v_O(t)$ can be found from Eq. (B.33):

$$\left. \frac{dv_O}{dt} \right|_{t=0} = \frac{V_S}{\tau} e^{-t/\tau} \bigg|_{t=0} = \frac{V_S}{\tau} = \frac{V_S}{RC} \quad (B.36)$$

The transient response of $v_O(t)$, which rises exponentially due to a step input, is shown in Fig. B.21(b).

### B.8.2 Step Response of Series CR Circuits

In a CR circuit, the output voltage is taken across resistance $R$ instead of capacitance $C$, as shown in Fig. B.22(a). The output voltage $v_O$, which is the voltage across resistance $R$, can be found from Eq. (B.32); that is,

$$v_O(t) = Ri(t) = V_S e^{-t/\tau} \quad (B.37)$$

which, in the steady state ($at t = \infty$), gives

$$i(t = \infty) = 0$$

$$v_O(t = \infty) = 0$$

At $t = \tau$, Eq. (B.37) gives

$$v_O(t = \tau) = V_S e^{-1} = 0.368V_S \quad (B.38)$$

From Eq. (B.37), the initial slope of the tangent to $v_O(t)$ is

$$\left. \frac{dv_O}{dt} \right|_{t=0} = -\frac{V_S}{\tau} e^{-t/\tau} \bigg|_{t=0} = -\frac{V_S}{\tau} = -\frac{V_S}{RC} \quad (B.39)$$

The response of $v_O(t)$, which falls exponentially due to a step voltage input, is shown in Fig. B.22(b).
B.8.3 Pulse Response of Series RC Circuits

An input pulse $v_S$ of duration $T$, shown in Fig. B.23(a), is applied to the circuit of Fig. B.21(a). The response due to a pulse signal will depend on the ratio of time constant $\tau$ to duration $T$. We consider three cases: $\tau = T$, $\tau << T$, and $\tau >> T$.

In case 1, $\tau = T$, the output voltage $v_O(t)$ has just enough time to reach a near steady-state value of $V_S$. The capacitor $C$ is charged exponentially to approximately voltage $V_S$. When the input voltage $v_S(t)$ falls to zero at $t = T$, the output (or capacitor) voltage $v_O(t)$ falls exponentially to zero, as shown in Fig. B.23(b).

In case 2, $\tau << T$, the output voltage $v_O(t)$ reaches $V_S$ before the input voltage $v_S(t)$ falls to zero. The capacitor $C$ is charged to $V_S$ and then discharges exponentially. The response is shown in Fig. B.23(c).

In case 3, $\tau >> T$, the input pulse $v_S(t)$ is effectively constant for most of the time constant $\tau$. The capacitor $C$ charges to $0.9V_S$ and then discharges exponentially. The response is shown in Fig. B.23(d).
in Fig. B.23(b). The area under the input waveform must equal that under the output waveform. *Rise time* $t_r$ is defined as the time it takes for the output voltage to rise from 10% to 90% of the final value. *Fall time* $t_f$ is defined as the time it takes for the output voltage to fall from 90% to 10% of the initial value. *Delay time* $t_d$ is defined as the time it takes for the output voltage to rise from 0% to 10% of the final value.

At $t = t_1 = t_d$, $v_O(t) = 0.1V_S$; at $t_2$, $v_O(t) = 0.9V_S$. Thus, Eq. (B.33) gives

\[
0.1V_S = V_S(1 - e^{-t_1/\tau})
\]

\[
e^{-t_1/\tau} = 0.9
\]

\[
t_1 = -\tau \ln(0.9)
\]

and

\[
0.9V_S = V_S(1 - e^{-t_2/\tau})
\]

\[
e^{-t_2/\tau} = 0.1
\]

\[
t_2 = -\tau \ln(0.1)
\]

The rise time $t_r$, which is equal to the fall time $t_f$, can be found as follows:

\[
t_r = t_f = t_2 - t_1
\]

\[
= -\tau \ln(0.1) + \tau \ln(0.9) = \tau \ln(9) = 2.2\tau
\]

(B.40)

In case 2, $\tau << T$, $t_r$ and $t_f$ are much smaller than $T$. The output voltage $v_O(t)$ represents the input signal more closely, as shown in Fig. B.23(c). This condition is generally satisfied by choosing circuit parameters such that $10\tau = T$.

In case 3, $\tau >> T$, there is not enough time for the output voltage $v_O(t)$ to reach the steady-state value of $V_S$. The output voltage at $t = T$ is $V_1$, which is much smaller than $V_S$, as shown in Fig. B.23(d). The output voltage starts to decay exponentially toward zero before reaching its maximum value. Thus, the output voltage will not be a true representative of the input voltage. However, the output voltage is approximately the time integration of the input voltage, and the circuit behaves as an integrator. That is,

\[
v_O(t) = \frac{1}{\tau} \int_0^t V_S dt = \frac{V_S}{\tau} t \quad \text{(for } \tau >> T)\]

For this condition, $\tau = 10T$.

**EXAMPLE B.10**

**Using PSpice/SPICE to plot the pulse response of an RC circuit** Use PSpice/SPICE to plot the output voltage of the circuit in Fig. B.21(a) for $\tau = 0.1$ ms, 1 ms, and 5 ms. Assume $T = 2$ ms and $V_S = V_3 = 1$ V of pulse input.

**SOLUTION**

For $\tau = 0.1$ ms, let $C = 0.1$ $\mu$F. Then

\[
R = \frac{\tau}{C} = \frac{0.1 \text{ ms}}{0.1 \text{ } \mu\text{F}} = 1 \text{ kΩ}
\]
For $\tau = 1 \text{ ms}$, let $C = 0.1 \, \mu\text{F}$. Then

$$R = \frac{\tau}{C} = \frac{1 \, \text{ms}}{0.1 \, \mu\text{F}} = 10 \, \text{k}\Omega$$

For $\tau = 5 \, \text{ms}$, let $C = 0.1 \, \mu\text{F}$. Then

$$R = \frac{\tau}{C} = \frac{5 \, \text{ms}}{0.1 \, \mu\text{F}} = 50 \, \text{k}\Omega$$

The series $RC$ circuit for PSpice simulation is shown in Fig. B.24 with a pulse input voltage. The list of the circuit file, which is run by OrCAD PSpice A/D, is as follows.

Example B.10  Pulse Response of Series RC Circuit

```
.PARAM RVAL=1K
.STEP PARAM RVAL LIST  1K  10K  50K
VS  1  0  PULSE (0V  1V  0  1NS  1NS  2MS  4MS) ; Pulse input voltage
R  1  2  {RVAL}
C  2  0  0.1UF
.TRAN 0.1MS 4MS ; Transient analysis
.PROBE
.END
```

The PSpice plots of the output voltage $v_0(t)$ for three values of the time constant are shown in Fig. B.25. The lower the time constant $\tau$, the faster the output voltage rises and falls.

NOTE: You can use the PSpice Parametric command for variable $R$ to vary the time constant.
B.8.4 Pulse Response of Series CR Circuits

An input pulse $v_S$ of duration $T$, shown in Fig. B.26(a), is applied to the circuit of Fig. B.22(a). The response due to a pulse signal will depend on the ratio of time constant $\tau$ to duration $T$. Let us consider three cases: $\tau = T$, $\tau << T$, and $\tau >> T$.

In case 1, $\tau = T$, the capacitor voltage $v_C(t)$ starts to increase exponentially while the output voltage $v_O(t)$ starts to decay exponentially from $V_S$. This situation is shown in Fig. B.26(b). At $t = T$, the input signal $v_S$ falls to zero and the capacitor discharges exponentially through the resistance $R$ and the input source $v_S$. The output voltage $v_O$ decays exponentially from a negative value toward zero.

**FIGURE B.26** Pulse response of a series CR circuit

B.8.4 Pulse Response of Series CR Circuits

An input pulse $v_S$ of duration $T$, shown in Fig. B.26(a), is applied to the circuit of Fig. B.22(a). The response due to a pulse signal will depend on the ratio of time constant $\tau$ to duration $T$. Let us consider three cases: $\tau = T$, $\tau << T$, and $\tau >> T$.

In case 1, $\tau = T$, the capacitor voltage $v_C(t)$ starts to increase exponentially while the output voltage $v_O(t)$ starts to decay exponentially from $V_S$. This situation is shown in Fig. B.26(b). At $t = T$, the input signal $v_S$ falls to zero and the capacitor discharges exponentially through the resistance $R$ and the input source $v_S$. The output voltage $v_O$ decays exponentially from a negative value toward zero.
In case 2, $\tau << T$, the output voltage $v_O(t)$ decays exponentially with a small time constant to zero. During the period $0 \leq t \leq T$, the capacitor charges exponentially to reach a steady-state value of $V_S$. For $t > T$, the capacitor discharges exponentially with a small time constant through the resistance $R$ and the input source $v_S$. The output voltage $v_O(t)$ decays exponentially from a negative value toward zero. The waveforms for $v_O(t)$ and $v_C(t)$ are shown in Fig. B.26(c).

In case 3, $\tau >> T$, output voltage $v_O$ falls only a small amount. The portion of the exponential curve $v_O$ from $t = 0$ to $t = T$ will be almost linear, as shown in Fig. B.26(d). The fall in the output voltage $v_O$ can be found approximately from Fig. B.26(d) as

$$\Delta V = \frac{V_S}{\tau} T$$

(B.41)

The sag $S$ of the output voltage is defined as

$$S = \frac{\Delta V}{V_S} = \frac{V_S T/\tau}{V_S} = \frac{T}{\tau} = \frac{T}{RC}$$

(B.42)

EXAMPLE B.11

Using PSpice/SPICE to plot the pulse response of a CR circuit  Use PSpice/SPICE to plot the output voltage $v_O(t)$ of the circuit in Fig. B.22(a) for $\tau = 0.1$ ms, 1 ms, and 5 ms. Assume $T = 2$ ms and $v_S = V_S = 1$ V of pulse input.

SOLUTION

For $\tau = 0.1$ ms, let $C = 0.1$ $\mu$F. Then

$$R = \frac{\tau}{C} = \frac{0.1 \text{ ms}}{0.1 \mu\text{F}} = 1 \text{ k}\Omega$$

For $\tau = 1$ ms, let $C = 0.1$ $\mu$F. Then

$$R = \frac{\tau}{C} = \frac{1 \text{ ms}}{0.1 \mu\text{F}} = 10 \text{ k}\Omega$$

For $\tau = 5$ ms, let $C = 0.1$ $\mu$F. Then

$$R = \frac{\tau}{C} = \frac{5 \text{ ms}}{0.1 \mu\text{F}} = 50 \text{ k}\Omega$$

The series CR circuit for PSpice simulation is shown in Fig. B.27.
Appendix B  Review of Basic Circuits

1236

The list of the circuit file is as follows.

Example B.11  Pulse Response of Series CR Circuit

```
.PARAM RVAL=1K
.STEP PARAM RVAL LIST 1K 10K 50K
VS 1 0 PULSE (0V 1V 0 1NS 1NS 2MS 4MS) ; Pulse input voltage
R 2 0 {RVAL}
C 1 2 0.1UF
.TRAN 0.1MS 4MS ; Transient analysis
.PROBE
.END
```

The plots of the output voltage $v_O(t)$ for three values of the time constant are shown in Fig. B.28.

**FIGURE B.28**  Plots of $v_O(t)$ for Example B.11

**EXAMPLE B.12**

**Pulse response of a parallel RC circuit**  A constant-current source $i_S = I_S$, shown in Fig. B.29(a), feeds a parallel RC circuit with $C = 0.1 \mu F$ and $R = 100 \, \text{k}\Omega$, as shown in Fig. B.29(b). The input is a pulse current of duration $T = 0.5 \, \text{ms}$. Determine (a) the instantaneous current $i_C(t)$ through capacitance $C$, (b) the instantaneous current $i_R(t)$ through resistance $R$, and (c) the sag $S$ of the capacitor current.

**FIGURE B.29**  Parallel RC circuit with constant current source $I_S$
B.8.5 Step Response of Series \( RL \) Circuits

A series \( RL \) circuit with a step-function input is shown in Fig. B.30(a). The output voltage \( v_O \) is taken across inductance \( L \). The current \( i \) through the inductor can be deduced from

\[
V_S = v_L + v_R = L \frac{di}{dt} + Ri
\]  

(B.46)

\[
I_C(s) = \frac{R}{R + 1/CS} I_S(s) = \frac{s}{s + 1/RC} I_S(s) = \frac{s}{s + 1/RC} \times \frac{I_S}{s} = \frac{I_S}{s + 1/RC}
\]

The inverse transform of \( I_C(s) \) in Eq. (B.43) gives

\[
i_C(t) = I_S e^{-t/\tau}
\]

(B.44)

where \( \tau = RC \).

(b) The instantaneous current \( i_R(t) \) through resistance \( R \) is

\[
i_R(t) = I_S - i_C(t) = I_S(1 - e^{-t/\tau})
\]

(B.45)

(c) \( \tau = RC = 100 \times 10^3 \times 0.1 \times 10^{-6} = 10 \text{ ms} \), and \( T = 0.5 \text{ ms} \). Therefore, \( \tau >> T \), and Eq. (B.42) gives

\[
S = T/\tau = 0.5/10 = 5%\]

\[\text{Slope} = V_S/\tau\]

\[\text{Slope} = -V_S/\tau\]

\[v_O = v_L\]

\[0.368V_S\]

\[0.632V_S\]

\[V_S/R\]

\[V_S\]

\[V_O\]

\(t\)

\(t\)

\(t\)

\(t\)

FIGURE B.30 Step response of a series \( RL \) circuit
with an initial inductor current of zero, \( i(t = 0) = 0 \). In Laplace’s domain of \( s \), Eq. (B.46) becomes

\[
\frac{V_S}{s} = LsI(s) + RI(s)
\]

which, solved for \( I(s) \), gives

\[
I(s) = \frac{V_S}{s(sL + R)} = \frac{V_S}{Ls + 1/\tau} = \frac{V_S}{R} \left[ \frac{1}{s} - \frac{1}{(s + 1/\tau)} \right]
\]

(B.47)

where \( \tau = L/R \) is the time constant of an \( RL \) circuit. Taking the inverse transform of \( I(s) \) in Eq. (B.47) gives the instantaneous current as

\[
i(t) = \frac{V_S}{R} (1 - e^{-t/\tau})
\]

(B.48)

Using Eq. (B.48), we can find the voltage \( v_O(t) \) across the inductance \( L \):

\[
v_O(t) = v_L(t) = L \frac{di}{dt} = V_S e^{-t/\tau}
\]

(B.49)

In the steady state (at \( t = \infty \)),

\[
v_O(t) = 0 \quad \text{[from Eq. (B.49)]}
\]

\[
i(t) = \frac{V_S}{R} \quad \text{[from Eq. (B.48)]}
\]

If the output is taken across resistance \( R \), the output voltage \( v_O(t) \) becomes

\[
v_O(t) = v_R(t) = Ri(t) = V_S (1 - e^{-t/\tau})
\]

(B.50)

In the steady state (at \( t = \infty \)), \( v_R(t) = V_S \) and \( i(t) = V_S R \).

**NOTE** Under steady-state conditions, the current through the inductor is \( V_S R \). If the input voltage \( V_S \) is turned off with a sharp edge, a very high voltage will be induced by the inductor in order to oppose this change of current. This voltage could be destructive. A series \( RL \) circuit is not operated with a pulse (or step) signal input unless there is a protection circuit to suppress the voltage transient caused by the inductor. If the input voltage \( V_S \) switches between \( +V_S \) and \( -V_S \), the inductor current can rise and fall, resulting in positive and negative voltages.

### B.9 Resonant Circuits

The effective impedance of an \( RLC \) circuit is a function of the frequency, and the voltage or current becomes maximum at a frequency \( f_n \), known as the resonant (or natural) frequency. At resonance, the energy absorbed at any instant by one reactive element (say, inductor \( L \)) is exactly equal to that released by another element (say, capacitor \( C \)). The energy pulsates from one reactive element to the other, and a circuit with no resistive element requires no further reactive power from the input source. The average input power, which is the power dissipated in the resistive element, becomes maximum at resonance. Resonant circuits are of two types: series resonant circuits and parallel resonant circuits.
B.9.1 Series Resonant Circuits

A series resonant RLC circuit is shown in Fig. B.31, where $R_{Cl}$ is the internal resistance of the coil and $R_S$ is the source resistance. If we define $R = R_{Cl} + R_S$, the total series impedance $Z$ of the circuit is given by

$$Z = R + j(X_L - X_C) \tag{B.51}$$

The series resonance will occur at $f = f_n$ when

$$X_L = X_C \tag{B.52}$$

Equation (B.52) can also be stated as

$$\omega L = \frac{1}{\omega C} \quad \text{or} \quad 2\pi f_n L = \frac{1}{(2\pi f_n C)}$$

which gives the series resonant frequency $f_n$ as

$$f_n = \frac{1}{2\pi \sqrt{LC}} \tag{B.53}$$

The impedance $Z_n$ at resonance becomes

$$Z_n = Z = R \tag{B.54}$$

A series resonant circuit is normally defined by a quality factor $Q_s$, which is defined as the ratio of the reactive power stored in either the inductor or the capacitor to the average power dissipated in the resistor at resonance; that is,

$$Q_s = \frac{\text{Reactive power}}{\text{Average power}}$$

$$Q_s = \begin{cases} \frac{I^2X_L}{I^2R} = \frac{X_L}{R} = \frac{2\pi f_n L}{R} & \text{(for an inductive reactance)} \\ \frac{I^2X_C}{I^2R} = \frac{X_C}{R} = \frac{1}{2\pi f_n CR} & \text{(for a capacitive reactance)} \end{cases} \tag{B.55}$$
The quality factor $Q_{C1}$ of a coil is defined as the ratio of the reactive power stored in the coil to the power dissipated in the coil resistance $R_{C1}$; that is,

$$Q_{C1} = \frac{\text{Reactive power}}{\text{Power dissipated}} = \frac{X_L}{R_{C1}}$$

The rms voltage $V_L$ across inductor L at resonance can be found from

$$V_L = \frac{X_L V_s}{Z_n} = \frac{X_L V_s}{R} = Q_s V_s \quad (B.57)$$

The rms voltage $V_C$ across capacitor C at resonance can be found from

$$V_C = \frac{X_C V_s}{Z_n} = \frac{X_L V_s}{R} = Q_s V_s \quad (B.58)$$

In many electronic circuits, the quality factor $Q_s$ is high, in the range of 80 to 400. If $V_s = 30$ V and $Q_s = 80$, for example, then $V_C = V_L = 80 \times 30 = 2400$ V, and all electronic devices in the circuit will be subjected to this high voltage. Thus, a designer must be careful to protect the circuit from a high voltage across the inductor or the capacitor of a resonant circuit.

### B.9.2 Parallel Resonant Circuits

A parallel resonant $RLC$ circuit is shown in Fig. B.32(a). This circuit is also known as a tank circuit. The input signal to a tank circuit is usually a current source. This type of circuit is frequently used with active devices such as transistors, which have the characteristic of a constant current source. By replacing the series $RL$ combination of the inductor with a parallel combination, we can obtain the circuit in Fig. B.32(b), for which the admittance $Y_{RL}$ is

$$Y_{RL} = \frac{1}{R_{C1} + jX_L} = \frac{R_{C1}}{R_{C1}^2 + X_L^2} - j \frac{X_L}{R_{C1}^2 + X_L^2} = \frac{1}{R_p} - j \frac{1}{X_p}$$

![Parallel circuit and equivalent circuit](image_url)

**FIGURE B.32** Parallel resonant $RLC$ circuit
For the resonant condition,

\[ X_p = X_C \]

Substituting \( X_p \) from Eq. (B.60) into the preceding equation, we get

\[
\frac{R_{C1}^2 + X_L^2}{X_L} = X_C
\]

or

\[
R_{C1}^2 + X_L^2 = X_C X_L
\]

\[
X_L^2 = X_C X_L - R_{C1}^2
\]

which gives the parallel resonant frequency \( f_p \) as

\[
f_p = \frac{1}{2\pi L} \left[ \frac{L}{C} - R_{C1}^2 \right]^{1/2} = \frac{1}{2\pi \sqrt{LC}} \left[ 1 - \frac{CR_{C1}^2}{L} \right]^{1/2}
\]

\[
= f_n \left[ 1 - \frac{CR_{C1}^2}{L} \right]^{1/2}
\]

Thus, the parallel resonant frequency \( f_p \), which is dependent on the coil resistance \( R_{C1} \), is less than the series resonant frequency \( f_n \). For the conditions \( CR_{C1} L << 1 \) or \( R_{C1} << \sqrt{L/C} \), and for \( R_{C1} = 0 \), Eq. (B.63) gives

\[ f_p = f_n \]

The quality factor \( Q_p \) of the parallel resonant \( RLC \) circuit can be determined by the ratio of reactive power to the real power at resonance; that is,

\[
Q_p = \frac{V_p^2/X_p}{V_p^2/(R_S \parallel R_p)} = \frac{R_S \parallel R_p}{X_p}
\]

where \( V_p \) is the voltage across the parallel branches.
Appendix B  Review of Basic Circuits

EXAMPLE B.13

Finding the parallel resonant frequency  The parameters of the parallel resonant RLC circuit in Fig. B.32(a) are $R_{C1} = 47 \, \Omega$, $L = 5 \, \text{mH}$, $C = 50 \, \text{pF}$, $R_S = 20 \, \text{k}\Omega$, and the current source $I_s = 6 \, \text{mA}$. Calculate (a) the parallel resonant frequency $f_p$, (b) the voltage $V_p$ across the resonant circuit at resonance, (c) the quality factor $Q_{C1}$ of the coil, and (d) the quality factor $Q_p$ of the resonant circuit.

SOLUTION

$R_{C1} = 47 \, \Omega$, $L = 5 \, \text{mH}$, $C = 50 \, \text{pF}$, $R_S = 20 \, \text{k}\Omega$, and $I_s = 6 \, \text{mA}$.

(a) From Eq. (B.53),

$$f_p = \frac{1}{2\pi \sqrt{L \times C \times R_{C1}}} = 318.3 \, \text{kHz}$$

From Eq. (B.63),

$$f_p = 318.3 \times 10^3 \times \left[ 1 - 50 \times 10^{-12} \times \frac{47^2}{5 \times 10^{-3}} \right]^{1/2} \approx 318.3 \, \text{kHz}$$

(b) We know that

$$X_L = 2\pi f_p L = 2\pi \times 318.3 \times 10^3 \times 5 \times 10^{-3} = 9999.7 \, \Omega$$

From Eq. (B.59), the effective resistance $R_p$ of the parallel circuit is

$$R_p = \frac{47^2 + 9999.7^2}{47} = 2127.6 \, \text{k}\Omega$$

Using the current divider rule, the rms current $I_p$ through the parallel circuit is

$$I_p = \frac{R_S}{R_S + R_p} I_s = \frac{20 \, \text{k}\Omega \times 6 \, \text{mA}}{20 \, \text{k}\Omega + 2127.6 \, \text{k}\Omega} = 55.876 \, \mu\text{A}$$

$$V_p = I_p R_p = 55.876 \, \mu\text{A} \times 2127.6 \, \text{k}\Omega = 118.88 \, \text{V}$$

(c) We have

$$Q_{C1} = \frac{X_L}{R_{C1}} = \frac{9999.7}{47} = 212.8$$

(d) From Eq. (B.60), the effective inductive reactance $X_p$ of the parallel circuit is

$$X_p = \frac{47^2 + 9999.7^2}{9999.7} = 9999.92 \, \Omega$$

$$R_S \parallel R_p = \frac{20 \times 2127.6}{20 + 2127.6} = 19.81 \, \text{k}\Omega$$

From Eq. (B.64), $Q_p = 19,810 \times 9999.92 = 1.98$. 

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
B.10 Frequency Response of First- and Second-Order Circuits

A sine wave (or sinusoid) is generally used to characterize electronic circuits, such as amplifiers and filters. The frequency response refers to the output characteristic for a sine-wave input. If a sine-wave input voltage

\[ v_s(t) = V_m \sin \omega t \] (B.65)

where \( V_m \) is the peak input voltage and \( \omega \) is the frequency of the input voltage in radians per second, is applied to a circuit, the output voltage \( v_o(t) \) can have a different amplitude and phase than the input voltage. The output voltage \( v_o(t) \) will be of the form

\[ v_o(t) = V_p \sin (\omega t - \phi) \] (B.66)

where \( V_p \) is the peak output voltage. If \( f \) is the input frequency in hertz,

\[ \omega = 2\pi f \]

A typical relationship between the input and output voltages of an amplifier is shown in Fig. B.33. An amplifier has in general a voltage gain of \( A_v = V_p/V_m > 1 \). Thus, the peak output \( V_p \) has a higher value than the input voltage \( V_m \).

If \( V_s(j\omega) \) and \( V_o(j\omega) \) denote the rms values of input voltage and output voltage, respectively, as a function of frequency, the voltage gain \( G(j\omega) \) is defined as

\[ G(j\omega) = \frac{V_o(j\omega)}{V_s(j\omega)} \quad \text{(dimensionless)} \] (B.67)

\( G(j\omega) \) is a complex function with a magnitude and a phase. The magnitude \( |G(j\omega)| \) gives the magnitude response, and the phase of \( G(j\omega) \) gives the phase response. The magnitude and the phase are generally plotted against the frequency, with a logarithmic scale used for the frequency. The magnitude \( |G(j\omega)| \) is normally expressed in decibels (dB):

\[ \text{Magnitude in dB} = 20 \log_{10} |G(j\omega)| \]

We will review the frequency responses of the following circuits: first-order low-pass \( RC \) circuits, first-order high-pass \( CR \) circuits, second-order series \( RLC \) circuits, and second-order parallel \( RLC \) circuits.
B.10.1 First-Order Low-Pass RC Circuits

A typical low-pass RC circuit is shown in Fig. B.34(a). The output voltage \( v_o \) is taken across capacitance \( C \). The capacitor impedance in Laplace’s domain is \( \frac{1}{sC} \). Using the voltage divider rule, we can find the voltage gain \( G(s) \):

\[
G(s) = \frac{V_o(s)}{V_s(s)} = \frac{1/Cs}{R + (1/Cs)} = \frac{1}{1 + sRC} \quad \text{(dimensionless)}
\]

In the frequency domain, \( s = j\omega \), and

\[
G(j\omega) = \frac{1}{1 + j\omega RC} = \frac{1}{1 + j\omega\tau}
\]

where \( \tau = RC \). Thus, the magnitude \( |G(j\omega)| \) of the voltage gain can be found from

\[
|G(j\omega)| = \frac{1}{\left[1 + (\omega\tau)^2\right]^{1/2}} = \frac{1}{\left[1 + (\omega/\omega_o)^2\right]^{1/2}} \quad \text{(B.68)}
\]

and the phase angle \( \phi \) of \( G(j\omega) \) is given by

\[
\phi = -\tan^{-1}(\omega\tau) = -\tan^{-1}\left(\frac{\omega}{\omega_o}\right) \quad \text{(B.69)}
\]

where \( \omega_o = 1 \ RC = 1 \ \tau. \)
For \( \omega << \omega_o \),

\[
|G(j\omega)| \approx 1
\]

\[
20 \log_{10} |G(j\omega)| = 0
\]

\[
\phi = 0
\]

Therefore, at low frequency, the magnitude plot is a straight horizontal line at 0 dB. For \( \omega >> \omega_o \),

\[
|G(j\omega)| \approx \omega_o/\omega
\]

\[
20 \log_{10} |G(j\omega)| = 20 \log_{10} (\omega_o/\omega)
\]

\[
\phi = \pi/2
\]

For \( \omega = \omega_o \),

\[
|G(j\omega)| = 1/\sqrt{2}
\]

\[
20 \log_{10} |G(j\omega)| = 20 \log_{10} (1/\sqrt{2}) = -3 \text{ dB}
\]

\[
\phi = \pi/4
\]

Let us consider a high frequency such that \( \omega_1 >> \omega_o \). At \( \omega = \omega_1 \), the magnitude is \( 20 \log_{10} (\omega_o/\omega_1) \). At \( \omega = 10\omega_1 \), the magnitude is \( 20 \log_{10} (\omega_o/10\omega_1) \). The change in magnitude between \( \omega = \omega_1 \) and \( \omega = 10\omega_1 \) becomes

\[
20 \log_{10} (\omega_o/10\omega_1) - 20 \log_{10} (\omega_o/\omega_1) = 20 \log_{10} (1/10) = -20 \text{ dB}
\]

If the frequency is doubled so that \( \omega = 2\omega_1 \), the change in magnitude becomes

\[
20 \log_{10} (\omega_o/2\omega_1) - 20 \log_{10} (\omega_o/\omega_1) = 20 \log_{10} (1/2) = -6 \text{ dB}
\]

The frequency response is shown in Fig. B.34(b). If the frequency doubled, the interval between the two frequencies is called an *octave* on the frequency axis. If the frequency is increased by a factor of 10, the interval between the two frequencies is called a *decade*. Thus, for a decade increase in frequency, the magnitude changes by \(-20 \text{ dB}\). The magnitude plot is a straight line with a slope of \(-20 \text{ dB/decade}\) or \(-6 \text{ dB/octave}\). The magnitude curve is therefore defined by two straight-line asymptotes, which meet at the *corner frequency* (or *break frequency*) \( \omega_o \). The difference between the actual magnitude curve and the asymptotic curve is largest at the break frequency. The error can be found by finding the gain at \( \omega = \omega_o \); that is, \( |G(j\omega)| = 1/\sqrt{2} \) and \( 20 \log_{10} (1/\sqrt{2}) = -3 \text{ dB} \). This error is symmetrical with respect to the break frequency. The break frequency is also known as the *3-dB frequency*.

The circuit in Fig. B.34(a) passes only the low-frequency signal, and the amplitude falls at higher frequencies. A circuit with this type of response is known as a *low-pass circuit*. The gain function (commonly known as the transfer function) of a low-pass circuit has the general form

\[
G(s) = \frac{K}{1 + (s/\omega_o)} \quad (B.70)
\]
where $K$ is the magnitude of the gain function at $\omega = 0$ (or the low-pass gain). A low-pass circuit exhibits the characteristics of (a) finite output at a very low frequency, tending to zero, and (b) zero output at a very high frequency, tending to infinity.

**EXAMPLE B.14**

Using PSpice/SPICE to plot the frequency response of a low-pass $RC$ circuit Use PSpice/SPICE to plot the frequency response of the low-pass $RC$ circuit in Fig. B.34(a). Assume $V_m = 1$ V (peak AC), $R = 10$ k$\Omega$, and $C = 0.1$ $\mu$F. The frequency $f$ varies from 1 Hz to 100 kHz.

**SOLUTION**

The low-pass $RC$ circuit for PSpice simulation is shown in Fig. B.35. The PSpice plot of the magnitude and phase angle are shown in Fig. B.36, which gives $f_0 = 161$ Hz at $-3$ dB.

**B.10.2 First-Order High-Pass $CR$ Circuits**

A high-pass $CR$ circuit is shown in Fig. B.37(a). The output voltage $v_o$ is taken across resistance $R$. Using the voltage divider rule, we can find the voltage gain $G(s)$ in Laplace’s domain:

$$G(s) = \frac{V_o(s)}{V_i(s)} = \frac{R}{R + (1/Cs)} = \frac{sRC}{1 + sRC} \quad \text{(dimensionless)}$$

In the frequency domain, $s = j\omega$, and

$$G(j\omega) = \frac{j\omega RC}{1 + j\omega RC} = \frac{j\omega\tau}{1 + j\omega\tau}$$
where $\tau = RC$. Thus, the magnitude $|G(j\omega)|$ of the voltage gain can be found from

$$|G(j\omega)| = \frac{\omega \tau}{[1 + (\omega \tau)^2]^{1/2}} = \frac{\omega/\omega_0}{[1 + (\omega/\omega_0)^2]^{1/2}}$$

(B.71)

and the phase angle $\phi$ of $G(j\omega)$ is given by

$$\phi = \frac{\pi}{2} - \tan^{-1}\left(\frac{\omega}{\omega_0}\right)$$

(B.72)

where $\omega_0 = 1$ $RC = 1 \tau$.

For $\omega << \omega_0$,

$$|G(j\omega)| \approx \frac{\omega}{\omega_0}$$

$$20 \log_{10}|G(j\omega)| = 20 \log_{10}\left(\frac{\omega}{\omega_0}\right)$$

$$\phi = \frac{\pi}{2}$$

Therefore, for a decade increase in frequency, the magnitude changes by 20 dB. The magnitude plot is a straight line with a slope of $+20$ dB/decade or $+6$ dB/octave.

For $\omega >> \omega_0$,

$$|G(j\omega)| = 1$$

$$20 \log_{10}|G(j\omega)| = 0$$

$$\phi \approx 0$$

FIGURE B.37 First-order high-pass CR circuit
Therefore, at high frequency, the magnitude plot is a straight horizontal line at 0 dB. For \( \omega = \omega_0 \),

\[
|G(j\omega)| = \frac{1}{\sqrt{2}}
\]

\[
20 \log_{10} \left( \frac{1}{\sqrt{2}} \right) = -3 \text{ dB}
\]

\[\phi = \frac{\pi}{4}\]

The frequency response is shown in Fig. B.37(b). This circuit passes only the high-frequency signal, and the amplitude is low at low frequencies. This type of circuit is known as a high-pass circuit. The gain function of a high-pass circuit has the general form

\[
G(s) = \frac{sK}{1 + s/\omega_0}
\]  

(B.73)

where \( K \) is the high-pass gain. A high-pass circuit exhibits the characteristics of (a) zero output at a very low frequency, tending to zero, and (b) finite output at a very high frequency, tending to infinity.

**EXAMPLE B.15**

**Using PSpice/SPICE to plot the frequency response of a high-pass CR circuit** Use PSpice/SPICE to plot the frequency response of the high-pass CR circuit in Fig. B.37(a). Assume \( V_m = 1 \text{ V (peak AC)} \), \( R = 10 \text{ k\Omega} \), and \( C = 0.1 \mu\text{F} \). The frequency \( f \) varies from 1 Hz to 100 kHz.

**SOLUTION**

The high-pass CR circuit for PSpice simulation is shown in Fig. B.38. The PSpice plots of the magnitude and phase angle are shown in Fig. B.39, which gives \( f_0 = 157 \text{ Hz} \) at -3 dB.
B.10.3 Second-Order Series RLC Circuits

A series RLC circuit is shown in Fig. B.40. The output voltage $v_o$ is taken across resistance $R$. Using the voltage divider rule, we can find the voltage gain (or the transfer function) in Laplace’s domain of $s$:

$$G(s) = \frac{V_o(s)}{V(s)} = \frac{R}{R + sL + (1/Cs)} = \frac{sR/L}{s^2 + (sR/L) + (1/LC)} \quad \text{(dimensionless)} \quad (B.74)$$

Defining $\omega_n = 1/\sqrt{LC}$ as the natural frequency in rad/s and $\alpha = R/(2L)$ as the damping factor, we can write Eq. (B.74) as

$$G(s) = \frac{2\alpha s}{s^2 + 2\alpha s + \omega_n^2} \quad \text{(B.75)}$$

Let us define

$$\delta = \frac{\alpha}{\omega_n} = \frac{R}{2L} \sqrt{LC} = \frac{R}{2\sqrt{L}}$$

as the damping ratio. Then Eq. (B.75) becomes

$$G(s) = \frac{2\delta \omega_n s}{s^2 + 2\delta \omega_n s + \omega_n^2} \quad \text{(dimensionless)} \quad (B.76)$$

where $\delta < 1$. (Note that $\delta$ is not necessarily less than 1, but has been assumed at less than 1 for this discussion.) In the frequency domain, $s = j\omega$. Thus,

$$G(j\omega) = \frac{2\delta \omega_n j\omega}{(j\omega)^2 + 2\delta \omega_n (j\omega) + \omega_n^2} = \frac{j2\delta \omega/\omega_n}{-(\omega/\omega_n)^2 + (j2\delta \omega/\omega_n) + 1}$$

$$= \frac{j2\delta \omega/\omega_n}{1 + (j2\delta \omega/\omega_n) - (\omega/\omega_n)^2} \quad (B.77)$$

Let us define $u = \omega/\omega_n$ as the frequency ratio or the normalized frequency. Then Eq. (B.77) can be simplified to

$$G(j\omega) = \frac{j2\delta u}{1 + j2\delta u - u^2}$$
The magnitude $|G(j\omega)|$ can be found from

$$|G(j\omega)| = \frac{2\delta u}{[(1 - u^2)^2 + (2\delta u)^2]^{1/2}} \quad \text{(B.78)}$$

The phase angle $\phi$ of $G(j\omega)$ can be found from

$$\phi = \frac{\pi}{2} - \tan^{-1}\left(\frac{2\delta u}{1 - u^2}\right) \quad \text{(B.79)}$$

For low frequencies $u << 1$,

$$|G(j\omega)| \approx 2\delta u$$
$$20 \log_{10}|G(j\omega)| \approx 20 \log_{10}(2\delta u)$$
$$\phi \approx \frac{\pi}{2}$$

Therefore, at low frequencies, the magnitude plot is a straight line with a slope of $+20 \text{ dB/decade}$ or $+6 \text{ dB/octave}$. For $u = 1$, $|G(j\omega)| \approx 1$ only if

$$\delta \approx 1$$
$$20 \log_{10}|G(j\omega)| = 0 \text{ dB}$$
$$\phi = 0$$

For $u >> 1$,

$$|G(j\omega)| \approx \frac{2\delta u}{u^2} = \frac{2\delta}{u}$$
$$20 \log_{10}|G(j\omega)| \approx 20 \log_{10}(2\delta) - 20 \log_{10}(u) \approx -20 \log_{10}(u)$$
$$\phi \approx -\frac{\pi}{2}$$

Therefore, at high frequencies, the magnitude plot is a straight line with a slope of $-20 \text{ dB/decade}$ or $-6 \text{ dB/octave}$. The actual characteristic will differ considerably from the asymptotic lines, and the error will depend on the damping ratio $\delta$. The magnitude and frequency plots of the series $RLC$ circuit are shown in Fig. B.41. The magnitude peaks at $u = 1$.

If the output voltage of the series $RLC$ circuit falls below 70% of its maximum value, the output is not considered the significant value. The cutoff frequency is defined as that value of frequency for which the magnitude of the gain drops to 70.7% of its maximum value $|G(j\omega)|_{\text{max}} = 1$. Thus, at cutoff frequencies, Eq. (B.78) gives

$$|G(j\omega)| = \frac{2\delta u}{[(1 - u^2)^2 + (2\delta u)^2]^{1/2}} = 0.707 = \frac{1}{\sqrt{2}}$$

or

$$\sqrt{2}(2\delta u) = [(1 - u^2)^2 + (2\delta u)^2]^{1/2}$$
Squaring both sides yields
\[ 2(2\delta u)^2 = (1 - u^2)^2 + (2\delta u)^2 \]
or \[ (2\delta u)^2 = (1 - u^2)^2 \tag{B.80} \]
The possible solutions of Eq. (B.80) are
\[ 2\delta u_1 = 1 - u_1^2 \]
\[ u_1^2 + 2\delta u_1 - 1 = 0 \tag{B.81} \]
\[ 2\delta u_2 = -(1 - u_2^2) = u_2^2 - 1 \]
\[ u_2^2 - 2\delta u_2 - 1 = 0 \tag{B.82} \]
Solving Eq. (B.82) yields
\[ u_2 = \delta \pm \sqrt{1 + \delta^2} \]
Since the frequency cannot be negative, the upper cutoff frequency ratio \( u_2 \) is given by
\[ u_2 = \delta + \sqrt{1 + \delta^2} \tag{B.83} \]
and the upper cutoff frequency \( \omega_2 \) is
\[ \omega_2 = u_2\omega_n = \omega_n(\delta + \sqrt{1 + \delta^2}) \tag{B.84} \]
Solving Eq. (B.81) yields

\[ u_1 = -\delta \pm \sqrt{1 + \delta^2} \]

which will give both positive and negative values of \( u_1 \). Since the frequency cannot be negative, the lower cutoff frequency ratio \( u_1 \) is given by

\[ u_1 = -\delta + \sqrt{1 + \delta^2} \quad \text{(B.85)} \]

and the lower cutoff frequency \( \omega_1 \) is

\[ \omega_1 = u_1 \omega_n = \omega_n(-\delta + \sqrt{1 + \delta^2}) \quad \text{(B.86)} \]

The bandwidth (BW) of an amplifier, which is defined as the range of frequencies over which the gain remains almost constant within 3 dB (29.3%) of its maximum value, is thus the difference between the cutoff frequencies. Therefore, the bandwidth \( BW_s \) of a series resonant circuit can be found from

\[ BW_s = \omega_2 - \omega_1 = \omega_n(u_2 - u_1) = 2\delta\omega_n = \frac{R}{L} \quad \text{(in rad/s)} \quad \text{(B.87)} \]

\[ BW_s = f_2 - f_1 = \frac{1}{2\pi} \frac{R}{L} \quad \text{(in Hz)} \quad \text{(B.88)} \]

From Eq. (B.55), \( R \ L = 2\pi f_n \ Q_s \). Thus, Eq. (B.88) can be rewritten as

\[ BW_s = \frac{1}{2\pi} \frac{R}{L} = \frac{1}{2\pi} \frac{2\pi f_n}{Q_s} = \frac{f_n}{Q_s} \quad \text{(B.89)} \]

which shows that the larger the value of \( Q_s \), the smaller the value of bandwidth \( BW_s \), and vice versa. It can be shown that Eq. (B.89) can be also applied to calculate the bandwidth \( BW_p \) of a parallel resonant circuit; that is,

\[ BW_p = \frac{f_p}{Q_p} \quad \text{(B.90)} \]

where \( f_p \) is the parallel resonant frequency in Eq. (B.63) and \( Q_p \) is the quality factor of a parallel resonant circuit in Eq. (B.64).

**EXAMPLE B.16**

**Finding the frequency response of a series RLC circuit**  
The series RLC circuit in Fig. B.40 has \( R = 50 \Omega \), \( L = 4 \text{ mH} \), and \( C = 0.15 \mu F \).

(a) Determine the series resonant frequency \( f_n \), the damping ratio \( \delta \), the quality factor \( Q_s \), the cutoff frequencies, and the bandwidth \( BW_s \).

(b) Use PSpice/SPICE to plot the magnitude and phase angle of the output voltage for \( R = 50 \Omega \), 100 \( \Omega \), and 200 \( \Omega \). The frequency \( f \) varies from 100 Hz to 1 MHz. Assume \( V_m = 1 \text{ V peak AC} \).
**SOLUTION**

(a) \( R = 50 \, \Omega \), \( L = 4 \, \text{mH} \), and \( C = 0.15 \, \mu\text{F} \), so

\[
\omega_n = \frac{1}{\sqrt{LC}} = \frac{10^5}{\sqrt{4 \times 1.5}} = 40,825 \, \text{rad/s}
\]

The series resonant frequency is

\[
f_n = \frac{\omega_n}{2\pi} = \frac{40,825}{2\pi} = 6497.5 \, \text{Hz}
\]

Since \( \alpha = R \cdot (2L) = 50 \times (2 \times 4 \times 10^{-3}) = 6250 \), the damping ratio is

\[
\delta = \frac{\alpha}{\omega_n} = \frac{6250}{40,825} = 0.1531
\]

From Eq. (B.55),

\[
Q_s = \frac{R}{\omega_n} = \frac{40,825 \times 4 \times 10^{-3}}{50} = 3.266
\]

For the lower cutoff frequency, Eqs. (B.85) and (B.86) give

\[
u_1 = -\delta + \sqrt{1 + \delta^2} = -0.1531 + \sqrt{1 + 0.1531^2} = 0.85855
\]

\[
\omega_1 = u_1\omega_n = 0.85855 \times 40,825 = 35,050.4 \, \text{rad/s}
\]

Thus, \( f_1 = 35,050.4/2\pi = 5578 \, \text{Hz} \). For the upper cutoff frequency, Eqs. (B.83) and (B.84) give

\[
u_2 = \delta + \sqrt{1 + \delta^2} = 0.1531 + \sqrt{1 + 0.1531^2} = 1.16475
\]

\[
\omega_2 = u_2\omega_n = 1.16475 \times 40,825 = 47,551 \, \text{rad/s}
\]

Thus, \( f_2 = 47,551/2\pi = 7568 \, \text{Hz} \). From Eq. (B.89), the bandwidth is

\[
\text{BW}_s = f_2 - f_1 = \frac{f_n}{Q_s} = \frac{6497.5}{3.266} = 1989.4 \, \text{Hz}
\]

**FIGURE B.42** Series \( RLC \) circuit for PSpice simulation
B.10.4 Second-Order Parallel RLC Circuits

A parallel RLC circuit is shown in Fig. B.44. The output voltage $v_o$ is taken across the parallel combination of $R$, $L$, and $C$. The transfer function $G(s) = V_o(s)/I_s(s)$ in Laplace’s domain of $s$ is the equivalent impedance $Z(s)$.

The function

$$\frac{1}{Z(s)} = \frac{1}{R} + \frac{1}{sL} + sC = \frac{sL + R + s^2LC}{sRL}$$

$$= \frac{s^2 + (s/RC) + (1/LC)}{s/C} \text{ (in siemens, or mhos)}$$

![Parallel RLC circuit diagram](image)

**FIGURE B.44** Parallel RLC circuit

(b) The series RLC circuit for PSpice simulation is shown in Fig. B.42. The list of the circuit file is as follows.

Example B.16 Frequency Response of Series RLC Circuit

```plaintext
.PARAM RVAL = 50
.STEP PARAM RVAL LIST 50 100 200
Vm 1 0 AC IV ; AC input of 1 V peak
L 1 2 4MH
C 2 3 0.15UF
R 3 0 {RVAL}
.AC DEC 100 100HZ 1MEGHZ ; AC analysis from f = 100 Hz to 1 MHz
; with a decade change and 100 points per decade
.PROBE
.END
```

The PSpice plots of the magnitude and phase angle (using EXB-16.SCH) are shown in Fig. B.43. The plot for $R = 50 \Omega \text{ gives } f_1 = 5578 \text{ Hz, } f_2 = 7568 \text{ Hz, } f_3 = 6457 \text{ Hz, and BWs } f_2 - f_1 = 1990 \text{ Hz.}$

**FIGURE B.43** Frequency response plots for Example B.16
gives the transfer function $G(s)$ as

$$G(s) = \frac{V_o(s)}{I_o(s)} = \frac{s/C}{s^2 + (s/RC) + (1/LC)}$$

$$= R \frac{s/RC}{s^2 + (s/RC) + (1/LC)} \quad \text{(in ohms)} \quad (B.91)$$

Defining $\omega_n = \sqrt{LC}$ as the resonant frequency in rad/s and $\alpha = 1/(2RC)$ as the damping factor, we can write Eq. (B.91) as

$$G(s) = R \frac{2\alpha s}{s^2 + 2\alpha s + \omega_n^2} \quad \text{(in ohms)} \quad (B.92)$$

Let us define

$$\delta = \frac{\alpha}{\omega_n} = \frac{1}{2RC} \times \sqrt{LC} = \frac{1}{2R} \sqrt{\frac{L}{C}} \quad (B.93)$$

as the damping ratio. Then Eq. (B.92) becomes

$$G(s) = R \frac{\delta \omega_n s}{s^2 + 2\delta \omega_n s + \omega_n^2} \quad \text{(in ohms)} \quad (B.94)$$

where $\delta < 1$. (Note that $\delta$ is not necessarily less than 1, but has been assumed less than 1 for this discussion.) The right-hand side of Eq. (B.94) is $R/2$ multiplied by Eq. (B.76). Following the development of Eqs. (B.78) and (B.79), we find the magnitude $|G(j\omega)|$ as

$$|G(j\omega)| = \frac{2\delta u R}{[(1 - u^2)^2 + (2\delta u)^2]^{1/2}} \quad \text{(in ohms)} \quad (B.95)$$

and the phase angle $\phi$ of $G(j\omega)$ as

$$\phi = \frac{\pi}{2} - \tan^{-1}\left(\frac{2\delta u}{1 - u^2}\right) \quad (B.96)$$

The magnitude and frequency plots of a parallel RLC circuit are shown in Fig. B.45. The maximum value $|G(j\omega)|_{\text{max}} = |Z(j\omega)|_{\text{max}} = 1$. At the cutoff frequencies, the magnitude of the gain drops to 70.7% of its maximum value $R$. Thus, Eq. (B.95) gives

$$|G(j\omega)| = \frac{2\delta u R}{[(1 - u^2)^2 + (2\delta u)^2]^{1/2}} = 0.707R = \frac{R}{\sqrt{2}}$$

or

$$\sqrt{2}(2\delta u) = [(1 - u^2)^2 + (2\delta u)^2]^{1/2}$$

Squaring both sides yields

$$2(2\delta u)^2 = (1 - u^2)^2 + (2\delta u)^2$$

or

$$2(2\delta u)^2 = (1 - u^2)^2 \quad (B.97)$$
which is the same as Eq. (B.80). Equations (B.81) to (B.86) can be applied to find \( \omega_1 \) and \( \omega_2 \). Then the bandwidth \( BW_p \) of the parallel resonant circuit can be found from

\[
BW_p = \omega_p - \omega_1 = \omega_n(u_2 - u_1) = 2\delta\omega_n
\]

\[
= 2 \frac{1}{2R} \sqrt{\frac{L}{C}} \times \frac{1}{\sqrt{LC}} = \frac{1}{RC} \quad \text{(in rad/s)}
\]

\[
(B.98)
\]

**NOTE** For a parallel circuit, \( BW_p = 1 \) \( RC \) only; for a series circuit, \( BW_s = R/L \).

**EXAMPLE B.17**

**Finding the frequency response of a parallel RLC circuit** The parallel RLC circuit in Fig. B.44 has \( R = 50 \) \( \Omega \), \( L = 4 \) mH, and \( C = 0.15 \) \( \mu F \).

(a) Determine the parallel resonant frequency \( f_p \), the damping ratio \( \delta \), the cutoff frequencies, the bandwidth \( BW_p \), and the quality factor \( Q_p \) of the circuit.

(b) Use PSpice/SPICE to plot the magnitude and phase angle of the output voltage for \( R = 50 \) \( \Omega \), \( 100 \) \( \Omega \), and \( 200 \) \( \Omega \). The frequency \( f \) varies from 100 Hz to 100 kHz. Assume \( I_m = 1 \) A peak AC.

**SOLUTION**

(a) \( R = 50 \) \( \Omega \), \( L = 4 \) mH, \( C = 0.15 \) \( \mu F \), and \( I_m = 1 \) A peak AC, so

\[
\omega_n = \frac{1}{\sqrt{LC}} = \frac{10^5}{\sqrt{4 \times 1.5}} = 40,825 \text{ rad/s}
\]
The parallel resonant frequency is

\[ f_p = \frac{\omega_n}{2\pi} = \frac{40,825}{2\pi} = 6497.5 \text{ Hz} \]

Since \( \alpha = 1 \) \((2RC) = 1 \ (2 \times 50 \times 0.15 \times 10^{-6}) = 66.667 \times 10^3\), the damping ratio is

\[ \delta = \frac{\alpha}{\omega_n} = 66.667 \times \frac{10^3}{40,825} = 1.633 \]

For the lower cutoff frequency, Eqs. (B.85) and (B.86) give

\[ u_1 = -\delta + \sqrt{1 + \delta^2} = -1.633 + \sqrt{1 + 1.633^2} = 0.28186 \]
\[ \omega_1 = u_1 \omega_n = 0.28186 \times 40,825 = 11,507 \text{ rad/s} \]

Thus, \( f_1 = 11,507/2\pi = 1831 \text{ Hz} \). For the upper cutoff frequency, Eqs. (B.83) and (B.84) give

\[ u_2 = \delta + \sqrt{1 + \delta^2} = 1.633 + \sqrt{1 + 1.633^2} = 3.54786 \]
\[ \omega_2 = u_2 \omega_n = 3.54786 \times 40,825 = 144,841 \text{ rad/s} \]

Thus, \( f_2 = 144,841/2\pi = 23,052 \text{ Hz} \). From Eq. (B.98), the bandwidth is

\[ BW_p = \frac{f_2 - f_1}{RC} = \frac{1}{50 \times 0.15 \times 10^{-6}} = 133,333.3 \text{ rad/s, or 21,220 Hz} \]

Using Eq. (B.90), we can find the quality factor:

\[ Q_p = \frac{f_p}{BW_p} = \frac{6497.5}{21,220} = 0.3062 \]

Parameters:

RVAL  50

\[ I_m = 1 \text{ A} \]
\[ I_{AC} = 4 \text{ mH} \]
\[ C = 0.15 \mu\text{F} \]

**FIGURE B.46** Parallel RLC circuit for PSpice simulation
B.11 Time Constants of First-Order Circuits

We have seen that the transient and frequency responses of first-order circuits depend on their time constants. The time constant of an $RC$ network is $\tau = RC$, and that of an $RL$ circuit is $\tau = L/R$. Many circuits have more than two components. The effective time constant can be determined by finding the effective resistance and capacitance of the circuit. The steps in finding the effective time constant are as follows:

**Step 1.** Set the voltage source(s) to zero and the current source(s) as an open circuit.

**Step 2.** If there is more than one capacitor (or inductor) and only one resistor, find the effective capacitance or inductance seen by the resistor.

**Step 3.** If there is more than one resistor and only one capacitive (or inductive) element, find the effective resistance seen by the capacitor (or inductor).

(b) The parallel $RLC$ circuit for PSpice simulation is shown in Fig. B.46. The list of the circuit file is as follows.

```
Example B.17 Frequency Response of a Parallel RLC Circuit
.PARAM RVAL = 50
.STEP PARAM RVAL LIST 50 100 200
.IM 0 1 AC 1A ; ac input of 1 V peak
.L 1 0 4MH
.C 1 0 0.15UF
.R 1 0 {RVAL}
.AC DEC 100 100HZ 1MEGHZ
.PROBE
.END
```

The PSpice plots of the magnitude and phase angle (using EXB-17.SCH) are shown in Fig. B.47. The plot for $R = 50 \, \Omega$ gives $f_1 = 1834 \, \text{Hz}, f_2 = 22.56 \, \text{kHz}, f_p = 6457 \, \text{Hz}$, and $BW_p = f_2 - f_1 = 20.726 \, \text{Hz}$.

![FIGURE B.47 Frequency response plots for Example B.17](image)
EXAMPLE B.18

Finding the effective time constant  The circuit in Fig. B.48 has $R_1 = R_2 = R_3 = 6 \, \text{k}\Omega$ and $C = 0.1 \, \mu\text{F}$. Determine (a) the effective time constant $\tau$, (b) the cutoff frequency $\omega_c$, and (c) the bandwidth $\text{BW}$.

SOLUTION

If the source is shorted, the effective resistance seen by capacitor $C$ is the parallel combination of $R_1$, $R_2$, and $R_3$. The effective resistance $R$ is given by

$$\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} \quad \text{or} \quad R = \frac{R_1}{3} = \frac{6 \, \text{k}\Omega}{3} = 2 \, \text{k}\Omega$$

(a) The effective time constant is

$$\tau = CR = 2 \, \text{k}\Omega \times 0.1 \, \mu\text{F} = 0.2 \, \text{ms}$$

(b) The cutoff frequency is

$$\omega_c = \frac{1}{\tau} = \frac{1}{0.2 \, \text{ms}} = 5000 \, \text{rad/s}, \text{ or } 795.8 \, \text{Hz}$$

(c) At $\omega = 0$, capacitor $C$ is open-circuited, and the output voltage has a finite value. At a high frequency, tending to infinity ($\omega = \infty$), capacitor $C$ is short-circuited, and the output voltage becomes zero. This is a low-pass circuit with $f_1 = 0$ and $f_2 = f_0 = 795.8 \, \text{Hz}$. Thus, the bandwidth is

$$\text{BW} = f_2 - f_1 = 795.8 \, \text{Hz}$$

EXAMPLE B.19

Finding the effective time constant  The circuit in Fig. B.49 has $R_1 = R_2 = R_3 = 10 \, \text{k}\Omega$ and $C_1 = 0.1 \, \mu\text{F}$. Determine (a) the effective time constant $\tau$ and (b) the cutoff frequency $\omega_c$.

SOLUTION

If the source is shorted, the effective resistance is the sum of $R_1$ and $(R_2 \parallel R_3)$; that is,

$$R = R_1 + (R_2 \parallel R_3) = 10 \, \text{k} + 10 \, \text{k} \parallel 10 \, \text{k} = 15 \, \text{k}\Omega$$
(a) The effective time constant is
\[ \tau = CR = 15 \, \text{k}\Omega \times 0.1 \, \mu\text{F} = 1.5 \, \text{ms} \]

(b) The cutoff frequency is
\[ \omega_0 = \frac{1}{\tau} = \frac{1}{1.5 \, \text{ms}} = 667 \, \text{rad/s}, \text{ or } 106 \, \text{Hz} \]
A bipolar transistor can be represented by hybrid \((h)\) parameters. If \(i_b, v_{be}, i_c,\) and \(v_{ce}\) are the small-signal variables of a transistor, as shown in Fig. C.1(a), they are related to the hybrid parameters as follows:

\[
\begin{align*}
v_{bc} &= h_{ie} i_b + h_{re} v_{ce} \\
i_c &= h_{fe} i_b + h_{oe} v_{ce}
\end{align*}
\]

where \(h_{ie}\) is the *short-circuit input resistance* (or simply the *input resistance*), defined by

\[
h_{ie} = \left. \frac{v_{bc}}{i_b} \right|_{v_{ce}=0} \text{ (in ohms)}
\]

\(h_{re}\) is the *open-circuit reverse voltage ratio* (or the *voltage-feedback ratio*), defined by

\[
h_{re} = \left. \frac{v_{bc}}{v_{ce}} \right|_{i_b=0} \text{ (dimensionless)}
\]

\(h_{fe}\) is the *short-circuit forward-transfer current ratio* (or the *small-signal current gain*), defined by

\[
h_{fe} = \left. \frac{i_c}{i_b} \right|_{v_{ce}=0} \text{ (dimensionless)}
\]

and \(h_{oe}\) is the *open-circuit output admittance* (or simply the *output admittance*), defined by

\[
h_{oe} = \left. \frac{i_c}{v_{ce}} \right|_{i_b=0} \text{ (in siemens)}
\]

The low-frequency hybrid model is shown in Fig. C.1(b). The input has a voltage-controlled voltage source in which the controlling voltage is the output voltage. The output circuit has a current-controlled current source in which the controlling current is the input current. The subscript \(e\) on the \(h\) parameters indicates that these hybrid parameters are derived for a common-emitter configuration. It is also possible to have common-base and common-collector configurations.

BJT manufacturers specify the common-emitter hybrid parameters. The parameter \(h_{re}\), which takes into account the effect of \(v_{CE}\) on \(i_B\), is very small, with a typical value of \(0.5 \times 10^{-4}\). The parameter \(h_{oe}\), which represents the admittance of the CE junction, is also very small; its value is typically \(10^{-6}\) S. The parameters \(h_{re}\) and \(h_{oe}\) can often be omitted from the circuit model without significant loss of accuracy, especially for hand calculations.

**FIGURE C.1** Low-frequency hybrid model
The $\pi$-model parameters of Fig. 8.12(a) can be related to the $h$ parameters by applying short-circuit and open-circuit tests. The parameters $h_{ie}$ and $h_{fe}$, which are short-circuit parameters, can be derived in terms of $\pi$-model parameters from Fig. C.2(a) by shorting the collector-to-emitter terminals.

\[ h_{ie} = \frac{v_{be}}{i_b \mid v_{ce}=0} = r_\pi \| r_\mu = \frac{r_\pi r_\mu}{r_\pi + r_\mu} \]  
\[ \text{(C.7)} \]

\[ h_{fe} = \frac{i_c}{i_b \mid v_{ce}=0} = \frac{g_m v_{be}}{v_{be}/r_\pi} = g_m r_\pi \]  
\[ \text{(C.8)} \]

As shown in Fig. C.2(b), $h_{oc}$ and $h_{re}$ can be determined by open-circuiting the base-to-emitter terminals and then applying the voltage divider rule.

\[ h_{re} = \frac{v_{be}}{v_x \mid i_b=0} = \frac{r_\pi}{r_\pi + r_\mu} \]  
\[ \text{(C.9)} \]

Summing the currents at the collector node in Fig. C.2(b), we get

\[ i_x = \frac{v_x}{r_o} + g_m \frac{r_\pi v_x}{r_\pi + r_\mu} + \frac{v_x}{r_\pi + r_\mu} = v_x \left[ \frac{1}{r_o} + \frac{g_m r_\pi + 1}{r_\pi + r_\mu} \right] \]

which gives

\[ h_{oc} = \frac{i_x}{v_x} = \frac{1}{r_o} + \frac{g_m r_\pi + 1}{r_\pi + r_\mu} \]  
\[ \text{(C.10)} \]

Using Eq. (C.8), we get

\[ r_\pi = \frac{h_{fe}}{g_m} \]  
\[ \text{(C.11)} \]

Using Eq. (C.7) and $h_{fe} = g_m r_\pi$ from Eq. (C.8), we get

\[ r_\mu = \frac{r_\pi + r_\mu}{r_\pi} h_{ie} = \frac{h_{ie}}{h_{re}} \]  
\[ \text{(C.12)} \]
From Eqs. (C.10) and (C.8) we find

\[
\frac{1}{r_o} = h_{oc} - \frac{g_m r_{\pi} + 1}{r_{\pi} + r_{\mu}} = h_{oc} - \frac{h_{fe} + 1}{r_{\pi} + r_{\mu}} \quad \text{(C.13)}
\]

If \( h_{fe} \gg 1 \) and \( r_{\mu} \gg r_{\pi} \), which is usually the case, Eq. (C.13) can be approximated by

\[
\frac{1}{r_o} \approx h_{oc} - \frac{h_{fe}}{r_{\mu}}
\]

which gives

\[
r_o = \left[ h_{oc} - \frac{h_{fe}}{r_{\mu}} \right]^{-1}
\]

\[
= \frac{V_A}{I_C}
\]

\( r_{\mu} \) is very large, and \( h_{fe} \) is negligibly small. The \( \pi \) model in Fig. 8.12(b) becomes similar to the \( h \) model in Fig. C.2(b) such that \( r_{\pi} = h_{ie}, \beta_t = h_{fe}, \) and \( r_o = h_{oc}. \) Being open-circuited, \( r_{\mu} \) has a high value and can often be neglected.

**EXAMPLE C.1**

Converting hybrid parameters to \( \pi \)-model parameters The \( h \) parameters of a transistor are \( h_{ie} = 1 \) k\( \Omega \), \( h_{fe} = 100, h_{re} = 0.5 \times 10^{-4} \), and \( h_{oc} = 10 \times 10^{-6} \) S. The collector biasing current is \( I_C = 11.62 \) mA. Calculate the small-signal \( \pi \)-model parameters of the transistor. Assume \( V_T = 25.8 \) mV at 25°C.

**SOLUTION**

From Eq. (8.42),

\[
g_m = \frac{I_C}{V_T} = \frac{11.62 \text{ mA}}{25.8 \text{ mV}} = 0.4504 \text{ A/V}
\]

From Eq. (C.11),

\[
r_{\pi} = \frac{h_{fe}}{g_m} = \frac{100}{0.4504} = 222 \text{ } \Omega
\]

From Eq. (C.12),

\[
r_{\mu} = \frac{h_{ie}}{h_{fe}} = \frac{1 \text{ k} \Omega}{0.5 \times 10^{-3}} = 20 \text{ M}\Omega
\]

From Eq. (C.14),

\[
r_o = \left[ h_{oc} - \frac{h_{fe}}{r_{\mu}} \right]^{-1} = \left[ 10 \times 10^{-6} - \frac{100}{20 \times 10^6} \right]^{-1} = 200 \text{ k} \Omega
\]
EXAMPLE C.2

Finding the hybrid-model parameters of the circuit shown in Fig. C.3  The circuit values are $R_1 = 4 \, \text{k} \Omega$, $R_2 = 10 \, \text{k} \Omega$, and $R_I = 16 \, \text{k} \Omega$. Determine the hybrid parameters $h_{ie}$, $h_{re}$, $h_{fe}$, and $h_{oe}$.

SOLUTION

$R_1 = 4 \, \text{k} \Omega$, $R_2 = 6 \, \text{k} \Omega$, and $R_I = 20 \, \text{k} \Omega$. Using Eq. (C.3) and the equivalent circuit as shown in Fig. C.4(a), $h_{ie}$ can be determined from

$$h_{ie} = \left. \frac{v_S}{i_S} \right|_{v_O = 0} = R_1 \| R_I = 4 \, \text{k} \Omega \| 16 \, \text{k} \Omega = 3.2 \, \text{k} \Omega$$

Using Eq. (C.4) and the equivalent circuit shown in Fig. C.4(b), $h_{re}$ can be determined from

$$h_{re} = \left. \frac{v_S}{v_O} \right|_{i_S = 0} = \frac{R_1}{R_1 + R_I} = \frac{4 \, \text{k} \Omega}{4 \, \text{k} \Omega + 16 \, \text{k} \Omega} = 0.2 \, \text{V/V}$$

Using Eq. (C.5) and the equivalent circuit shown in Fig. C.4(c), $h_{fe}$ can be determined from

$$h_{fe} = \left. \frac{i_O}{i_S} \right|_{v_O = 0} = \frac{-R_1}{R_1 + R_I} = \frac{-4 \, \text{k} \Omega}{4 \, \text{k} \Omega + 16 \, \text{k} \Omega} = -0.2 \, \text{A/A}$$

Using Eq. (C.6) and the equivalent circuit shown in Fig. C.4(d), $h_{oe}$ can be determined from

$$h_{oe} = \left. \frac{i_O}{v_O} \right|_{i_S = 0} = \frac{1}{R_1 \| (R_1 + R_I)} = \frac{1}{4 \, \text{k} \Omega (4 \, \text{k} \Omega + 16 \, \text{k} \Omega)} = 1.5 \times 10^{-4} \, \text{V}$$
A transistor is a nonlinear device that can be modeled using the nonlinear characteristics of diodes. The Ebers–Moll model is a large-signal model commonly used for modeling BJTs. One version of the model is based on the assumption of one forward-biased diode and one reverse-biased diode. This arrangement is shown in Fig. D.1 for an npn transistor. This model, referred to as the injection version of the Ebers–Moll model, is valid for active, saturation, and cutoff regions. Under normal operation in the active region, one junction of the BJT is forward biased and the other is reverse biased.

The emitter–base and collector–base diodes can be described using the Shockley diode characteristic of Eq. (4.1):

\[ I_F = I_{ES} \left( \exp \frac{V_{BE}}{V_T} - 1 \right) \]  \hspace{1cm} (D.1)

\[ I_R = I_{CS} \left( \exp \frac{V_{BC}}{V_T} - 1 \right) \]  \hspace{1cm} (D.2)

where \( V_T = kT q = 25.8 \text{ mV at } 25^\circ \text{C} \)

\( I_{ES} \) = reverse saturation current of base–emitter diode

\( I_{CS} \) = reverse saturation current of base–collector diode

Both \( I_{ES} \) and \( I_{CS} \) are temperature dependent. If \( V_{BE} > 0 \), diode \( D_F \) is forward biased and its current \( I_F \) causes a corresponding current \( \alpha_F I_F \). If \( V_{BC} > 0 \), diode \( D_R \) is reverse biased. The subscripts \( F \) and \( R \) are used to designate forward and reverse conditions, respectively. Using Kirchhoff’s current law (KCL) at the emitter and collector terminals, we can write the emitter current \( I_E \) as

\[ I_E = -I_F + \alpha_R I_R \]  \hspace{1cm} (D.3)

and the collector current \( I_C \) as

\[ I_C = \alpha_F I_F - I_R \]  \hspace{1cm} (D.4)

\[ = \alpha_F I_{ES} \left( \exp \frac{V_{BE}}{V_T} - 1 \right) - I_{CS} \left( \exp \frac{V_{BC}}{V_T} - 1 \right) \]

FIGURE D.1  Ebers–Moll injection version model for npn transistor
Appendix D  Ebers–Moll Model of Bipolar Junction Transistors

If $V_{BE} = 0$, $\alpha_R I_{CS} = I_S$ represents the reverse saturation leakage current of diode $D_R$. Similarly, if $V_{BE} = 0$, $\alpha_F I_{ES} = I_S$ represents the reverse saturation leakage current of diode $D_F$. If we assume ideal diodes, the forward and reverse saturation leakage currents are related by

$$\alpha_R I_{CS} = \alpha_F I_{ES} = I_S \quad (D.5)$$

where $I_S$ is known as the transistor saturation current.

The current from the collector to the base with the emitter open-circuited can be found by letting $I_C = I_{CBO}$ and $I_E = 0$. Since the collector–base junction is normally reverse biased, $V_{BC} < 0$ and $|V_{BC}| >> V_T$, $\exp(V_{BC}/V_T) << 1$. With these conditions in Eqs. (D.3) and (D.4), we get

$$0 = -I_{ES}\left(\exp\frac{V_{BE}}{V_T} - 1\right) - \alpha_R I_{CS}$$

$$I_{CBO} = \alpha_F I_{ES}\left(\exp\frac{V_{BE}}{V_T} - 1\right) + I_{CS}$$

Solving these two equations for $I_{CBO}$ yields

$$I_{CBO} = -\alpha_F \alpha_R I_{CS} + I_{CS} = (1 - \alpha_R \alpha_F) I_{CS} = I_{CS} - \alpha_F I_S \quad (D.6)$$

The current from the emitter to the base with the collector open-circuited can be found by letting $I_E = I_{EBO}$ and $I_C = 0$. Since the emitter junction is reverse biased, $V_{BE} < 0$ (i.e., $V_{EB} > 0$) and $|V_{BE}| >> V_T$, $\exp(V_{BE}/V_T) << 1$. Equations (D.3) and (D.4) give

$$I_{EBO} = I_{ES} + \alpha_R I_{CS}\left(\exp\frac{V_{BC}}{V_T} - 1\right)$$

$$0 = -\alpha_F I_{ES} - I_{CS}\left(\exp\frac{V_{BC}}{V_T} - 1\right)$$

Solving these two equations for $I_{EBO}$ gives

$$I_{EBO} = I_{ES} - \alpha_R \alpha_F I_{ES} = (1 - \alpha_R \alpha_F) I_{ES} \quad (D.7)$$

From Eqs. (D.5), (D.6), and (D.7), we get

$$\alpha_F I_{EBO} = I_{ES}(1 - \alpha_R \alpha_F)\alpha_F = \alpha_R I_{CS}(1 - \alpha_R \alpha_F) = \alpha_R I_{CBO} \quad (D.8)$$

Since diode $D_F$ is forward biased and diode $D_R$ is reverse biased, $V_{EB} < V_{BC}$. Thus, $I_{EBO}$ is less than $I_{CBO}$, and $\alpha_F$ is greater than $\alpha_R$. In the active region, diode $D_R$ is reverse biased, and $I_R \approx 0$; that is, $I_E = -I_F$, and $I_C = \alpha_F I_F = -\alpha_F I_E$. Thus, Fig. D.1(a) can be approximated by Fig. D.1(b).

The circuit model of Fig. D.1(a) relates the dependent sources to the diode currents. In circuit analysis, it is convenient to express the current source in a form that is controlled by the terminal currents. Eliminating $[\exp(V_{BE}/V_T) - 1]$ from Eqs. (D.3) and (D.4) and then using Eq. (D.6), we get

$$I_C = -\alpha_F I_E - (1 - \alpha_F \alpha_R) I_{CS}\left(\exp\frac{V_{BC}}{V_T} - 1\right)$$

$$= -\alpha_F I_E - I_{CBO}\left(\exp\frac{V_{BC}}{V_T} - 1\right) \quad (D.9)$$
Similarly, eliminating $[\exp (V_{BC}/V_T) - 1]$ from Eqs. (D.3) and (D.4) and then using Eq. (D.7) gives

$$I_E = -\alpha_R I_C - (1 - \alpha_R \alpha_F) I_{ES} \left( \exp \frac{V_{BE}}{V_T} - 1 \right)$$

$$= -\alpha_R I_C - I_{EBO} \left( \exp \frac{V_{BE}}{V_T} - 1 \right)$$

(D.10)

The circuit model corresponding to Eqs. (D.9) and (D.10) is shown in Fig. D.2(a). The current sources are controlled by collector current $I_C$ and emitter current $I_E$. This model, referred to as the transport version of the Ebers–Moll model, is normally used in computer simulations with PSpice/SPICE. In fact, the linear models of Fig. 8.12 are the approximate versions of the Ebers–Moll model in Fig. D.2(a).

Assuming that $V_{BC} > 0$ and $I_R \approx 0$, $I_C = -\alpha_F I_E$ and Fig. D.2(a) can be approximated by Fig. D.2(b). If we substitute $I_C = -\alpha_F I_E$, Eq. (D.10) becomes

$$I_E = \alpha_R \alpha_F I_E - I_{EBO} \left( \exp \frac{V_{BE}}{V_T} - 1 \right)$$

which relates $I_E$ to $\alpha_R$, $\alpha_F$, and $V_{BE}$ by

$$I_E = \frac{I_{EBO} \exp (V_{BE}/V_T - 1)}{1 - \alpha_R \alpha_F}$$

(D.11)

$I_{ES}$ and $I_{CS}$ are also known as the short-circuit saturation currents, whereas $I_{CBO}$ and $I_{EBO}$ are known as the open-circuit saturation currents. Note that the injection and transport versions of the Ebers–Moll model are interchangeable. Once the parameters of one version are known, the parameters of the other version can be found.
Finding the currents for the Ebers–Moll model of an npn transistor

An npn transistor is biased so that $V_{BE} = 0.3\ V$ and $V_{CE} = 6\ V$. If $\alpha_F = 0.99$, $\alpha_R = 0.90$, and $I_{CBO} = 5\ \text{nA}$, find all the currents for the injection version of the Ebers–Moll model of Fig. D.1(a). Assume thermal voltage $V_T = 25.8\ \text{mV}$.

**SOLUTION**

$V_T = 25.8\ \text{mV} = 0.0258\ V$. Since $V_{BE}$ is positive, the base–emitter junction is forward biased. The voltage at the collector–base junction is

$$V_{CB} = V_{CE} + V_{EB} = V_{CE} - V_{BE} = 6 - 0.3 = 5.7\ V$$

or $V_{BC} = -5.7\ V$

Since $V_{CB}$ is positive, the collector–base junction is reverse biased, and the transistor operates in the active region. From Eq. (D.8),

$$I_{EBO} = \frac{\alpha_R I_{CBO}}{\alpha_F} = \frac{0.9 \times 5\ \text{nA}}{0.99} = 4.545\ \text{nA}$$

From Eq. (D.6),

$$I_{CS} = \frac{I_{CBO}}{1 - \alpha_R \alpha_F} = \frac{5\ \text{nA}}{1 - 0.9 \times 0.99} = 45.87\ \text{nA}$$

From Eq. (D.7),

$$I_{ES} = \frac{I_{EBO}}{1 - \alpha_R \alpha_F} = \frac{4.545\ \text{nA}}{1 - 0.9 \times 0.99} = 41.697\ \text{nA}$$

From Eq. (D.1), the forward diode current is

$$I_F = 41.697 \times 10^{-9} \times \left(\exp \frac{0.3}{0.0258} - 1\right) \approx 4.678\ \text{mA}$$

From Eq. (D.2), the reverse diode current is

$$I_R = 45.87 \times 10^{-9} \times \left(\exp \frac{-5.7}{0.0258} - 1\right) \approx -45.87\ \text{nA}$$

and $\alpha_R I_R \approx -0.9 \times 45.87\ \text{nA} = -41.28\ \text{nA}$

$\alpha_F I_F \approx 0.99 \times 4.678\ \text{mA} = 4.63\ \text{mA}$

$I_E = -I_F + \alpha_R I_R = -4.678 \times 10^{-3} - 41.28 \times 10^{-6} \approx -4.719\ \text{mA}$

$I_C = \alpha_F I_F - I_R = 4.63 \times 10^{-3} + 45.87 \times 10^{-9} \approx 4.63\ \text{mA}$

$I_B = -(I_E + I_C) = -(-4.719 \times 10^{-3} + 4.63 \times 10^{-3}) = 89\ \mu\text{A}$
**EXAMPLE D.2**

**Finding the currents for the Ebers–Moll model of a pnp transistor**  
The model parameters of the pnp transistor in Fig. 8.8(b) are $\alpha_R = 0.9$, $\alpha_F = 0.99$, $I_{ES} = I_{CS} = 45 \text{ nA}$, $V_{EB} = 0.4 \text{ V}$, and $V_{CB} = 0.3 \text{ V}$. Determine (a) the currents for the transport version of the Ebers–Moll model in Fig. D.2(a) and (b) $\beta_F = \beta_{\text{forced}}$. Assume thermal voltage $V_T = 25.8 \text{ mV}$.

**SOLUTION**

For $V_{EB} = 0.4 \text{ V}$ and $V_{CB} = 0.3 \text{ V}$, the collector–base and emitter–base junctions are forward biased. Therefore, the transistor is operated in the saturation region.

(a) For a pnp transistor, all the polarities of voltages and currents are reversed; thus, Eqs. (D.3) and (D.4) become

$$I_E = I_{ES} \left( \exp \frac{V_{EB}}{V_T} - 1 \right) - \alpha_R I_{CS} \left( \exp \frac{V_{CB}}{V_T} - 1 \right)$$

$$= 45 \times 10^{-9} \left( \exp \frac{0.4}{25.8 \text{ mV}} - 1 \right) - 0.9 \times 45 \times 10^{-9} \left( \exp \frac{0.3}{25.8 \text{ mV}} - 1 \right)$$

$$= 243.48 \text{ mA} - 4.54 \text{ mA} = 238.94 \text{ mA}$$

and

$$I_C = -\alpha_F I_{ES} \left( \exp \frac{V_{EB}}{V_T} - 1 \right) + I_{CS} \left( \exp \frac{V_{CB}}{V_T} - 1 \right)$$

$$= -0.99 \times 45 \times 10^{-9} \left( \exp \frac{0.4}{25.8 \text{ mV}} - 1 \right) + 45 \times 10^{-9} \left( \exp \frac{0.3}{25.8 \text{ mV}} - 1 \right)$$

$$= -241.04 \text{ mA} + 5.05 \text{ mA} = -235.99 \text{ mA}$$

Thus, $I_B = -(I_E + I_C) = -(238.94 - 235.99) = -2.95 \text{ mA}$

(b) Since the transistor is operated in the saturation region, the value of the forward current gain becomes less than that of the active region. The forward current gain is known as the forced current gain $\beta_{\text{forced}}$:

$$\beta_{\text{forced}} = \frac{I_C}{I_B} = \frac{235.99 \text{ mA}}{2.95 \text{ mA}} = 80$$

**EXAMPLE D.3**

**Finding the collector–emitter saturation voltage of a BJT**  
An nnp transistor is operated in the saturation region, and its parameters are $\alpha_R = 0.9$, $\alpha_F = 0.989$, $\beta_F = 89.91$, and $V_T = 25.8 \text{ mV}$. Calculate the collector–emitter saturation voltage $V_{CE(sat)}$.

**SOLUTION**

From Eq. (8.6),

$$\beta_F = \frac{\alpha_F}{1 - \alpha_F} = \frac{0.989}{1 - 0.989} = 89.91$$
We know that
\[ V_{BC} = V_{BE} + V_{EC} = V_{BE} - V_{CE} \]

In the saturation region, \( V_{BE} > 0 \) and \( V_{BC} > 0 \); that is,
\[ \exp \left( \frac{V_{BE}}{V_T} \right) \gg 1 \quad \text{and} \quad \exp \left( \frac{V_{BC}}{V_T} \right) \gg 1 \]

Using Eqs. (D.3) and (D.5), we get
\[
I_E = -I_{ES} e^{V_{BE}/V_T} + \alpha_F I_{ES} e^{V_{BC}/V_T} \\
\approx -I_{ES} e^{V_{BE}/V_T} + \alpha_F I_{ES} e^{(V_{BE} - V_{CE})/V_T} \\
\approx -I_{ES} e^{V_{BE}/V_T} \left[ 1 - \alpha_F e^{-V_{CE}/V_T} \right]
\]

But \( I_E + I_C + I_B = 0 \), so \( -I_E = (I_C + I_B) \); that is,
\[ I_C + I_B = I_{ES} e^{V_{BE}/V_T} \left[ 1 - \alpha_F e^{-V_{CE}/V_T} \right] \tag{D.14} \]

Similarly, using Eqs. (D.4) and (D.5), we get
\[
I_C = \alpha_F I_{ES} \left( e^{V_{BE}/V_T} - 1 \right) - I_C \left( e^{V_{BC}/V_T} - 1 \right) \\
= \alpha_F I_{ES} e^{V_{BE}/V_T} \left[ 1 - \frac{1}{\alpha_R} e^{-V_{CE}/V_T} \right] \tag{D.15}
\]

Dividing Eq. (D.14) by Eq. (D.15) yields
\[
\frac{I_C + I_B}{I_C} = 1 + \frac{I_B}{I_C} = \frac{1 - \alpha_F e^{-V_{CE}/V_T}}{\alpha_F [1 - (e^{-V_{CE}/V_T}) / \alpha_R]}
\]

which, after simplification, becomes
\[ V_{CE} = V_T \ln \left\{ \frac{\alpha_F [1 + I_C / I_B (1 - \alpha_R)]}{\alpha_R [\alpha_F + I_C / I_B (\alpha_F - 1)]} \right\} \tag{D.16} \]

In the active region, the base current \( I_B \) is related to the collector current \( I_C \) by \( I_B = I_C / \beta_F \). The saturation region is considered to begin at the point where the forward current gain \( \beta_F \) is 90% of the value in the active region. That is, \( \beta_{sat} = \beta_{forced} = 0.9 \beta_F \) and \( I_C = 0.9 I_B \beta_F \). Equation (D.16) gives the collector–emitter saturation voltage as
\[ V_{CE(sat)} = V_T \ln \left\{ \frac{\alpha_F [1 + 0.9 \beta_F (1 - \alpha_R)]}{\alpha_R [\alpha_F + 0.9 \beta_F (\alpha_F - 1)]} \right\} \]
Substituting $\beta_F = \alpha_F (1 - \alpha_F)$ in the denominator gives

$$V_{CE(sat)} = V_T \ln \left[ 1 + \frac{0.9 \beta_F (1 - \alpha_R)}{\alpha_R (1 - 0.9)} \right]$$

(D.17)

For $V_T = 0.0258$ V, $\alpha_R = 0.9$, and $\beta_F = 89.91$, we get

$$V_{CE(sat)} = 0.0258 \times \ln \left[ \frac{1 + 0.9 \times 89.91 \times (1 - 0.9)}{0.9 \times (1 - 0.9)} \right] = 0.119\text{ V}$$
APPENDIX E

PASSIVE COMPONENTS
## E.1 Resistors

### TABLE E.1 Standard values (in ohms) for metal film resistors (tolerance ±1%) and carbon resistors (tolerance ±5% and ±10%)

<table>
<thead>
<tr>
<th>Value (1%)</th>
<th>Value (5%)</th>
<th>Value (10%)</th>
<th>Value (1%)</th>
<th>Value (5%)</th>
<th>Value (10%)</th>
<th>Value (1%)</th>
<th>Value (5%)</th>
<th>Value (10%)</th>
<th>Value (1%)</th>
<th>Value (5%)</th>
<th>Value (10%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10.0</td>
<td>10</td>
<td>10</td>
<td>20.0</td>
<td>20</td>
<td>20</td>
<td>40.2</td>
<td>41.2</td>
<td>52</td>
<td>61.9</td>
<td>62</td>
<td></td>
</tr>
<tr>
<td>10.2</td>
<td>20.5</td>
<td>21.0</td>
<td>21.5</td>
<td>22.1</td>
<td>22</td>
<td>44.2</td>
<td>45.3</td>
<td>54</td>
<td>68.1</td>
<td>68</td>
<td></td>
</tr>
<tr>
<td>10.5</td>
<td>21.5</td>
<td>22.6</td>
<td>23.2</td>
<td>22.6</td>
<td>53</td>
<td>49.9</td>
<td>51.1</td>
<td>56</td>
<td>75.0</td>
<td>75</td>
<td></td>
</tr>
<tr>
<td>10.7</td>
<td>22.1</td>
<td>22.6</td>
<td>23.2</td>
<td>23.2</td>
<td>56</td>
<td>52.3</td>
<td>53.6</td>
<td>58</td>
<td>76.8</td>
<td>76.8</td>
<td></td>
</tr>
<tr>
<td>11.0</td>
<td>22</td>
<td>22</td>
<td>23.7</td>
<td>23.7</td>
<td>57</td>
<td>54.9</td>
<td>56.2</td>
<td>60</td>
<td>80.6</td>
<td>80.6</td>
<td></td>
</tr>
<tr>
<td>11.3</td>
<td>22.6</td>
<td>23.2</td>
<td>24.3</td>
<td>24.3</td>
<td>57</td>
<td>55.1</td>
<td>56.2</td>
<td>62</td>
<td>82.5</td>
<td>82</td>
<td></td>
</tr>
<tr>
<td>11.5</td>
<td>23.2</td>
<td>23.7</td>
<td>24.9</td>
<td>24.9</td>
<td>59</td>
<td>57.6</td>
<td>58.3</td>
<td>64</td>
<td>84.5</td>
<td>84.5</td>
<td></td>
</tr>
<tr>
<td>11.8</td>
<td>24</td>
<td>26.7</td>
<td>26.1</td>
<td>26.7</td>
<td>60</td>
<td>59.0</td>
<td>59.0</td>
<td>66</td>
<td>86.6</td>
<td>86.6</td>
<td></td>
</tr>
<tr>
<td>12.1</td>
<td>25.5</td>
<td>26.7</td>
<td>27.4</td>
<td>27.4</td>
<td>60</td>
<td>60.4</td>
<td>60.4</td>
<td>68</td>
<td>88.7</td>
<td>88.7</td>
<td></td>
</tr>
<tr>
<td>12.4</td>
<td>24.9</td>
<td>27.4</td>
<td>28.0</td>
<td>28.0</td>
<td>62</td>
<td>62.2</td>
<td>62.2</td>
<td>71</td>
<td>90.9</td>
<td>90.9</td>
<td></td>
</tr>
<tr>
<td>12.7</td>
<td>25.5</td>
<td>27.4</td>
<td>28.7</td>
<td>28.7</td>
<td>64</td>
<td>64.5</td>
<td>64.5</td>
<td>74</td>
<td>93.1</td>
<td>93.1</td>
<td></td>
</tr>
<tr>
<td>13.0</td>
<td>26.1</td>
<td>28.0</td>
<td>29.4</td>
<td>29.4</td>
<td>66</td>
<td>66.5</td>
<td>66.5</td>
<td>77</td>
<td>95.3</td>
<td>95.3</td>
<td></td>
</tr>
<tr>
<td>13.3</td>
<td>26.7</td>
<td>28.7</td>
<td>30.1</td>
<td>30.1</td>
<td>68</td>
<td>68.7</td>
<td>68.7</td>
<td></td>
<td>97.6</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13.7</td>
<td>27.4</td>
<td>30.1</td>
<td>30.9</td>
<td>30.9</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14.0</td>
<td>28.0</td>
<td>30.9</td>
<td>31.6</td>
<td>31.6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14.3</td>
<td>28.7</td>
<td>31.6</td>
<td>32.4</td>
<td>32.4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14.7</td>
<td>29.4</td>
<td>32.4</td>
<td>33.2</td>
<td>33.2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15.0</td>
<td>30.1</td>
<td>33.2</td>
<td>34.0</td>
<td>34.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15.4</td>
<td>30.9</td>
<td>34.0</td>
<td>34.8</td>
<td>34.8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15.8</td>
<td>31.6</td>
<td>34.8</td>
<td>35.7</td>
<td>35.7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16.2</td>
<td>32.4</td>
<td>35.7</td>
<td>36.5</td>
<td>36.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16.5</td>
<td>33.2</td>
<td>36.5</td>
<td>37.4</td>
<td>37.4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16.9</td>
<td>34.0</td>
<td>37.4</td>
<td>38.3</td>
<td>38.3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17.4</td>
<td>34.8</td>
<td>38.3</td>
<td>39.2</td>
<td>39.2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17.8</td>
<td>35.7</td>
<td>39.2</td>
<td>39.3</td>
<td>39.3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18.2</td>
<td>36.5</td>
<td>39.3</td>
<td>39.6</td>
<td>39.6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18.7</td>
<td>37.4</td>
<td>39.6</td>
<td>39.9</td>
<td>39.9</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19.1</td>
<td>38.3</td>
<td>39.9</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19.6</td>
<td>39.2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
The available values for carbon are $1 \Omega \leq R \leq 100 \text{ M}\Omega$ and for metal film are $10 \Omega \leq R \leq 10 \text{ M}\Omega$. The available values are obtained by multiplying the sequence number by a power of 10 (i.e., $10^{-1}$, $10^0$, $10^1$, $10^2$, $10^3$, and so on).

**TABLE E.2** Standard values (in ohms) for wire-wound resistors (tolerance ±5%)

<table>
<thead>
<tr>
<th>Value</th>
<th>0.008</th>
<th>0.01</th>
<th>0.02</th>
<th>0.03</th>
<th>0.05</th>
<th>0.1</th>
<th>0.15</th>
<th>0.2</th>
<th>0.26</th>
<th>0.3</th>
<th>0.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.75</td>
<td>7.5</td>
<td>72</td>
<td>62</td>
<td>35</td>
<td>2.5</td>
<td>4.0</td>
<td>1.5</td>
<td>2.0</td>
<td>5.0</td>
<td>6.0</td>
<td>7.0</td>
</tr>
<tr>
<td>0.8</td>
<td>8</td>
<td>63</td>
<td>54</td>
<td>36</td>
<td>2.6</td>
<td>4.2</td>
<td>1.6</td>
<td>2.1</td>
<td>5.2</td>
<td>6.2</td>
<td>8.0</td>
</tr>
<tr>
<td>0.1</td>
<td>0.75</td>
<td>7.2</td>
<td>6.2</td>
<td>3.4</td>
<td>2.5</td>
<td>3.8</td>
<td>1.4</td>
<td>1.6</td>
<td>4.7</td>
<td>5.7</td>
<td>7.0</td>
</tr>
<tr>
<td>0.01</td>
<td>0.8</td>
<td>8</td>
<td>6.3</td>
<td>3.5</td>
<td>2.6</td>
<td>3.8</td>
<td>1.6</td>
<td>2.1</td>
<td>5.2</td>
<td>6.2</td>
<td>8.0</td>
</tr>
<tr>
<td>0.02</td>
<td>0.1</td>
<td>0.75</td>
<td>7.2</td>
<td>6.2</td>
<td>3.4</td>
<td>2.5</td>
<td>3.8</td>
<td>1.4</td>
<td>1.6</td>
<td>4.7</td>
<td>5.7</td>
</tr>
<tr>
<td>0.03</td>
<td>0.02</td>
<td>0.1</td>
<td>0.75</td>
<td>7.2</td>
<td>6.2</td>
<td>3.4</td>
<td>2.5</td>
<td>3.8</td>
<td>1.4</td>
<td>1.6</td>
<td>4.7</td>
</tr>
<tr>
<td>0.05</td>
<td>0.05</td>
<td>0.02</td>
<td>0.1</td>
<td>0.75</td>
<td>7.2</td>
<td>6.2</td>
<td>3.4</td>
<td>2.5</td>
<td>3.8</td>
<td>1.4</td>
<td>1.6</td>
</tr>
<tr>
<td>0.1</td>
<td>0.1</td>
<td>0.01</td>
<td>0.01</td>
<td>0.75</td>
<td>7.2</td>
<td>6.2</td>
<td>3.4</td>
<td>2.5</td>
<td>3.8</td>
<td>1.4</td>
<td>1.6</td>
</tr>
<tr>
<td>0.15</td>
<td>0.15</td>
<td>0.02</td>
<td>0.1</td>
<td>0.02</td>
<td>0.1</td>
<td>0.01</td>
<td>0.01</td>
<td>0.75</td>
<td>7.2</td>
<td>6.2</td>
<td>3.4</td>
</tr>
<tr>
<td>0.2</td>
<td>0.2</td>
<td>0.15</td>
<td>0.15</td>
<td>0.02</td>
<td>0.1</td>
<td>0.01</td>
<td>0.01</td>
<td>0.75</td>
<td>7.2</td>
<td>6.2</td>
<td>3.4</td>
</tr>
<tr>
<td>0.26</td>
<td>0.26</td>
<td>0.15</td>
<td>0.15</td>
<td>0.02</td>
<td>0.1</td>
<td>0.01</td>
<td>0.01</td>
<td>0.75</td>
<td>7.2</td>
<td>6.2</td>
<td>3.4</td>
</tr>
<tr>
<td>0.3</td>
<td>0.3</td>
<td>0.26</td>
<td>0.26</td>
<td>0.15</td>
<td>0.15</td>
<td>0.02</td>
<td>0.01</td>
<td>0.01</td>
<td>0.75</td>
<td>7.2</td>
<td>6.2</td>
</tr>
<tr>
<td>0.5</td>
<td>0.5</td>
<td>0.3</td>
<td>0.26</td>
<td>0.26</td>
<td>0.15</td>
<td>0.15</td>
<td>0.02</td>
<td>0.01</td>
<td>0.01</td>
<td>0.75</td>
<td>7.2</td>
</tr>
</tbody>
</table>

**TABLE E.3** Power ratings for resistors

<table>
<thead>
<tr>
<th>Type</th>
<th>Tolerance</th>
<th>Power Rating</th>
</tr>
</thead>
<tbody>
<tr>
<td>Carbon resistors</td>
<td>5% and 10%</td>
<td>$\frac{1}{8}$ W, $\frac{1}{4}$ W, $\frac{1}{2}$ W, 1 W, 2 W, 2 W, 5 W, 12 W, 25 W, 50 W, 100 W, 225 W</td>
</tr>
<tr>
<td>Metal film resistors</td>
<td>1%</td>
<td>$\frac{1}{8}$ W, $\frac{1}{4}$ W, $\frac{1}{2}$ W, 1 W, 2 W, 2 W, 5 W, 12 W, 25 W, 50 W, 100 W, 225 W</td>
</tr>
<tr>
<td>Wire-wound resistors</td>
<td>5%</td>
<td>$\frac{1}{8}$ W, $\frac{1}{4}$ W, $\frac{1}{2}$ W, 1 W, 2 W, 2 W, 5 W, 12 W, 25 W, 50 W, 100 W, 225 W</td>
</tr>
</tbody>
</table>
### Appendix E Passive Components

#### TABLE E.4 Color code for resistors

<table>
<thead>
<tr>
<th>Digit</th>
<th>Digit</th>
<th>Multiplier</th>
<th>Tolerance</th>
<th>Color</th>
<th>Digit</th>
<th>Digit</th>
<th>Digit</th>
<th>Multiplier</th>
<th>Tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>$10^0 = 1$</td>
<td>—</td>
<td>Black</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>$10^0 = 1$</td>
<td>—</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>$10^1 = 10$</td>
<td>—</td>
<td>Brown</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>$10^1 = 10$</td>
<td>±1%</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>$10^2 = 100$</td>
<td>—</td>
<td>Red</td>
<td>2</td>
<td>2</td>
<td>2</td>
<td>$10^2 = 100$</td>
<td>—</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>$10^3 = 1 k$</td>
<td>—</td>
<td>Orange</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>$10^3 = 1 k$</td>
<td>—</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>$10^4 = 10 k$</td>
<td>—</td>
<td>Yellow</td>
<td>4</td>
<td>4</td>
<td>4</td>
<td>$10^4 = 10 k$</td>
<td>—</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>$10^5 = 100 k$</td>
<td>—</td>
<td>Green</td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>$10^5 = 100 k$</td>
<td>—</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>$10^6 = 1 M$</td>
<td>—</td>
<td>Blue</td>
<td>6</td>
<td>6</td>
<td>6</td>
<td>$10^6 = 1 M$</td>
<td>—</td>
</tr>
<tr>
<td>7</td>
<td>7</td>
<td>$10^7 = 10 M$</td>
<td>—</td>
<td>Violet</td>
<td>7</td>
<td>7</td>
<td>7</td>
<td>$10^7 = 10 M$</td>
<td>—</td>
</tr>
<tr>
<td>8</td>
<td>8</td>
<td>—</td>
<td>±5%</td>
<td>Gray</td>
<td>8</td>
<td>8</td>
<td>8</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>9</td>
<td>9</td>
<td>—</td>
<td>±10%</td>
<td>White</td>
<td>9</td>
<td>9</td>
<td>9</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>—</td>
<td>—</td>
<td>—</td>
<td>±20%</td>
<td>Gold</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>Silver</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>No band</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
</tr>
</tbody>
</table>

#### E.2 Potentiometers

#### TABLE E.5 Standard values (in ohms) for carbon composition, linear taper potentiometers (tolerance ±10%; power rating 2.25 W)

<table>
<thead>
<tr>
<th>Value</th>
<th>1 k</th>
<th>10 k</th>
<th>100 k</th>
<th>1 M</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>1.5 k</td>
<td>15 k</td>
<td>150 k</td>
<td>1.5 M</td>
</tr>
<tr>
<td>150</td>
<td>2 k</td>
<td>20 k</td>
<td>200 k</td>
<td>2 M</td>
</tr>
<tr>
<td>200</td>
<td>2.5 k</td>
<td>25 k</td>
<td>250 k</td>
<td>2.5 M</td>
</tr>
<tr>
<td>250</td>
<td>3.5 k</td>
<td>35 k</td>
<td>350 k</td>
<td>3.5 M</td>
</tr>
<tr>
<td>350</td>
<td>5 k</td>
<td>50 k</td>
<td>500 k</td>
<td>5 M</td>
</tr>
<tr>
<td>500</td>
<td>7.5 k</td>
<td>75 k</td>
<td>750 k</td>
<td></td>
</tr>
</tbody>
</table>

#### TABLE E.6 Standard values (in ohms) for conductive plastic potentiometers (tolerance ±10%; power rating 1/2 W)

<table>
<thead>
<tr>
<th>Value</th>
<th>1 k</th>
<th>10 k</th>
<th>100 k</th>
<th>1 M</th>
</tr>
</thead>
<tbody>
<tr>
<td>250</td>
<td>1 k</td>
<td>10 k</td>
<td>100 k</td>
<td>1 M</td>
</tr>
<tr>
<td>2.5 k</td>
<td>25 k</td>
<td>250 k</td>
<td>2.5 M</td>
<td></td>
</tr>
<tr>
<td>5 k</td>
<td>50 k</td>
<td>500 k</td>
<td>5 M</td>
<td></td>
</tr>
</tbody>
</table>

#### TABLE E.7 Standard values (in ohms) for CERMET potentiometers (tolerance ±10%; power rating 1/2 W)

<table>
<thead>
<tr>
<th>Value</th>
<th>1 k</th>
<th>10 k</th>
<th>100 k</th>
<th>1 M</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>1 k</td>
<td>10 k</td>
<td>100 k</td>
<td>1 M</td>
</tr>
<tr>
<td>100</td>
<td>2 k</td>
<td>20 k</td>
<td>200 k</td>
<td>2 M</td>
</tr>
<tr>
<td>200</td>
<td>5 k</td>
<td>50 k</td>
<td>500 k</td>
<td></td>
</tr>
</tbody>
</table>
## E.3 Capacitors

**TABLE E.8** Standard values for polarized aluminum electrolytic capacitors (tolerance $-10\%$ to $+50\%$)

<table>
<thead>
<tr>
<th>Voltage (V)</th>
<th>Capacitance (µF)</th>
<th>Voltage (V)</th>
<th>Capacitance (µF)</th>
<th>Voltage (V)</th>
<th>Capacitance (µF)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>22</td>
<td>25</td>
<td>10</td>
<td>50</td>
<td>0.1</td>
</tr>
<tr>
<td>33</td>
<td>22</td>
<td>33</td>
<td>33</td>
<td>0.1</td>
<td>0.22</td>
</tr>
<tr>
<td>47</td>
<td>47</td>
<td>100</td>
<td>47</td>
<td>0.33</td>
<td>0.47</td>
</tr>
<tr>
<td>100</td>
<td>100</td>
<td>220</td>
<td>100</td>
<td>1.0</td>
<td>2.2</td>
</tr>
<tr>
<td>220</td>
<td>220</td>
<td>330</td>
<td>220</td>
<td>3.3</td>
<td>3.3</td>
</tr>
<tr>
<td>330</td>
<td>330</td>
<td>470</td>
<td>330</td>
<td>4.7</td>
<td>4.7</td>
</tr>
<tr>
<td>470</td>
<td>470</td>
<td>1000</td>
<td>470</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>1000</td>
<td>1000</td>
<td>2200</td>
<td>1000</td>
<td>22</td>
<td>22</td>
</tr>
<tr>
<td>2200</td>
<td>2200</td>
<td>3300</td>
<td>3300</td>
<td>33</td>
<td>33</td>
</tr>
<tr>
<td>3300</td>
<td>3300</td>
<td>4700</td>
<td>3300</td>
<td>47</td>
<td>47</td>
</tr>
<tr>
<td>4700</td>
<td>4700</td>
<td>6800</td>
<td>4700</td>
<td>100</td>
<td>100</td>
</tr>
<tr>
<td>6800</td>
<td>6800</td>
<td>10,000</td>
<td>6800</td>
<td>220</td>
<td>220</td>
</tr>
<tr>
<td>10,000</td>
<td></td>
<td></td>
<td></td>
<td>330</td>
<td>330</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>470</td>
<td>470</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1000</td>
<td>1000</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2200</td>
<td>2200</td>
</tr>
</tbody>
</table>

**TABLE E.9** Standard values for ceramic disc capacitors (tolerance ±10%)

<table>
<thead>
<tr>
<th>Voltage (V)</th>
<th>Capacitance (pF)</th>
</tr>
</thead>
<tbody>
<tr>
<td>200</td>
<td>10</td>
</tr>
<tr>
<td></td>
<td>15</td>
</tr>
<tr>
<td></td>
<td>22</td>
</tr>
<tr>
<td></td>
<td>33</td>
</tr>
<tr>
<td></td>
<td>47</td>
</tr>
<tr>
<td></td>
<td>68</td>
</tr>
<tr>
<td></td>
<td>100</td>
</tr>
<tr>
<td></td>
<td>150</td>
</tr>
<tr>
<td></td>
<td>220</td>
</tr>
<tr>
<td></td>
<td>330</td>
</tr>
<tr>
<td></td>
<td>470</td>
</tr>
<tr>
<td></td>
<td>680</td>
</tr>
<tr>
<td></td>
<td>1000</td>
</tr>
<tr>
<td></td>
<td>1500</td>
</tr>
<tr>
<td></td>
<td>2200</td>
</tr>
<tr>
<td></td>
<td>3300</td>
</tr>
<tr>
<td></td>
<td>4700</td>
</tr>
<tr>
<td></td>
<td>6800</td>
</tr>
<tr>
<td></td>
<td>10,000</td>
</tr>
<tr>
<td></td>
<td>15,000</td>
</tr>
</tbody>
</table>
### TABLE E.10  Standard values for mylar polyester capacitors (tolerance ± 10%)

<table>
<thead>
<tr>
<th>Voltage (V)</th>
<th>Capacitance (µF)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>0.001</td>
</tr>
<tr>
<td></td>
<td>0.0015</td>
</tr>
<tr>
<td></td>
<td>0.0022</td>
</tr>
<tr>
<td></td>
<td>0.0033</td>
</tr>
<tr>
<td></td>
<td>0.0047</td>
</tr>
<tr>
<td></td>
<td>0.0068</td>
</tr>
<tr>
<td></td>
<td>0.0082</td>
</tr>
<tr>
<td></td>
<td>0.01</td>
</tr>
<tr>
<td></td>
<td>0.015</td>
</tr>
<tr>
<td></td>
<td>0.022</td>
</tr>
<tr>
<td></td>
<td>0.027</td>
</tr>
<tr>
<td></td>
<td>0.033</td>
</tr>
<tr>
<td></td>
<td>0.039</td>
</tr>
<tr>
<td></td>
<td>0.047</td>
</tr>
<tr>
<td></td>
<td>0.056</td>
</tr>
<tr>
<td></td>
<td>0.068</td>
</tr>
<tr>
<td></td>
<td>0.082</td>
</tr>
<tr>
<td></td>
<td>0.1</td>
</tr>
<tr>
<td></td>
<td>0.12</td>
</tr>
<tr>
<td></td>
<td>0.15</td>
</tr>
<tr>
<td></td>
<td>0.18</td>
</tr>
<tr>
<td></td>
<td>0.22</td>
</tr>
<tr>
<td></td>
<td>0.27</td>
</tr>
<tr>
<td></td>
<td>0.33</td>
</tr>
<tr>
<td></td>
<td>0.39</td>
</tr>
<tr>
<td></td>
<td>0.47</td>
</tr>
<tr>
<td></td>
<td>0.56</td>
</tr>
<tr>
<td></td>
<td>0.68</td>
</tr>
<tr>
<td></td>
<td>0.82</td>
</tr>
<tr>
<td></td>
<td>1</td>
</tr>
</tbody>
</table>

### TABLE E.11  Standard values for ceramic variable capacitors (tolerance ± 10%)

<table>
<thead>
<tr>
<th>Capacitance (pF)</th>
<th>Voltage (V)</th>
<th>Min</th>
<th>Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>250</td>
<td>1</td>
<td>4.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>2.5</td>
<td>10</td>
<td></td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>18</td>
<td></td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>35</td>
<td></td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>40</td>
<td></td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>50</td>
<td></td>
</tr>
</tbody>
</table>
Mini Design Projects

F.1 Design a circuit that will sum the sensing signals generated by the probes of an electrocardiogram. The output can be expressed in terms of the input signals \( V_A, V_B, V_C, \) and \( V_D \) as follows:
\[
V_O = 1.4V_A + 5V_B + 3V_C + 0.6V_D
\]
The accuracy should be better than 2%. The DC power supplies are \( \pm 15 \) V.

F.2 Design a BJT buffer amplifier to give a midfrequency voltage gain of \( |A_v| = v_L/v_s \approx 1 \) and an input resistance of \( R_{in} = v_s/i_s \geq 50 \) k\( \Omega \). The load resistance is \( R_L = 10 \) k\( \Omega \). Assume a source resistance of \( R_S = 500 \) \( \Omega \) and \( V_{CC} = V_{EE} = 15 \) V.

F.3 Design an NMOS buffer amplifier to give a midfrequency voltage gain of \( |A_v| = v_L/v_s \approx 1 \) and an input resistance of \( R_{in} = v_s/i_s \geq 500 \) k\( \Omega \). The load resistance is \( R_L = 10 \) k\( \Omega \). Assume a source resistance of \( R_S = 500 \) \( \Omega \) and \( V_{DD} = 15 \) V.

F.4 Design a BJT buffer amplifier with an active load to give a midfrequency voltage gain of \( |A_v| = v_L/v_s \approx 1 \) and an output resistance of \( R_{in} = v_s/i_s \geq 50 \) k\( \Omega \). The load resistance is \( R_L = 10 \) k\( \Omega \). Assume a source resistance of \( R_S = 500 \) \( \Omega \) and \( V_{CC} = V_{EE} = 15 \) V.

Medium Design Projects

F.5 Design a DC power supply to electronic equipment from an AC supply of 120 V (rms) \( \pm 10\% \), 60 Hz. The load requires \( \pm 12 \) V \( \pm 5\% \) at 0.5 A. Use discrete devices only and design for minimum expense.

F.6 a. Design a BJT amplifier to give a midfrequency voltage gain of \( |A_v| = v_L/v_s = 20 \pm 5\% \) at a load resistance of \( R_L = 10 \) k\( \Omega \). Assume a source resistance of \( R_S = 500 \) \( \Omega \) and \( V_{DD} = 15 \) V.

b. Modify the design so that the amplifier operates in the frequency range from 10 Hz to 100 kHz.

F.7 a. Design a depletion NMOS amplifier to give a midfrequency voltage gain of \( |A_v| = v_L/v_s = 20 \pm 5\% \) at a load resistance of \( R_L = 10 \) k\( \Omega \). Assume a source resistance of \( R_S = 1.5 \) k\( \Omega \) and \( V_{DD} = 15 \) V.

b. Modify the design so that the amplifier operates in the frequency range from 5 Hz to 50 kHz.

F.8 a. Design a MOSFET amplifier to give a midfrequency voltage gain of \( |A_v| = v_L/v_s = 10 \pm 5\% \) at a load resistance of \( R_L = 10 \) k\( \Omega \). Assume a source resistance of \( R_S = 1.5 \) k\( \Omega \) and \( V_{DD} = 12 \) V.

b. Modify the design so that the amplifier operates in the frequency range from 10 Hz to 50 kHz.

F.9 a. Design an NMOS amplifier with an active load to give a midfrequency voltage gain of \( |A_v| = v_L/v_s \approx 250 \pm 5\% \) at a load resistance of \( R_L = 20 \) k\( \Omega \). Assume a source resistance of \( R_S = 1.5 \) k\( \Omega \) and \( V_{DD} = 15 \) V.

b. Modify the design so that the amplifier operates in the frequency range from 10 Hz to 50 kHz.

Large Design Projects

F.10 The input signal to an amplifier is \( V_s = 2 \) mV, and the amplifier has a source resistance of \( R_S = 1 \) k\( \Omega \).

a. Design a BJT amplifier to give a midfrequency voltage gain \( A_v (= v_L/v_s \) with a load resistance \( R_L = 10 \) k\( \Omega \)) of greater than 650. The input resistance \( R_{in} \) of the amplifier should be greater than 70 k\( \Omega \), and the output resistance \( R_{out} \) should be less than 250 \( \Omega \). The DC supply voltage is \( V_{CC} = 12 \) V.

b. Modify the design so that the amplifier operates in the frequency range from 10 kHz to 80 kHz.

c. Apply feedback and modify the design. The input resistance must be increased by 20 (i.e., \( R_{in} = 2R_{in} \)), and the output resistance must be decreased by 20 (i.e., \( R_{of} = R_{of}/20 \)).

d. Apply feedback and modify the design so that the amplifier oscillates at a frequency of \( f_0 = 20 \) kHz.
F.11 The input signal to an amplifier is $v_s = 2 \text{ mV}$, and the amplifier has a source resistance of $R_S = 1 \text{ k}\Omega$.

a. Design an NMOS amplifier to give a midfrequency voltage gain $A_v$ ($=v_L/v_s$) with a load resistance $R_L = 10 \text{ k}\Omega$ of greater than 450. The input resistance $R_{in}$ of the amplifier should be greater than 500 kΩ, and the output resistance $R_{out}$ should be less than 250 Ω. The DC supply voltage is $V_{DD} = 12 \text{ V}$.

b. Modify the design so that the amplifier operates in the frequency range from 20 kHz to 60 kHz.

c. Apply feedback and modify the design. The input resistance must be increased by 10 (i.e., $R_{in} \leftarrow R_{in} + 10$), and the output resistance must be decreased by 10 (i.e., $R_{out} \leftarrow R_{out} - 10$).

d. Apply feedback and modify the design so that the amplifier oscillates at a frequency of $f_o = 20 \text{ kHz}$.

F.12 Design a BJT operational amplifier to give a large-signal differential gain of $10^3 \pm 10\%$. The input resistance should be higher than 100 kΩ ± 5%, and the output resistance should be less than 210 Ω ± 5%. The op-amp should have a common-mode rejection ratio of CMRR $= 10^4 \pm 10\%$. The unity-gain bandwidth must be better than $10^4 \pm 10\%$.

F.13 Design a multistage BJT amplifier to meet the following specifications: voltage gain, $|A_v| = v_L/v_s = 600 \pm 5\%$ (with load); input resistance, $R_i = v_d/i_s \approx 25 \text{ k}\Omega$; output resistance, $R_o \leq 300 \Omega$; load resistance, $R_L = 25 \text{ k}\Omega$; source resistance, $R_s = 1 \text{ k}\Omega$; DC supply, $V_{CC} = 15 \text{ V}$; input signal, $v_s = 1 \text{ mV}$ to 2 mV (peak sinusoidal), 1 kHz. The amplifier should operate in the frequency range from 10 Hz to 30 kHz.

F.14 Design a multistage MOS amplifier to meet the following specifications: voltage gain, $|A_v| = v_L/v_s = 400 \pm 5\%$ (with load); input resistance, $R_i = v_d/i_s \approx 25 \text{ k}\Omega$; output resistance, $R_o \leq 300 \Omega$; load resistance, $R_L = 25 \text{ k}\Omega$; source resistance, $R_s = 1 \text{ k}\Omega$; DC supply, $V_{DD} = 15 \text{ V}$; input signal, $v_s = 1 \text{ mV}$ to 2 mV (peak sinusoidal), 1 kHz. The amplifier should operate in the frequency range from 100 Hz to 25 kHz.

F.15 Design a differential amplifier to satisfy the following conditions: a differential voltage gain of $A_o \geq 2500$; common-mode rejection ratio of CMRR $\geq 4500$; load of $R_L = 50 \text{ k}\Omega$; coupling capacitor of $C_L = \infty$. Use all BJTs and only one resistance. Use PSpice to verify your design with $v_d = 5 \mu\text{V}$ and $v_c = 10 \text{ mV}$. Supply voltages are $V_{CC} = V_{EE} = 15 \text{ V}$. The amplifier should operate in the frequency range up to 25 kHz.

F.16 Design a differential amplifier to satisfy the following conditions: a differential voltage gain of $A_o \geq 2500$; common-mode rejection ratio of CMRR $\geq 4500$; load of $R_L = 50 \text{ k}\Omega$; coupling capacitor of $C_L = \infty$. Use all MOSFETs. Use PSpice to verify your design with $v_d = 5 \mu\text{V}$ and $v_c = 10 \text{ mV}$. Supply voltages are $V_{DD} = V_{SS} = 15 \text{ V}$. The amplifier should operate in the frequency range up to 25 kHz.
Chapter 1

1.5 80, 60°

1.7 6.66 mS

1.9 (a) \( v_{CE} = (6 + 0.1 \sin 2000 \pi t) \) V,
(b) \( v_{BE} = (700 + 1 \sin 2000 \pi t) \) mV

(b) 100

1.10 (a) \( v_{DS} = (6 + 0.05 \sin 1000 \pi t) \) V,
(b) \( v_{GS} = (3 + 0.002 \sin 1000 \pi t) \) V

Chapter 2

2.1 (a) 42.28 dB, 62.28 dB, 104.56 dB, 50 kΩ
(b) 0.94%  (c) 115.4 mV

2.4 (a) 133.6, 56.84, 7594

2.6 225 Ω

2.13 (a) 191.7, 3333, 639 \times 10^3

2.14 2.33 A

2.16 \( R_o = 47.5 \) kΩ, \( R_1 = 112 \) Ω

2.21 12.25 kΩ

2.34 (a) 62.72 dB, 100.89 dB, 163.6 dB

2.35 (a) 94.77 dB, 97.62 dB, 192.39 dB

2.37 (a) –3.96

2.38 (a) \( R_1 = 1 \) MΩ, \( C_1 = 15.9 \) pF, \( C_2 = 637 \) pF

2.43 (a) \( \omega_1 = 20.48 \) rad s, \( \omega_2 = 1.02 \times 10^6 \) rad s

(b) \( \frac{956.48}{(1 + j \omega/20.48)[1 + j \omega/(1.02 \times 10^6)]} \)

2.44 (a) 20.1 μF  (b) 0.2 μF

2.45 (a) \( A_v = -19.63 \)

Chapter 3

3.1 (a) ±75 μV  (b) ±37.5 pA

3.4 –14 V

3.5 +14 V

3.6 (a) 40 μV  (b) 80 μV  (c) 6.32
(d) \( (8 \pm 50 \times 10^{-6}) \) V

Chapter 4

4.12 (b) 2.14 \times 10^{-14} \) A

4.16 3.56 mA

4.22 (a) 3.58 V  (b) 6.42 mA

4.25 (a) 0.613 V, 18.8 mA  (b) 16.3 Ω
(c) 0.3 V  (d) 2.36 \leq 79.2^\circ \) mV rms

4.29 (a) 6.73 V  (b) 6.715 V, 6.707 V
(c) 2.5 V

4.31 (a) 1317 Ω, 131.7 mW

4.34 (a) \( R_1 = 1398 \) kΩ, \( R_2 = 66.7 \) kΩ

4.37 (a) \( v_{D2} = V_Z, v_{D1} = V - v_{D2} \)
(b) \( V_{D1} = 0.8 \) V, \( V_{D2} = 6.7 \) V
(c) 115.3 mA

Chapter 5

5.2 \( v_O(t) = \frac{169.7}{\pi} + \frac{169.7}{2} \sin 314 t \)

\(- \frac{2 \times 169.7}{\pi} \cos 62 t \)

\( + \frac{2 \times 169.7}{15 \pi} \cos 1256 t \)

\(- \frac{2 \times 169.7}{35 \pi} \cos 1884 t + \cdots \)

5.4 (a) 5.4 V  (b) 2.04 V  (c) 0.577
5.6 (a) \( v_0(t) = \frac{2V_m}{\pi} \times \left[ 1 + 2 \sum_{n=1}^{\infty} \frac{1}{(1 - 4n^2)} \cos n\omega t \right] \), 
\( f_o = 50 \text{ Hz}, \ V_m = 31.1 \text{ V} \)

5.8 (a) 566.5 V  \hspace{1cm} (b) 1.0396 
5.9 (a) \( R_s = 3485 \Omega \)
5.13 (a) 14.9 H 
5.15 (a) 64 \mu F 
5.16 (a) 126 \mu F \hspace{1cm} (b) \( V_{\text{av}} = 169.7 \text{ V}, \ V_{\text{av(no load)}} = 158.7 \text{ V} \)
5.19 (a) 6.47 H 
5.22 (a) 150 V, 0.4714 
5.31 (a) \( R_s = 70 \Omega, 100 k\Omega, 0.1 \mu F \)
5.32 (a) \( R_s = 50 \Omega, 100 k\Omega, 0.1 \mu F \)

Chapter 7

7.16  1.94 V 
7.18 (a) 3.4 mA \hspace{1cm} (b) −5.5 V \hspace{1cm} (c) −2.0 V \hspace{1cm} (d) 3.33 
7.21 (a) 2.9 mA, 3.98 V, −3.34 V \hspace{1cm} (b) 1447 \Omega 
7.24 (a) −0.67 V \hspace{1cm} (b) 89.3 \Omega, 1244 \Omega 
7.28  11.04 mA, 6.96 V, 0.52 V 
7.31  9.05 V, 0.905 mA, 2.95 V 
7.34 (a) 100.5 k\Omega \hspace{1cm} (b) −4.9 \hspace{1cm} (c) 4.63 k\Omega \hspace{1cm} (d) −3.35 
7.37 (a) 10 M\Omega \hspace{1cm} (b) −68.3 \hspace{1cm} (c) 3.13 k\Omega \hspace{1cm} (d) −42 
7.44 (a) 10 M\Omega \hspace{1cm} (b) 0.78 \hspace{1cm} (d) 0.64 
7.47 (a) 4.58 mA \hspace{1cm} V, 58.36 k\Omega \hspace{1cm} (b) 60 M\Omega, 0.813, 0.714 
7.63 (a) \( C_{gs} = 2.3 \text{ pF, } C_{gd} = 2.87 \text{ pF} \) \hspace{1cm} (b) \( 9.6 \times 10^{10} \text{ rad s} \) 
7.65 (a) \( C_{sb} = 0.3 \text{ pF, } C_{gd} = 1.5 \text{ pF} \) \hspace{1cm} (b) \( 9.9 \times 10^{-9} \text{ rad s} \)

Chapter 8

8.12 (a) 150.5 \hspace{1cm} (b) 3.76 mA \hspace{1cm} (c) 3.787 mA 

Chapter 9

9.13 \( W = 39, V_i = 5.06 \text{ V} \)
9.19 (a) \( I_{D1,2} = 0.168 \text{ mA} \) \hspace{1cm} (b) \( A_d = -1.832, A_c = 9.945 \times 10^{-3}, \text{ CMRR} = 184.2, \) \( v_0 = -54.96 \text{ mV} \)
9.21 (a) 96.46 \mu A, 103.54 \mu A 
9.22 (a) 0.715 mA \hspace{1cm} (b) 139.8 k\Omega \hspace{1cm} (c) 100 V \hspace{1cm} (d) 1.2909 
9.24 (a) 1.36 mA \hspace{1cm} (b) 73.53 k\Omega \hspace{1cm} (c) 100 V \hspace{1cm} (d) 1.282 
9.26 (a) 4.152 mA \hspace{1cm} (b) 24 k\Omega \hspace{1cm} (c) 100 V \hspace{1cm} (d) 1.142 
9.28 \( R_2 = 11.88 \text{ k\Omega, } R_o = 55.69 \text{ M\Omega} \)
9.30 (a) \( R_1 = 2.86 \text{ M\Omega} \hspace{1cm} (b) \) \( R_o = 760 \text{ M\Omega} \)
9.31 1.231 mA, 9.26 M\Omega 
9.35 For \( V_A = 100 \) and \( \beta_F = 100, r_o = 100 \) k\Omega; for \( V_{CC} = 15 \text{ V, } R_1 = 13.6 \text{ k\Omega} \)
9.37 (a) 2.955 mA, 2.011 mA \hspace{1cm} (b) −961.5, −0.25, 38.44 \times 10^6, 20.8 \Omega, 6.04 \text{ M\Omega} 
9.41 769.2, 2.08 M\Omega, 8 M\Omega, 587.9 
9.45 −28.57, 0, 0
Chapter 10

10.1 (a) $-9.5 \times 10^{-3}$
(b) $-101.26$
10.4 (a) $22.173, 10.4\%$  (b) $9.99, 4\%$
10.5 9
10.6 (a) $2$  (b) $10^6$ Hz  (c) $2 \times 10^6$
10.9 (a) $3.17 \times 10^{11}$ Ω  (b) $0.475$ mΩ
(c) $1.25$
10.11 (a) $804$ kΩ  (b) $7.29$ Ω
(c) $1.31$
10.17 $5,355,006$ Ω, $0.3$ Ω, $7.43$
10.20 (a) $14.06$ kΩ  (b) $30.2$ Ω
(c) $1427.7$
10.24 (a) $28.5$ kΩ  (c) $3.96 \times 10^{-4}$ A
10.25 (a) $3.15$ MΩ  (b) $6.3$ MΩ
(c) $0.4 \times 10^{-3}$ A
10.28 $R_{if} = 1,321,326$ Ω, $A_f = 93.6$ mA
10.32 (a) $3.5$ kΩ  (b) $49$ Ω
(c) $-97.67$ Ω
10.35 $26.26$ Ω, $13.72$ Ω, $-3973$ V A
10.38 (a) $30.5$ Ω  (b) $342$ Ω
(c) $225,077$ V A
10.45 (a) $17.029$ Ω  (b) $37,072$ Ω
(c) $1.88$
10.47 (a) $12.86$ Ω  (b) $5$ kΩ
(c) $19.61$
10.51 $-\frac{1}{4} + \frac{3}{20}e^{-4t} + \frac{3}{30}e^{6t}$
10.52 $e^{0.5t}[2 \sin 10t]$
10.56 $50 (0.234 - j 0.643)$
10.58 44.44 Hz
10.60 (a) $401.9$ kHz  (b) $127$ pF
10.61 $10.47$ μF
10.64 (a) $3.32$ nF, $4.8$ kΩ
(b) $79.7$ pF, $199.8$ kΩ

Chapter 11

11.2 $R_L = 1.43$ kΩ, $29$ V
11.6 (a) $v_O = 14.5$ V, $R_1 = 493$ Ω
(b) $525.5$ mW  (c) $60.4\%$
11.8 (a) $I_C = 3.75$ A  (b) $28.125$ W
(c) $50\%$
11.11 $V_p = 14.8$ V, $P_L = 10.952$ W, $77.5\%$, $R_1 = 47$ Ω
11.14 (a) $7.5$ A, $V_{p(max)} = 9.55$ V
(b) $P_{L(max)} = 56.25$ W  (c) $78.55\%$
11.16 (a) $R_{ref} = 255$ Ω
11.32 $76.48$ W

Chapter 13

13.1 $14.29, 135^\circ$
13.5 $27$ Ω
13.7 $318$ Ω
13.10 For $C = 0.1$ μF, $318$ Ω, $20$ kΩ
13.14 $58.1$ kHz, $2.5$ Ω
13.16 For $C = 0.01$ μF, $25.3$ mH, $133$ Ω
13.19 $435.8$ kHz, $R_L = 2R_1$
13.24 $0.046$ pF

Chapter 14

14.3 $26$ mV
14.5 $1$ pV
14.9 (a) $\pm 2.7$ mV  (b) $-0.5 \pm 2.7$ mV
14.12 (a) $0.6$ mV  (b) $2.6$ mV
14.17 (a) $835.96$ kΩ  (b) $7.75$ mA  V
(c) $6480$  (d) $125$ kΩ
14.20 (a) $0.2$ μA  (b) $258$ kΩ
(c) $6.16$ MHz
14.21 (a) $0.1$ μA  (b) $516$ kΩ
14.23 $521,963$
14.24 347,104
14.28 (a) ±66.66 V μs  (b) 205.6 MHz

Chapter 15

15.1 (a) 2 kΩ, 3.28 kΩ  (b) 94
15.2 (a) 89.25 ps  (b) 4.635 mW
15.4 27.38 mA V²
15.7 (a) 3 V  (b) 0.4 V, 3.04 mA
15.9 (a) 3.5 V, 0.39 V  (b) 1.5 V, 3.37 mA, 8.438 mW, 0 W
15.11 4.2 V
15.16 (a) 2.5 V, 4 V, 1 V  (b) 0
15.17 (a) 4.914 V, 1.914 V
15.18 (a) 2.5 V, 3.5 V, 1.5 V
15.19 0.464
15.22 (a) 2.5 V, 3.5 V, 1.5 V  (b) 4.95 V, 0.25 mA, 0.05 V, 0.25 mA
15.23 (a) 1.15  (b) 2.1 V, 2.276 V
(c) 1.02 ns  (d) 0.51 pJ
15.28 (a) 17.55 kΩ  (b) 2
(c) 0.049 mW
15.30 (a) 1175 Ω, 3384 Ω  (b) 1.006, 1.63 V
(c) 42
15.32 (a) 4.8 kΩ, 14.4 kΩ  (b) N = 1
15.36 0.2 V, I_B = 2.102 mA, I_3 = 1.92 mA, N = 58
15.37 I_1 = 0.58 mA, I_2 = 1.64 mA, I_B3 = 2.102 mA, N = 64
15.38 R_1 = 4 kΩ, I_B1 = 725 μA, I_LD = 410 μA
15.41 21.15 mA, 1.15 mA, 0.846 mA
15.43 (a) v_O1 = −0.8 V, R_C1 = 400 Ω, R_4 = 4.5 kΩ
15.44 (a) v_A = 2.1 V, R_E = 2.3 kΩ, v_O1 = 2.2 V, R_4 = 3.7 kΩ, V_B6 = 0.2 V
15.45 R_3 = 1.75 kΩ, R_E = 1.75 kΩ, R_4 = 1250 Ω
15.47 (a) 0.106 ns

Chapter 16

16.4 (a) 3.32 V, 0.57 V  (b) −2.66 V, 0.456 V
(c) −3.1 V
16.10 For R_1 = 10 kΩ, R_P = 14 kΩ, R_x = 5833 Ω
16.14 For R_1 = 10 kΩ, R_P = 50 kΩ, V_ref = 4.8 V
16.18 2127.6 Hz
16.21 (a) 16 kHz  (b) 12.8 kHz
16.26 (c) 196 Hz
16.31 (a) 2.5 kHz  (b) ±200 3 kHz
(c) 543 Hz
16.37 0.747 V
16.38 8
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Absolute noise margin</td>
<td>988</td>
</tr>
<tr>
<td>Absolute position encoders</td>
<td>5</td>
</tr>
<tr>
<td>Absolute stability/instability</td>
<td>699</td>
</tr>
<tr>
<td>AC-coupled amplifiers</td>
<td>62–63, 72–73</td>
</tr>
<tr>
<td>AC-coupled bootstrapped voltage followers</td>
<td>162–163</td>
</tr>
<tr>
<td>Accreditation Board for Engineering and Technology (ABET)</td>
<td>20</td>
</tr>
<tr>
<td>Accuracy</td>
<td>1169</td>
</tr>
<tr>
<td>AC-DC converter</td>
<td>238</td>
</tr>
<tr>
<td>AC linear model</td>
<td>112–113</td>
</tr>
<tr>
<td>AC output power</td>
<td>240–241, 250</td>
</tr>
<tr>
<td>AC resistance</td>
<td>200</td>
</tr>
<tr>
<td>AC sweep analysis</td>
<td>1191</td>
</tr>
<tr>
<td>Acquisition time</td>
<td>1156</td>
</tr>
<tr>
<td>Active-biased common-emitter amplifier</td>
<td>468–471</td>
</tr>
<tr>
<td>Active-biased emitter follower</td>
<td>476–479</td>
</tr>
<tr>
<td>Active-biased source follower</td>
<td>376–378</td>
</tr>
<tr>
<td>Active current-source biasing</td>
<td>458–459</td>
</tr>
<tr>
<td>Active filters</td>
<td>804–805, 805–807</td>
</tr>
<tr>
<td>Active-filter tuned oscillators</td>
<td>899–901</td>
</tr>
<tr>
<td>Active load</td>
<td>613</td>
</tr>
<tr>
<td>Actuators</td>
<td>5–6</td>
</tr>
<tr>
<td>A/D converters</td>
<td>87–88</td>
</tr>
<tr>
<td>Addition-subtraction amplifier</td>
<td>149–151</td>
</tr>
<tr>
<td>Adjustable negative voltage limiter</td>
<td>1087, 1089</td>
</tr>
<tr>
<td>Adjustable-voltage limiters</td>
<td>1086–1092</td>
</tr>
<tr>
<td>AF amplifier</td>
<td>16</td>
</tr>
<tr>
<td>AF oscillators</td>
<td>15–16, 15–16</td>
</tr>
<tr>
<td>All-pass filters</td>
<td>848–949</td>
</tr>
<tr>
<td>Amplifier(s)</td>
<td>15–16, 15–16</td>
</tr>
<tr>
<td>Amplifier(s) specific type of amplifier</td>
<td>149–151</td>
</tr>
<tr>
<td>Amplifiers</td>
<td>15–16, 15–16</td>
</tr>
<tr>
<td>Amplification</td>
<td>16</td>
</tr>
<tr>
<td>Band-pass characteristic</td>
<td>68–69</td>
</tr>
<tr>
<td>BiCMOS differential</td>
<td>620–626</td>
</tr>
<tr>
<td>BJT, 497–532, 508–528, 620–621, 656, 921</td>
<td></td>
</tr>
<tr>
<td>BJT differential, 602–619</td>
<td></td>
</tr>
<tr>
<td>bootstrapped, circuit, 16</td>
<td></td>
</tr>
<tr>
<td>bridge, 97–92</td>
<td></td>
</tr>
<tr>
<td>capacitive-coupled, 62–63</td>
<td></td>
</tr>
<tr>
<td>capacitor-coupled casceded</td>
<td>383–384, 488–489</td>
</tr>
<tr>
<td>cascaded current, 60–62</td>
<td></td>
</tr>
<tr>
<td>cascaded voltage, 59–60</td>
<td></td>
</tr>
<tr>
<td>cascode BiCMOS, 622–625</td>
<td></td>
</tr>
<tr>
<td>cascoded, 384–386, 489–491</td>
<td></td>
</tr>
<tr>
<td>cascode differential, 617–619</td>
<td></td>
</tr>
<tr>
<td>class A, 745–755</td>
<td></td>
</tr>
<tr>
<td>class AB, 766–777</td>
<td></td>
</tr>
<tr>
<td>class B, 756–766</td>
<td></td>
</tr>
<tr>
<td>class C, 777–780</td>
<td></td>
</tr>
<tr>
<td>class D, 781–783</td>
<td></td>
</tr>
<tr>
<td>class E, 784–786</td>
<td></td>
</tr>
<tr>
<td>CMOS, 620–621, 921–922</td>
<td></td>
</tr>
<tr>
<td>common-base, 483–487, 518–522</td>
<td></td>
</tr>
<tr>
<td>common-collector BJT, 514–518</td>
<td></td>
</tr>
<tr>
<td>common-drain, 375–380, 401–403</td>
<td></td>
</tr>
<tr>
<td>common-emitter, 749–755</td>
<td></td>
</tr>
<tr>
<td>common-emitter BJT, 467–476, 508–514</td>
<td></td>
</tr>
<tr>
<td>common-gate, 380–383, 403–408</td>
<td></td>
</tr>
<tr>
<td>common-source, 364–375, 396–401</td>
<td></td>
</tr>
<tr>
<td>complementary push-pull, 756–761</td>
<td></td>
</tr>
<tr>
<td>current, 16, 53–54, 87</td>
<td></td>
</tr>
<tr>
<td>DC, 16</td>
<td></td>
</tr>
<tr>
<td>DC-coupled, 62–63</td>
<td></td>
</tr>
<tr>
<td>depletion MOS differential, 580–585</td>
<td></td>
</tr>
<tr>
<td>design, 88–91</td>
<td></td>
</tr>
<tr>
<td>design of, 88–91</td>
<td></td>
</tr>
<tr>
<td>direct-coupled, 16, 62–63</td>
<td>384, 489</td>
</tr>
<tr>
<td>double-cascode BiCMOS, 623–625</td>
<td></td>
</tr>
<tr>
<td>feedback, 642, 656–657, 698–710</td>
<td></td>
</tr>
<tr>
<td>frequency types of, 16</td>
<td></td>
</tr>
<tr>
<td>functional types of, 16</td>
<td></td>
</tr>
<tr>
<td>high-pass characteristic</td>
<td>66–68</td>
</tr>
<tr>
<td>IC power, 789–792</td>
<td></td>
</tr>
<tr>
<td>impedance, 16</td>
<td></td>
</tr>
<tr>
<td>instrumentation, 145–147</td>
<td></td>
</tr>
<tr>
<td>interstage coupling types</td>
<td>16</td>
</tr>
<tr>
<td>inverting, 121–128</td>
<td></td>
</tr>
<tr>
<td>inverting summing, 148–149</td>
<td></td>
</tr>
<tr>
<td>LC-coupled, 16</td>
<td></td>
</tr>
<tr>
<td>level-shifted DC, 497–501</td>
<td></td>
</tr>
<tr>
<td>level-shifted MOS, 388–392</td>
<td></td>
</tr>
<tr>
<td>linear, 16, 40</td>
<td></td>
</tr>
<tr>
<td>load types, 16</td>
<td></td>
</tr>
<tr>
<td>low-pass characteristic</td>
<td>63–66</td>
</tr>
<tr>
<td>MOS differential, 566–580</td>
<td></td>
</tr>
<tr>
<td>MOSFET, 353–356, 393–413, 656</td>
<td></td>
</tr>
<tr>
<td>multistage, 86, 383–386, 488–491, 522–528</td>
<td></td>
</tr>
<tr>
<td>noninverting, 114–121</td>
<td></td>
</tr>
<tr>
<td>noninverting summing, 147–148</td>
<td></td>
</tr>
<tr>
<td>nonlinearity, 46–47</td>
<td></td>
</tr>
<tr>
<td>operational, 642, 643, 656</td>
<td>788–791</td>
</tr>
<tr>
<td>power, 16, 53, 740–743, 789–796</td>
<td></td>
</tr>
<tr>
<td>PSpice/SPICE, 87–88</td>
<td></td>
</tr>
<tr>
<td>quasi-complementary class AB, 775–776</td>
<td></td>
</tr>
<tr>
<td>RC-coupled, 16</td>
<td></td>
</tr>
<tr>
<td>RF, 16</td>
<td></td>
</tr>
<tr>
<td>saturation, 43–44</td>
<td></td>
</tr>
<tr>
<td>scaling, 148</td>
<td></td>
</tr>
<tr>
<td>servo, 16</td>
<td></td>
</tr>
<tr>
<td>signal types, 15–16</td>
<td></td>
</tr>
<tr>
<td>summing, 148</td>
<td></td>
</tr>
<tr>
<td>transformer-coupled, 16</td>
<td></td>
</tr>
<tr>
<td>transformer-coupled class AB, 776–777</td>
<td></td>
</tr>
<tr>
<td>transformer-coupled load</td>
<td></td>
</tr>
<tr>
<td>common-emitter, 753–755</td>
<td></td>
</tr>
<tr>
<td>transformer-coupled load push-pull, 761–765</td>
<td></td>
</tr>
</tbody>
</table>
Index

Amplifier(s) (continued)
transimpedance, 57–58, 88
tuned, 16, 68
types of, 50–58
UHF, 16
video, 16
voltage, 15, 51–52, 87
wideband, 16
Amplifier characteristics, 40–50
current gain, 42
input and output resistances, 43
logarithmic gain, 42–43
nonlinearity, 46–47
power gain, 42
rise time, 46–47
saturation, 43–44
SR, 48–49
voltage gain, 41–42
Amplifier frequency response,
62–71
band-pass characteristic, 68–69
gain and bandwidth relation, 69–70
high-pass characteristic, 66–67
low-pass characteristic, 63–66
Amplitude modulated waveform, 273
Amplitude stability, 866
Analog electronics, 10
Analog filters, 804
Analog-to-digital (A/D) converters,
7–8
IC, 1167–1169
successive-approximation, 1165–1167
Analog integrated circuits, 1169
Analysis, v. design, 20
AND, 983, 984
Aperture time, 1156
Approximate method, of analysis, 193
Astable multivibrator, 1110, 1113,
1131–1139
applications of, 1134–1138
in 555 timer, 1131–1134
Audio amplifier, 16
Audio-frequency (AF) oscillators,
867–881
phase-shift, 866–870
quadrature, 871–872
ring, 878–880
three-phase, 873–874
Wien-bridge, 874–877
Avalanche breakdown, 316
Average static power dissipation, 993
Averaging amplifier, 148
Background music decoder,
1145–1146
Band-elimination filters, 843
Band-pass characteristic amplifier,
68–69
Band-pass (BP) filters, 807,
837–843
narrow-band-pass, 840–843
wide-band-pass, 837–840
Band-reject filters, 843–848
narrow-band-reject, 846–848
wide-band-reject, 844–846
Band-stop filters, 807, 843
Bandwidth (BW), 13–14, 66, 804,
1252.
unity-gain, 68
Base-charging capacitance, 501
Base narrowing, 441–443
Base recovery circuit, 1043
Base terminal, 28
Base-to-emitter junction (BEJ), 1028
Base transit time, 501
Base-width modulation, 441
Basic circuits
first-order high-pass CR
circuits, 1246–1248
frequency response of first- and second-order
circuits, 1243–1244
KCL, 1214–1216
KVL, 1216–1219
maximum power transfer
theorem, 1227–1228
Norton’s theorem, 1226
resonant circuits, 1238–1242
second-order parallel RLC
circuits, 1254–1258
second-order series RLC
circuits, 1249–1254
superposition theorem,
1219–1221
Thevenin’s theorem, 1221–1226
time constants of first-order
circuits, 1258–1259
transient response of first-order
circuits, 1228–1238
Basic CMOS op-amp, 933
Basic common-emitter amplifier,
749–752
Basic MOSFET current source,
559–561
B-E depletion capacitance, 502
B-E input capacitance, 503
BEJ. See Base-to-emitter junction
Bell Telephone Laboratories, 2
Biasing
active current-source, 458–459
with \(V_{BE}\) multiplier circuit, 772–774
of BJTs, 457–467
of cascoded amplifier, 384–386, 489
circuit design for, 463–467
class AB amplifier, 767–774
class B amplifier, 764
with diodes, 767–771
emitter resistance-feedback, 460–461
of MOSFET, 357–364
single-base resistor, 459–460
substrate, 345–346
thermal runaway and, 768
two-base resistor, 461–462
voltages, 436
Biasing circuit
of BiCMOS op-amp LH0022, 967
of LM741 op-amp, 945–946
BiCMOS amplifiers, 621–622
analyzing, 624–625
cascade, 622–623
differential, 620–626
double-cascade, 623–624
BiCMOS inverters, 1057
propagation delay, 1058–1059
BiCMOS op-amps, 962–974
CA3130, 962–963
CA3140, 964–965
LF411, 969–971
LH0022, 965–968
LH0032, 973–974
LH0062, 971–973
Bioelectronics, 34–35
Biofuel cells, 35
Biomaterial, 34
Bioreporters, 35
Biosensors, 35
Biotechnology, 34
Bipolar junction transistor (BJT),
28, 106, 162, 434–436, 533, 984
base narrowing in, 441–443
biasing of, 457–467
circuit models for, 449–455
cutoff/saturation/inverse-active
modes of, 440–441
forward-active mode of, 436–440
frequency response of, 501–508
high-frequency model of, 501–503
input/output characteristics of, 447–449
model parameters for, 1205
MOSFETs v., 528
operation of, 436–447
PSpice model for, 503–504
saturation current in, 444–446
Bipolar transistors, 1203–1204
Biquadratic equation, 810–813
Bistable multivibrator, 1101, 1109
BJT. See Bipolar junction transistor
BJT amplifiers, 921
CMOS amplifier v., 620–621
design of, 528–532
feedback in, 656
frequency response of, 508–528
with level shifting, 497–501
thermal voltage drift in, 921
BJT current sources, 558–559,
586–602
basic, 586–588
cascade, 595–596
modified basic, 589–591
Widlar current source, 591–595
Wilson current source, 596–601
BJT differential amplifiers,
602–619
with basic current mirror active load, 613–615
cascade, 617–619
with modified current mirror, 615–617
pair with resistive load, 602–612
BJT inverters, 1026–1033
designing, 1030–1033
switching characteristics of, 1028–1029
voltage transfer characteristics of, 1027–1028
BJT op-amps, 940–944
LM124, 940–942
LM741, 942–944
BJT switch, 455–457
Bode plots, 707–710, 712, 807
Body effect
of NMOS inverter with depletion load, 1008
of NMOS inverter with enhancement load, 1001
Bootstrapped amplifier circuit, 163
Bootstrapped voltage followers, AC-coupled, 162–163
BP filters. See Band-pass filters
Breakdown condition, 315–316
Breakdown region, of practical diodes, 187
Breakdown voltage, 208
Break frequency, 65, 67, 1245
Bridge amplifier, 791–792
Buffer stage, 116
Built-in junction potential, 308–310
Built-in potential, 307, 327, 503
Bulk resistance, 205
Butterworth filters, 814–818
Butterworth functions
for high-order filters, 816–817
for n = 1, 815
for n = 2, 815–816
for n = 3, 816
Butterworth high-pass filters, 834–836
Butterworth low-pass filters, 826–829
Butterworth response, 814
BW. See Bandwidth
CA3130 BiCMOS op-amp, 962
differential stage of, 963–964
gain stage of, 964
output stage of, 964
parameters of, 963
schematic for, 963
CA3140 BiCMOS op-amp
output stage of, 965
parameters of, 965
schematic for, 964
Capacitance
depletion, 326–327
diffusion, 327–328
junction, 317–318
Miller, 78
parasitic, 393
transition, 327
Capacitive-coupled amplifiers, 62–63
Capacitor-coupled casceded amplifiers, 383–384,
488–489
Capacitors, 1202, 1279
Capture mode, 1139
Capture program, 1180
Capture range, 1140
Carrier concentrations, 305–307
Carriers, 301
Casceded amplifiers, 59–62
Casceded current amplifiers, 60–62
Casceded voltage amplifiers, 59–60
Cascode BiCMOS amplifiers, 622–625
Cascode current source, 558–559
Cascode BiCMOS, 623–625
Cascode differential amplifier, 617–619
Cascoded MOS differential amplifier, 578–580
CB amplifier. See Common-base amplifiers
C-B junction capacitance, 503
C-B resistance, 451
CCCS. See Current-controlled current source
CCVS. See Current-controlled voltage source
Center frequency, 68, 1139
C filters, 264–269
Channel length modulation, 343–345
Channel modulation length, 352
Channel modulation voltage, 352
Characteristic equation, 699
Charging mode, of VCO, 1120
Circuit design, using analog integrated circuits, 1169
Circuit implementation, of VCO, 1121–123
Circuit-level design process, 21–24
Circuits
historical levels of integration, 3
with op-amps and diodes, 1081–1097
pulse response of, 12
Clamp diodes, 1101
Clamping circuits, 279–284
fixed-shift clampers, 280–281
precision, 1085–1086
variable-shift clampers, 281–282
Class A amplifiers, 745–755
basic common-emitter, 749–752
common-emitter amplifiers with active load, 752–753
emitter followers, 745–749
transformer-coupled load common-emitter, 753–755
Class AB push-pull amplifiers, 766–777
biasing with diodes, 767–768
biasing with diodes and active current source, 769–770
biasing with VBE multiplier circuit, 772–774
output power/efficiency of, 767
quasi-complementary, 775–776
transfer characteristic of, 767, 770
transformer-coupled, 776–777
Class B push-pull amplifiers, 756–766
complementary, 756–761
DC biasing of, 764
transformer-coupled load, 761–765
Class C amplifiers, 777–780
Class D amplifiers, 781–783
Class E amplifiers, 784–786
Clippers, 276–279
Clock-tunable resistor, 850
Closed-loop gain, 644–651
frequency of, 698–699
low-frequency, 648
poles of, 699, 700–702
sensitivity of, 646–647
stability of, 698–702
CMOS. See Complementary metal oxide semiconductor
CMOS amplifiers, 921–922
BJT amplifier v., 620–621
designing, 576–578
differential, 575–578
thermal voltage drift in, 921–922
CMOS driving TTL, 1062–1063
CMOS families, 1025
CMOS inverters, 1016–1022
CMOS logic circuits, 1022–1026
NMOS gates v., 1026
NOR/NAND gates, 1024–1025
transmission gates, 1022–1024
TTL driven by, 1062–1063
TTL driving, 1060–1062
CMOS NAND gates, 1024–1025
CMOS NOR gates, 1024–1025
CMOS op-amps, 933–940
basic, 933
MC14573, 937
TLC1078, 937–940
CMOS technology, 576
CMOS transmission gates, 1022–1023, 1026
CMRR. See Common-mode rejection ratio
Collector saturation current, 445
Collector terminal, 28
Color television, 2
Colpitts oscillators, 881–888
Common-base amplifiers (CB amplifier), 483–487
frequency response of, 518–522
Common-collector BJT amplifiers, 514–518
Common-drain amplifiers, 375–380
active-biased source follower, 376–378
frequency response of, 401–403
resistive-biased source follower, 378–380
Common-emitter amplifiers with active load, 752–753
basic, 749–752
transformer-coupled load, 753–755
Common-emitter BJT amplifiers, 467–476
frequency response of, 508–514
Common-gate amplifiers, 380–383
frequency response of, 403–408
Common ground, 40
Common-mode rejection ratio (CMRR), 108–111, 556, 606, 609–610, 925
of differential amplifier, 144–145
of instrumentation amplifier, 146–147
of inverting amplifiers, 122–123
differential, 575–578
of noninverting amplifiers, 116–117
in op-amps, 925
Common-source amplifiers (CS amplifier), 364–375
with current source load, 364–367
with depletion MOSFET load, 369
with enhancement load, 368–369
frequency response of, 396–401
with resistive load, 370–374
Comparators, 1097–1100
op-amps v., 1098
output-side connection, 1098
threshold, 1098–1100
transfer characteristics of, 1097
Compensation techniques, 711–720
adding dominant pole, 711–712
changing dominant pole, 713–714
Miller compensation/pole splitting, 715–716
modifying feedback path, 717–720
Compensating techniques, 711–720
Complementary metal oxide semiconductor (CMOS), 346
Compensate-pull-pull amplifiers, 756–761
dead-zone minimization for, 759
output power/efficiency of, 757–758
Constant current sources, 159
Constant-current DC model, 196–197
Conversion speed, 1169
Converters
AC-DC, 238
analog-to-digital, 7–8, 1165–1169
digital-to-analog, 8, 1158–1164
frequency-to-voltage, 1151–1155
IC A/D, 1167–1169
IC D/A, 1162
MC1408 D/A, 1163
negative impedance, 158
NE/SE-5018 D/A, 1163–1164
R-2R ladder network D/A, 1160–1162
successive-approximation A/D, 1165–1167
TelCom 9400 series, 1147, 1150, 1154, 1155
voltage-current, 155–156
voltage-to-frequency, 1120, 1147–1151, 1155
weighted-resistor D/A, 1159–1160
Corner frequency, 65, 67, 1245
Coupling capacitors, 62, 467, 479
Crossover distortion, 756–757, 762, 766
Crystal oscillators, 895–899
CS amplifier. See Common-source amplifiers
Current amplifiers, 16, 53–54, 87
Current boosters, 788
Current-controlled current source (CCCS), 53–54, 87
Current-controlled voltage source (CCVS), 88
Current gain, 42, 53
Current mirror source, 388–390, 497–498
Current-sensing/current-comparing feedback, 653–655. See also Shunt-series feedback
Current-sensing/voltage comparing feedback, 653. See also Series-series feedback
Current sink, 559
Current sinking source, 458
Current source biasing with active, 458–459
BJT, 586–602
level shifting with, 387–388, 497
Current switch, 1122
Cut-in voltage, 186
Cutoff frequency, 65, 67, 806, 1250
Cutoff region
  of depletion MOSFETs, 349
  of enhancement MOSFETs, 338

D/A converters. See Digital-to-analog converters

Damping factor, 1255
Damping ratio, 1255
Darlington emitter follower, 965
Darlington pair transistor, 491–495, 615, 925–926
DC amplifier, 16
DC analysis
  of LM741 BJT op-amp, 944–951
  of LM741 op-amp, 944–945
DC and small-signal specifications, 14–15
DC biasing
  of BiCMOS op-amp LF411, 970
  of BJTs, 457–467
  of cascoded amplifier, 384–386, 489
  of class B amplifier, 764
  of CMOS op-amp MC14573, 935
  of CMOS op-amp TLC1078, 939
  of MOSFET, 357–364
DC-coupled amplifiers, 62–63
DC filters, 260
DC level shifting, 386–388, 495–501
DC linear model, 111–112
DC millivoltmeters, 157
DC sweep analysis, 1191
DC transfer characteristics, of BJT differential amplifier, 603–605
DC voltmeters, 156
Deadband, 1103, 1107, 1109
Dead zones, 756, 759, 766
Decade, 1245
Decade increase, 65
Degenerative feedback, 642. See also Negative feedback
Delay equalizer, 848
Delay-power product (DP), 995
Delay time, 12, 1028
Demodulators, 272–275
Depletion capacitance, 326–327
Depletion load, of NMOS inverter, 1005
Depletion MOS differential amplifiers, 580–585
Depletion MOS differential pair with active load, 585
Depletion MOS differential pair with resistive load, 581–585
Depletion MOSFETs, 30–31, 352, 369
  operation, 346–348
  output/transfer characteristics of, 348–349
Depletion ratio, 315
Depletion region, 307, 315
Depletion region width, 316–317, 320
Design constraints, 22
Design criteria, 22
Design perspective, 25
Design problems
  large design projects, 1282–1283
  medium design projects, 1282
  mini-design projects, 1282
Design projects, 25–26
Design report, 26–27
Design statement, 22
Detectors
  diode peak, 272–275
  edge-triggered phase, 1140–1141
  missing-pulse, 1129–1130
  most positive signal, 1081
  phase, 1140–1141
  photo, 154–155
  precision peak voltage, 1081–1082
  zero-crossing, 1100–1101
Differential amplifiers, 143–145
  cascaded MOS, 578–580
  characteristics of, 554–556
  CMOS, 575–578
  depletion MOS, 580–585
  design of, 628
  frequency response of, 626–628
  internal structure of, 554–558
  MOS, 566–580
  Differential-mode half circuit, 606
Differential stage
  of BiCMOS op-amp CA3130, 963–964
  of BiCMOS op-amp LH0022, 965–967
  of BiCMOS op-amp LH0062, 971
  of BJT op-amps LM124, 940
  of CMOS op-amp MC14573, 935
  of CMOS op-amp TLC1078, 938
  Differentiating, 201
  Differentiators, 137–143
  Differentiator time constant, 138
  Diffusion capacitance, 327–328
Digital electronics, 10–11, 982
Digital filters, 804
Digital integrated circuits, 984–985
Digital logic circuits, 982
Digital-to-analog (D/A) converters, 8, 1158–1164
IC, 1162–1164
R-2R ladder network, 1158–1166
weighted-resistor, 1159–1160
Diode(s), 1203. See also Light-emitting diodes; Organic light-emitting diodes; Practical diodes
Superdiode biasing with, 767–771
breakdown region of practical, 187
circuits with op-amps and, 1081–1097
clippers, 276–279
determination of constants, 187–189
forward biased, 180
forward-biased region of practical, 186
ideal, 180–183
model parameters for, 1204
op-amp circuits with, 1081–1097
operation point of, 193
power rating, 220–222
PSpice/SPICE model, 205–207
as rectifiers, 238–260
reverse biased, 180
reverse-biased region of practical, 186–187
semiconductor, 27, 180
Shockley, equation, 185
simple circuit, 192
small-signal, transconductance, 200
solid-state point-contact, 2
zener, 208–219, 224–225
Diode applications
clamping circuits, 279–284
clippers, 276–279
diode rectifier, 238–260
function generators, 287–290
output filters for rectifiers, 260–272
peak detectors and demodulators, 272–275
voltage multipliers, 284–287
Diode circuits
  simple, 192
  transfer characteristics of, 183–184
Diode clamping circuits, 279–284
Diode clippers, 276–279
  parallel clippers, 276–277
  series clippers, 277–278
Diode data sheets, 222–225
Diode function generators, 287–290
Diode peak detectors and demodulators, 272–275
Diode rectifier, 238–260
  single-phase full-wave bridge rectifiers, 254–260
  single-phase full-wave center-tapped rectifiers, 247–254
  single-phase half-wave rectifiers, 238–247
Diode-transistor logic (DTL), 1026
Diode voltage multipliers, 284–287
Direct-coupled amplifiers, 16, 62–63, 384, 489
Discharging mode, of VCO, 1121
Distortion, 13, 40, 649–651, 804
  crossover, 756–757
  measurement of, 740
  total harmonic, 740
Dominant pole
  adding, 711–712
  changing, 713–714
Donor impurity, 302
Doping, 301
Double-cascode BiCMOS amplifiers, 623–625
DP. See Delay-power product
Drain terminal, 29
Drop decay rate, 1156
DTL. See Diode-transistor logic
Dynamic power, 993–994
Dynamic resistance, 200
Early effect, 441
Early voltage, 442, 451
Ebers-Moll model
  of bipolar junction transistors, 1267–1274
  for npn transistor, 1268
  transport version model, 1270
ECL. See Emitter-coupled logic
Edge-triggered phase detector, 1140–1141
Effective density states function in the conduction band, 306
  in the valence band, 306
Electrical Engineering Design Compendium, 25
Electric field distribution, 310–311
Electron-hole generation and recombination, 303–304
Electronic circuits design, 20–27
  analysis v. design, 20
  circuit-level design process, 21–22
  design report, 26–27
  engineering design defined, 20
  large design projects, 26
  medium design projects, 26
  mini design projects, 26
  perspective for study benefits, 25
  project types, 25
  short design projects, 25
Electronic devices, 27–31
  bipolar junction transistors, 28
  field-effect transistors, 29
  semiconductor diodes, 27
Electronic signals and notation, 6–7
  analog-to-digital converters, 7–8
  digital-to-analog converters, 8–10
  notation, 8–10
Electromagnetic systems
  distortion, 13
  frequency specifications, 13–14
Emerging electronics, 32–33
  bioelectronics, 34–35
  memristors, 32–33
  organic electronics, 33–34
Emission coefficient, 185
Emitter-coupled logic (ECL), 1027, 1162
  OR/NOR gates, 1049–1057
Emitter-coupled pair, 1049
Emitter-follower biasing, 461
Emitter followers, 476–483, 745–749
  active-biased, 476–479
  biasing with, 461
  class A amplifiers and, 745–749
  Darlington, 965
  level shifting with, 496
  resistive-biased, 479–483
Emitter resistance-feedback biasing, 460–461
Emitter scaling, 445
Emitter terminal, 28
Encoders, absolute position, 5
Engineering design, 20–21
Enhancement load, 996–997
Enhancement MOSFETs, 29–30, 337–338, 352, 368–369
  channel length modulation, 343–344
  operation of, 338–341
  output/transfer characteristics of, 342
  substrate biasing effects, 345–346
Equilibrium electrostatic potential, 1001
Error
  gain, 1163
  linearity, 1163
  nonlinearity, 1163
  offset, 1163
  quantizing, 1169
  signal, 642
  voltage, 1139
Exposed region, 982
Extrinsic, 301
Fabrication process constant, 1001
Fairchild Semiconductor, 2
Falloff rate, 807
Fall time, 12, 991, 1029
Fan-in, 988–989
Fan-out, 988–989
Feedback
  in BJTs, 656
  characteristics of, 644–651
  compensation techniques for, 711–720
  configurations of, 652–655
  frequency response of, 648–649
  modifying path of, 717–720
  in MOSFETs, 656
  negative, 642, 648, 649–650
  in op-amps, 656
  oscillators and, 699, 863
  positive, 642, 645, 768
  series-series, 652, 653,
  656–657, 667–676
  series-shunt, 643, 652, 653,
  656–667, 692–693
  shunt-series, 652, 653–655,
  656–657, 686–691, 694–697
  shunt-shunt, 644, 652, 653, 654,
  656–657, 677–686
  thermal runaway and, 768
  topologies of, 652–655, 721
  types of, 642, 721
Feedback amplifier
analysis of, 656–657
stability analysis of, 698–710
Feedback circuit design, 692–698
Feedback current signal, 643
Feedback factor, 644, 646–647, 698
Feedback impedance, 71
Feedback ratio, 644
voltage-, 1262
Feedback relationships, 655
Feedback signal, 642, 643
Fermi function, 304–305
FETs. See Field-effect transistors
Field-effect device, 2
Field-effect transistors (FETs), 29, 106, 336. See also Metal oxide semiconductor field-effect transistors
Figure of merit, 866
Filtering circuit, 805
Filters. See also Active filters.
High-pass filters; Low-pass filters
all-pass, 848–849
analog, 804
band-elimination, 843
band-stop, 807, 843
BP, 807, 837–843
Butterworth, 814–818
Butterworth functions for high-order, 816–817
Butterworth low-pass, 834–836
Butterworth low-pass, 826–829
C, 264–269
DC, 260
design guidelines, 854
digital, 804
first-order, 808–810
first-order high-pass, 829–831
first-order low-pass, 819–822
ideal characteristics of, 806
L, 261–263
LC, 269–272, 781
low-pass, 819–829
MF5 universal monolithic switched-capacitor, 852
multiple feedback, 840
narrow-band-pass, 840–843
narrow-band-reject, 846–848
notch, 844, 846
output, 260–272
passive, 804–805
realistic characteristics of, 807
second-order, 813
second-order Butterworth, 852–853
second-order high-pass, 831–833
second-order low-pass, 822–826
switched-capacitor, 849–854
universal, 851
universal switched-capacitor, 851–852
wide-band-pass, 837–840
wide-band-reject, 844–846
First-order filters, 808–810
First-order high-pass CR circuits, 1246–1249
First-order high-pass filters, 829–831
First-order low-pass filters, 819–822
First-order low-pass RC circuits, 1244–1246
555 timer, 1126
astable multivibrators in, 1131–1134
functional block diagram of, 1126–1127
monostable multivibrators, 1127–1128
Fixed-shift clamps, 280–281
Fixed-voltage limiters, 1086
FM. See Frequency modulation
Forward base transit time, 502
Forward biased diode, 180
Forward-biased model, 328
Forward-biased pn junction, 319–323
depletion region width, 320–321
minority carrier charge distribution, 321–323
Forward-biased region, of practical diodes, 186
Forward-current amplification factor, 449
Forward current gain, of transistor, 28
Forward diffusion current, 308
Free-running frequency, 1139
Free-running mode, 1139
Free-running multivibrator, 1110, 1113, 1131
Frequency amplifiers, 16
Frequency divider, 1129
Frequency modulation (FM), 2, 1120
Frequency multiplier, 1143–1144
Frequency ratio, 1249
Frequency response, 927–930
with active load, 627–628
of amplifiers, 62–71
ofBJT amplifiers, 508–528
of BJTs, 501–508
of differential amplifiers, 626–628
of feedback, 648–649
of first- and second-order circuits, 1243–1244
of LM741 op-amp, 958–961
of MOSFET amplifiers, 393–408
of op-amp differentiators, 141–143
of op-amp integrators, 134–137
of practical op-amps, 927–930
with resistive load, 626–627
Frequency-response analysis, 961, 1198–1199
Frequency response methods, 62–71
high-frequency transfer function method, 75–77
high-frequency zero-value method, 82–84
low-frequency short-circuit method, 80–82
low-frequency transfer function method, 73–75
midband voltage gain, 84
Miller’s capacitor method, 77–78
multistage amplifiers, 86
Frequency response sweep analysis, 1191
Frequency scaling, 854
Frequency-shift keying (FSK) modulator, 1137–1139, 1145
Frequency specifications, 13–14
Frequency stability, 866
Frequency-to-voltage (F/V) converters, 1151–1155
FSK demodulator, 1145
FSK modulator. See Frequency-shift keying modulator
Full-wave voltage doubler circuit, 286
Functional amplifiers, 16
Functional block diagram, 1126–1127
Function generators, 287–290
F/V converters. See Frequency-to-voltage converters
Gain
active filters and, 804
bandwidth relation and, 69–71
Gain-bandwidth product (GBW), 648–649, 692, 712, 721
Gain crossover frequency, 705
Gain margin, 704–705
Gain parameter, 40
Gain sensitivity, 646
loop, 645
Gain stage
of BiCMOS op-amp CA3130, 964
of BiCMOS op-amp LH0022, 967
of BiCMOS op-amp LH0062, 972
of BJT op-amps LM124, 941
of CMOS op-amp MC14573, 935
of CMOS op-amp TLC1078, 938
of LM741 op-amp, 948–949
of small-signal AC analysis, 957–958
Gate terminal, 29
GBW. See Gain-bandwidth product
General Electric Company, 2
Graphical method, of analysis, 192–193
Half-power frequency, 65, 67
Half-wave rectifier, single phase, 238–247
Half-wave voltage doubler
circuit, 285
Hard limiters, 1094–1097
Harmonics, 260
Hartley oscillators, 888–891
Heat flow, 793–794
Heat sinks, 793–794
High cutoff frequencies
of common-base BJT
amplifiers, 520–521
of common-collector BJT
amplifiers, 516–517
of common-drain amplifiers, 402
of common-emitter BJT
amplifiers, 510–511
of common-gate amplifiers, 404–405
of common-source amplifiers, 398–399
High-frequency AC model, 326–329
depletion capacitance, 326–327
diffusion capacitance, 327–328
forward-biased model, 328
reverse-biased model, 329
High-frequency BJT model,
501–503
High-frequency MOSFET models,
393–395
High-frequency transfer function
method, 75–77
High-frequency zero-value method,
82–84
High noise margin, 987
High-pass characteristic amplifier,
66–68
High-pass circuit, 1248
High-pass (HP) filters, 806,
829–837
Butterworth, 834–836
first-order, 829–831
second-order, 831–833
High-pass gain, 1248
High-speed TTL NAND gates,
942–949
Hole, 301
HP filters. See High-pass filters
Hybrids, 33
Hysteresis, 1103, 1109
output voltage and, 1107–1108
IC. See Integrated circuits
IC operational amplifier, 2
IC power amplifiers, 789–792
Ideal diodes, 27–28, 180–183
Ideality factor, 185
Ideal series-series feedback
network, 669–670
Ideal series-shunt feedback
network, 658–660
Ideal shunt-series feedback
network, 688
Ideal shunt-shunt feedback
network, 678–680
Impedance amplifier, 16
Impurities, 301
Inductance simulators, 161–162
Inductors, 1202–1203
Injection version, of Ebers-Moll
model, 1268–1269
Input biasing current, 912–914
Input offset current, 914–917
Input offset voltage, 919, 920
Input resistance, 43, 925–926,
1262
Input signal, 1168
Input stage
of LM741 op-amp, 946–948
of small-signal AC analysis,
952–957
Instrumentation amplifiers,
145–147
Integrated circuits (IC), 2, 336
A/D converters, 1167–1169
D/A converters, 1162
digital, 984–985
PLL, 1141–1146
SAH, 1157–1158
Integration time constant, 129
Integrators, 128–137
practical inverting, 130
switched-capacitor, 851
Intel, 2
Interstage coupling amplifiers, 16
Intrinsic Fermi energy, 306
Intrinsic material, 300
Inverse-active mode, 1036
Inverter, 983
Inverting amplifiers, 121–128
Inverting Schmitt trigger,
1102–1105
Inverting summing amplifiers,
148–149
Inverting threshold comparator,
1099, 1100
Iterative method, of analysis,
193–194
JFETs. See Junction field-effect
transistors
Junction capacitance, 317–318
Junction current density, 323–325
Junction field-effect transistors
(JFETs), 29, 31, 1204, 1205
Junction potential distribution,
311–312
KCL. See Kirchhoff’s current law
Kirchoff’s current law (KCL),
458, 1214–1216
Kirchoff’s voltage law (KVL),
349, 1216–1219
KVL. See Kirchoff’s voltage law
Large design projects, 26,
1282–1283
Large-signal characteristics, 15
Large-signal voltage gain, 15
LC-coupled amplifier, 16
LC filter, 781
LC filters, 269–272, 781
LEDs. See Light-emitting diodes
Level-shifted DC amplifier,
497–501
Level-shifted MOS amplifier,
388–392
Level shifter, 279
Level shifting, 386–388, 495–501
methods for, 387–388, 496–497
MC14573 CMOS op-amp analyzing, 935–937
DC biasing for, 935
parameters of, 934
schematic for, 934
stages of, 935
Medical telesensors, 35
Medium design projects, 26, 1282
MESFETs. See Metal-Schottky barrier FETs
Metal oxide semiconductor field-effect transistors (MOSFET), 29, 336–337, 984, 1162, 1204, 1206. See also Depletion MOSFETs; n-channel depletion MOSFET; p-channel enhancement MOSFET
BJTs v., 528
DC biasing of, 357–364
depletion, 346–349, 352, 369
enhancement, 337–346, 352, 368–369
high-frequency models of, 393–395
model parameters for, 1206
models of, 349–356
small-signal analysis models of, 351–352, 395–396
Metal-Schottky barrier FETs (MESFETs), 984
MF5 universal monolithic switched-capacitor filter, 852
Microbial fuel cells, 35
Microcantilevers, 35
Microchip implants, 35
Midband voltage gain, 84–86
Miller capacitance, 78
Miller’s capacitor method, 77–80
Miller’s effect, 77–78, 503, 928
Miller’s theorem, 71–72
Mimristor, 32–33
Mini design projects, 26, 1282
Minority carrier charge distribution, 321–323
Minority carrier lifetime, 323
Minority carriers, 303–304
Missing-pulse detector, 1129–1130
Model, AC linear, 112–113
Models
AC linear, 112–113
BJT circuit, 449–455
BJT circuits, 449–455
constant-drop DC, 196–197
DC linear, 111–112
Ebers-Moll, 1267–1274
forward-biased, 328
high-frequency AC, 326–329
high-frequency BJT, 501–503
high-frequency MOSFET, 393–395
linear DC, 450
low-frequency hybrid BJT, 1262–1265
low-frequency small-signal, 199–205
MOSFET, 349–356
nonlinear macro, 113–114
op-amp circuit, 105–107
piecewise linear, of zener diodes, 215
piecewise linear DC, 197–199
practical diodes, 196–207
PSpice devices and elements for, 1200–1206
PSpice/SPICE amplifier, 87–88
PSpice/SPICE diode, 205–207
PSpice/SPICE operation amplifier, 111–114
reverse-biased, 329
small-signal AC, 450–452
small-signal hybrid, 452
Model statement, of PSpice/SPICE, 205
Modem, 1137
Modification of feedback path, 717–720
Modified basic MOSFET current source, 561
Modulation index, 781
Monostable multivibrators applications of, 1129–1130
in 555 timer, 1127–1128
MOS differential amplifiers, 566–580
MOSFET. See Metal oxide semiconductor field-effect transistors
MOSFET amplifier, 353–356
design of, 408–413
feedback in, 656
frequency response of, 393–408
MOSFET biasing circuit, 357–364
MOSFET current sources, 558–566
MOSFET models, 349–353
MOSFET switch, 356–357
MOS oscillators, two-stage, 891–894
Most positive signal detectors, 1081
Multimitter bipolar transistor, 1034
Multiple feedback filter, 840
NAND, 983, 984
Nanobiotechnology, 35
Nanofabrication, 35
Nanomedicine, 35
Nanotechnology, 34
Narrowbanding, 712
Narrow-band-pass filters, 840–843
Narrow-band-reject filters, 846–848
National Semiconductor, 789
Natural frequency, 1249
n-channel depletion MOSFET, 346
n-channel enhancement MOSFET (NMOS), 337–340, 342–343, 346, 349–350, 353, 781, 996
NE5034, 1167–1168
Negative feedback, 642, 649–650
Negative impedance converters, 158
Negative logic, 982
Negative threshold voltage, 1103
Negative voltage limiter, 1086
adjustable, 1087, 1089
NE/SE-565 PLL applications of, 1143–1146
center frequency of, 1141
diagrams of, 1142
as frequency multiplier, 1143–1144
as FSK demodulator, 1145
as SCA background music decoder, 1145–1146
NE/SE-566 VCO, 1123–1124, 1126
NE/SE-5018 D/A converter, 1163–1164
Netlists, 1206–1208
NMOS. See n-channel enhancement MOSFET;
n-channel enhancement MOSFET
NMOS differential pair, 566–575
NMOS inverters, 996–1014
comparison of, 1013–1014
with depletion load, 1005–1013
with enhancement load, 996–1005
NMOS logic circuits, 1014–1016
CMOS gates v., 1026
NAND gates, 1016
NOR gates, 1015
transmission gates, 1014–1015

Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
Index

NMOS NAND gates, 1016
NMOS NOR gates, 1015
NMOS transition gates, 1014, 1026
Noise, 11
Noise immunity, 987
Noise margins, 987–988
Noninverting, integrators, 159–160
Noninverting amplifiers, 114–121
CMRR of, 116–117
integrators, 128–137
inverting summing amplifiers, 148–149
negative impedance converters, 158
noninverting integrators, 159–160
noninverting summing amplifiers, 147–148
Noninverting summing amplifiers, 147–148
Noninverting threshold comparator, 1099
Nonlinearity error, 1163
Nonlinear macromodel, 113–114
Nonlinear ohmic region, 340–341
NOR, 983, 984
Normalized frequency, 1249
Norton’s theorem, 1226–1227
Notation, 8–10
Notch filter, 844, 846
Notch-out frequency, 847
NOT gate, 983
npn transistor, 28, 434–436, 442, 444, 448, 533, 756, 761, 775–776, 1268
n-type materials, 301–302
Nyquist plot, 703–704
Nyquist stability criterion, 703–704
Octave increase, 65
Offset error, 1163
Offset-minimizing resistance, 913
Offset voltage adjustment, 922–925
Off time, 12
Ohmic region
of depletion MOSFETs, 349
of enhancement MOSFETs, 339–341
linear, 339–340
nonlinear, 340–341
Ohm’s law, 339
OLEDs. See Organic light-emitting diodes
One-shot pulse-generating circuit, 1127
On time, 12
Op-amp. See Operational amplifiers
Op-amp applications, 128–163
AC-coupled bootstrapped voltage followers, 162–163
addition-subtraction amplifiers, 149–153
constant current sources, 159
DC millivoltmeters, 157
DC voltmeters, 156
differential amplifiers, 143–145
differentiators, 137–143
inductance simulators, 161
instrumentation amplifiers, 145–147
integrators, 128–137
inverting summing amplifiers, 148–149
negative impedance converters, 158
noninverting integrators, 159–160
noninverting summing amplifiers, 147–148
optocoupler drivers, 153–154
photodetectors, 154–155
voltage-current converters, 155–156
Op-amp circuits, 1081–1097
design, 164–165
SAH, 1157
Op-amp ideal circuits analysis, 114–128
inverting amplifiers, 120–127
noninverting amplifiers, 114–120
Op-amp PSpice/SPICE models, 111–114
AC linear model, 112–113
DC linear model, 111–112
nonlinear macromodel, 113–114
Open-circuit reverse voltage ratio, 1262
Open-circuit saturation currents, 1270
Open-circuit transimpedance, 57
Open-circuit voltage gain, 51, 351
Open-loop gain, 645, 646, 649, 656, 698
Open-loop transfer function, 649
Operational amplifiers (op-amps), 16, 656, 1098
BiCMOS, 962–974
characteristics of, 104–110
circuit design, 164–165
circuit model, 105–107
common-mode rejection ratio, 108–111, 925
comparators v., 1098
design of, 974–975
feedback in, 642, 643
frequency response, 107–108, 927–930
input biasing current for, 912–914
input offset current for, 914–917
input offset voltage for, 917–918
input resistance, 925–926
internal structure of, 910–911
offset voltage adjustment, 922–925
op-amp circuit model, 105–107
op-amp frequency response, 107–108
output resistance, 927
parameters/characteristics of practical, 911–932
power, 788–791
power supply rejection ratio, 918–919
SAH circuits with, 1157
slew rate of, 930–932
thermal voltage drift, 919–922
Operation point, of diode, 193
Optical isolators, 153
Optocoupler drivers, 153–154
OR, 983, 984
OrCad
adding text in, 1187–1188
analysis type selection, 1191–1193
beginning new project with, 1182–1183
circuit analysis process, 1181
creating netlists, 1206–1208
drawing circuit, 1182–1191
design of, 974–975
feedback in, 642, 643
frequency response, 107–108
getting/placing components in, 1183–1185
labeling components in, 1178
library files, 1208–1211
men commands for, 1190
modeling devices and elements, 1200–1206
overview, 1180–1181
PSpice simulation in, 1193
rotating components in, 1190–1191
schematic copying and capturing, 1195–1196
setting attributes in, 1188–1190
simulation results display, 1194–1195
Copyright 2011 Cengage Learning, Inc. All Rights Reserved. May not be copied, scanned, or duplicated, in whole or in part.
OrCAD (continued)
software installation, 1178–1180
varying parameters, 1196–1198
viewing schematic in, 1190
wiring components in, 1187
Organic electronics, 33–34
Organic light-emitting diodes (OLEDs), 33
Organic thin-film transistors (OTFTs), 33
Oscillations, conditions for, 863
Oscillators, 699
active-filter tuned, 899–901
AF, 867–881
amplitude stability of, 866
classification of, 863
Colpitts, 881–888
crystal, 895–899
design of, 902
feedback and, 699, 863
frequency stability of, 866
Hartley, 888–891
phase-shift, 867–870
principles of, 862–866
quadrature, 871–872
radio frequency, 881–895
ring, 878–880
two-stage MOS, 891–894
voltage-controlled, 1120–1126
Wien-bridge, 874–877
OTFTs. See Organic thin-film transistors
Output admittance, 1262
Output characteristics of depletion MOSFETs, 348–349
of enhancement MOSFETs, 342
Output decay rate, 1156
Output filters, for rectifiers, 260–272
C filters, 263–269
LC filters, 263–269
L filters, 261–263
Output offset voltage, 917
Output power AC, 240–241, 250
DC, 241, 250
/efficiency of class AB push-pull amplifiers, 767
/efficiency of complementary push-pull amplifiers, 757–758
/efficiency of transformer-coupled load push-pull amplifiers, 762–764
Output resistance, 43, 927
Output-side connection, 1098
Output stage of BiCMOS op-amp CA3130, 964
of BiCMOS op-amp CA3140, 965
of BiCMOS op-amp LH0022, 967
of BJT op-amps LM124, 942
of CMOS op-amp TLC1078, 938
of LM741 op-amp, 949–951
of small-signal AC analysis, 958–961
Output voltage, hysteresis effects of, 1107–1108
Overdrive factor, 1028
Overlap capacitance, 393
Overload protection of LM741 op-amp, 951
Parallel clippers, 276–277
Parallel resonant RLC circuits, 1240–1242
Parasitic capacitance, 393
Parasitic resistance, 205
Passband, 806
Pass-band gain, 84
Passive components capacitors, 1279
potentiometers, 1278
resistors, 1276–1278
Passive filters, active filters v., 804–805
p-channel enhancement MOSFET (PMOS), 337–338, 560, 781
Peak detectors and demodulators, 272–275
Piezoelectric crystal, 895, 896
Performance requirements, 22
Phase corrector, 848
Phase crossover frequency, 709–710
Phase detector, 1140–1141
Phase-lock loops (PLL), 1139–1143
565 IC PLL, 1141–1146
IC, 1141–1143
Phase-lock mode, 1139
Phase margin, 704–707, 709–710
Phase-shift oscillators, 867–870
Piezoelectric transducers, 154–155
Piecewise linear DC model, 197–199
Piecewise linear model of zener diodes, 215
Pinch-down voltage, 347
Plastic electronics, 33
PLL. See Phase-lock loops
PMOS. See p-channel enhancement MOSFET
pn junction, 436, 439
forward-biased, 319–323
reverse-biased, 314–318
zero-biased, 307–314
pn junction diodes, 300
pnpn triggering transistor, 2
pnp transistor, 28
pnp transistors, 434–436, 442, 448, 468, 533, 756, 775–776
Pole quality factor, 810
Pole splitting, 715–716
Positive feedback, 642, 645, 768
Positive feedback effects, 1103
Positive logic, 982
Positive threshold voltage, 1103
Positive voltage limiter, 1086, 1087
Potential divider level shifting, 387, 496
Potentiometers, 1278
Power amplifiers, 16, 53
classification of, 740–743
design of, 796
IC, 789–792
short-circuit/thermal protection for, 786–788
thermal considerations for, 792–796
Power dissipation, 794–796, 992–995
dynamic power, 993–994
static power, 993
Power electronics, 2, 10–11
Power gain, 42
Power op-amp LH0021, 789–790
Power op-amp LM380, 790–791
Power op-amps, 788–791
bridge amplifier, 791–792
IC power amplifiers, 789–791
Power rating, 220–222
Power supply rejection ratio (PSRR), 918–919
Power supply sensitivity (PSS), 919
Power transistors, 743–745, 792
heat flow in, 793–794
thermal considerations for, 792–796
thermal resistance in, 793
Practical diodes, 185–192
breakdown region, 187
characteristics of, 185–187
diode constants determination, 187–189
forward-biased region, 186
reverse-biased region, 186–187
temperature effects, 189–192
voltage-versus-current
characteristic of, 185
Practical diodes, 192–196
approximate method, 193
graphical method, 192–193
iterative method, 193–194
mathematical method, 194–196
Practical diodes modeling, 196–207
constant-drop DC model, 196–197
piecewise linear DC model, 197–199
PSpice/SPICE diode model, 205–207
Practical inverting integrator, 130
Practical op-amps
common-mode rejection ratio, 925
frequency response, 927–930
input biasing current, 912–914
input offset current, 914–917
input offset voltage, 917–918
input resistance, 925–926
offset voltage adjustment, 922–925
output resistance, 927
parameters-characteristics of, 911–932
power supply rejection ratio, 918–919
slew rate of, 930–932
thermal voltage drift, 919–922
Practical series-series feedback
network, 670–671
Practical series-shunt feedback
network, 661
Practical shunt-series feedback
network, 688–689
Practical shunt-shunt feedback
network, 680
Precision clamping circuits, 1085–1086
Precision full-wave rectifiers, 1084
Precision half-wave rectifiers, 1082–1083
Precision peak voltage detectors, 1081–1082
Probe program, 1180, 1194
Propagation delay, 991–992
of BiCMOS, 1058–1059
CMOS transmission gates, 1023–1024
Protection circuitry
of BiCMOS op-amp, LH0022, 967
of LM741 op-amp, 944
PSpice
BJTs in, 452–453, 503–504
commands for, 1209
modeling devices and elements
in, 1200–1206
MOSFETs in, 353, 395–396
OrCAD and, 1193
varying parameters for, 1196–1198
PSpice A/D, 1180
PSpice/SPICE amplifier models,
87–88
current amplifier, 87
operation, 111–114
transconductance amplifier, 88
transimpedance amplifier, 88
voltage amplifier, 87
PSpice/SPICE diode model,
205–207
model statement, 205–206
tabular representation, 206
PSRR. See Power supply rejection
ratio
PSS. See Power supply sensitivity
type materials, 302–303
Pull-up resistance, 984
Pulse response
of series CR circuits, 1234–1237
of series RC circuits, 1231–1234
Pulse stretcher, 1130
Pulse widener, 1130
Pulse-width modulation (PWM), 742, 781
Push-pull amplifiers. See also
Class AB push-pull amplifiers;
Class B push-pull amplifiers
complementary, 756–761
DC biasing of, 764
transformer-coupled load, 761–765
PWM. See Pulse-width
modulation
Q-point, 46, 193, 336, 434, 450, 529–530
Quadrature oscillators, 871–872
Quality factor, 837, 866, 895
Quantizing error, 1169
Quasi-complementary class AB
amplifiers, 775–776
Quiescent diode voltage, 193
Quiescent point, of diode, 193
Quiescent power, 993
R-2R ladder network D/A
converter, 1160–1162
Radar, 2
Radio circuits, 2
Radio frequency (RF) amplifiers, 16
Radio frequency identification
(RFID) tags, 34
Radio frequency oscillators, 881–895
Colpitts, 881–888
Hartley, 888–891
Ramp generator, 1135–1137
RC-coupled amplifier, 16
Rectifiers
diodes as, 238–260
output filters for, 260–272
precision full-wave, 1084
precision half-wave, 1082–1083
single-phase full-wave bridge, 254–260
single-phase half-wave, 238–247
Recycling, of design process, 18
Reference voltage, 208, 1097
Schmitt triggers with, 1106–1107
Regenerative feedback, 642. See also Positive feedback
Relative stability, 704–705
Resistance
AC, 200
bulk, 205
C-B, 451
dynamic, 200
input, 43, 925–926, 1262
offset-minimizing, 913
output, 43, 927
parasitic, 205
pull-up, 984
short-circuit input, 1262
small-signal, 200
thermal, 793
zener, 209
Resistive-biased common-emitter
amplifier, 471–475
Resistive-biased emitter follower, 479–483
Resistive-biased source follower, 378–380
Resistive load, 370–373
Resistors, 1201, 1276–1278
Resistor-transistor logic (RTL), 1026
Index

Resolution, 1163
Resonant circuits, 1238–1239
Resonant frequency, 1255
Reverse-biased diode, 180
Reverse-biased model, 329
Reverse-biased pn junction, 314–318
breakdown condition, 315–316
depletion region width, 316–317
junction capacitance, 317–318
Reverse-biased region, of practical junction,
pn–pn, 303–304
p–n, 301–302
n–p, 301–302
Reverse drift current, 308
Reverse saturation current, 324
RF amplifiers. See Radio frequency amplifiers
RFID tags. See Radio frequency identification tags
Ring oscillators, 878–880
Ripple elimination, 1153
Rise time, 12, 46–47, 48, 991, 1028
RLC circuit, 1181
Roll-off rate, 807
Root locus, 701
RTL. See Resistor-transistor logic
SAH. See Sample-and-hold circuits
Sallen-Key circuit, 822, 829, 831
modified, 824
Sample-and-hold circuits (SAH), 1155–1158
integrated circuits, 1157–1158
op-amp circuits, 1157
Sampling time, 7
SAR. See Successive-approximation register
Saturating charge, 1029
Saturation current, 444–447
Saturation-logic families, 1027
Saturation region, 347
of depletion MOSFETs, 347, 349
of enhancement MOSFETs, 341
Sawtooth-wave generators, 1117–1119
SCA. See Subsidiary carrier authorization decoder
Scaling amplifier, 148
Scaling summer circuit, 149
Schmitt triggers, 1101–1109
hysteresis and, 1107–1109
inverting, 1102–1105
noninverting, 1105–1106
with reference voltage, 1106–1107
Schottky TTL NAND gates, 1047–1049
SCR. See Silicon-controlled rectifier
Second-order Butterworth filter, 852–853
Second-order filters, 813
Second-order high-pass filters, 831–833
Second-order low-pass filters, 822–826
Second-order parallel RLC circuits, 1254–1258
Second-order series RLC circuits, 1249–1254
See Basic circuits
first-order law-pass RC circuits, 1244–1246
frequency response of first- and second-order circuits, 1243–1244
Semiconductor, 33
Semiconductor diodes, 27, 180
diode data sheets, 222–225
ideal diodes, 180–183
light-emitting diodes, 220
power rating, 220–222
practical diodes, 184–192
transfer characteristics of diode circuits, 183–184
zener diodes, 208–219
Semiconductor materials, 300–307
of depletion MOSFETs, 347, 349
of enhancement MOSFETs, 341
Signal waveforms, 746, 761–762
Signal amplification types, 15–16
Semiconductor, 33
Shockley diode equation, 185
Short-circuit current gain, 53
Short-circuit forward-transfer current ratio, 1262
Short-circuit input resistance, 1262
Short-circuit method, 80
low-frequency, 80–82
Short-circuit protection, 786–787
Short-circuit saturation currents, 1270
Short design projects, 25
Shunt regulator, 209
analysis of, 688–691
designing circuit for, 694–697
ideal, 688
practical, 688–689
Shunt-shunt feedback, 644, 652, 653, 654, 656–657, 677
analysis of, 678–686
ideal, 678–680
practical, 680
Signal amplification types, 15–16
Signal waveforms, 746, 761–762
Signetics Corporation, 1126
Silicon-controlled rectifier (SCR), 2
Silicon transistor, 2
Simple diode circuit, 192
Simulated resistor, 850
Single-base resistor biasing, 459–460
Single-phase bridge rectifier, with L filter, 261
Single-phase full-wave bridge rectifier, 254–260
Small common-mode signal, 607–609
Small differential signal, 606–607, 614–615
Small-signal AC models, 450–452
Small-signal analysis of BJT circuits, 453–455
Series-shunt feedback, 643, 652, 653, 656–657
analysis of, 658–667
designing circuit for, 692–693
ideal, 658–660
practical, 661
Servo amplifier, 16
Settling time, 1156, 1163
analysis of, 688–691
designing circuit for, 694–697
ideal, 688
practical, 688–689
Series-parallel feedback, 644, 652, 653, 654, 656–657, 677
analysis of, 678–686
ideal, 678–680
practical, 680
Series-parallel feedback, 644, 652, 653, 654, 656–657, 677
analysis of, 678–686
ideal, 678–680
practical, 680
Series-parallel feedback, 644, 652, 653, 654, 656–657, 677
analysis of, 678–686
ideal, 678–680
practical, 680
Series-parallel feedback, 644, 652, 653, 654, 656–657, 677
analysis of, 678–686
ideal, 678–680
practical, 680
Series-parallel feedback, 644, 652, 653, 654, 656–657, 677
analysis of, 678–686
ideal, 678–680
practical, 680
Servo amplifier, 12
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Servo amplifier, 16
Transfer characteristics (continued)
  of diode circuits, 183–184
  of emitter followers, 746
  of enhancement MOSFETs, 342
  with negative feedback, 650–651
  of Schmitt trigger, 1103, 1106
  of bipolar junction transistor; Diode-transistor logic; Junction field-effect transistors; Resistor-transistor logic
  BJT, 434–455, 457–467, 501–508, 528, 533
  Darlington pair, 491–495
  depletion metal oxide
  semiconductor field-effect, 30–31
  field-effect, 29, 106, 336
  forward current gain of, 28
  MOSFET, 336–337, 357–364, 528, 1162
  multiemitter bipolar, 1034
  npn, 28, 434–436, 442, 444, 448, 533, 756, 761, 775–776, 1268
  organic thin-film, 33
  pnp, 28, 434–436, 442, 448, 468, 533, 756, 775–776
  pnpn triggering, 2
  power, 743–745, 792
  silicon, 2
  unipolar field-effect, 2
  Transistor saturation current, 1269
  Transistor-transistor logic (TTL)
  TTL gates, 1026, 1033–1034
  CMOS driven by, 1060–1062
  CMOS driving, 1062–1063
  high-speed, 1042–1047
  Schottky, 1047–1049
  standard, 1034–1042
  Transition capacitance, 327
  Transition frequency, 505
  Transition region, 982, 986
  Transition frequency, 505
  Transition capacitance, 327
  Triode vacuum tube, 2
  Truth table, 984
  TTL. See Transistor-transistor logic
  TTL NAND gate, 1034–1039
  TTL driving CMOS, 1060–1062
  TTL driving CMOS, 1060–1062
  Transistor saturation current, 1269
  Unsymmetrical limiter, 215
  Unipolar field-effect transistor, 2
  Unipolar field-effect transistor, 2
  Upper threshold voltage, 1103
  VA. See Video amplifier
  Vacuum tubes, 2
  Variable-shift clamps, 281–282
  Varicaps, 327
  VBE multiplier circuit, 772–774
  VCCS. See Voltage-controlled voltage source
  VCCS. See Voltage-controlled voltage source
  VCO. See Voltage-controlled oscillator
  Very-large-scale integrated circuits (VLSI), 336
  V/F converter. See Voltage-to-frequency converters
  Video amplifier (VA), 16
  VLSI. See Very-large-scale integrated circuits
  Voltage
  AC-coupled bootstrapped followers, 162–163
  adjustable-, limiters, 1086–1092
  adjustable negative, limiter, 1087, 1089
  adjusting offset, 922–925
  amplifiers, 15, 51–52, 87
  biasing, 436
  breakdown, 208
  cascaded, amplifiers, 59–60
  channel modulation, 352
  current-controlled, source, 88
  cut-in, 186
  diode, multipliers, 284–287
  Early, 442, 451
  error, 1139
  fixed-, limiters, 1086
  frequency-to-, converters, 1151–1155
  full-wave, doubler circuit, 286
  half-wave, doubler circuit, 285
  large-signal, gain, 15
  lower threshold, 1103
  midband, gain, 84–86
  negative, limiter, 1086
  negative, limiter, 1086
  negative threshold, 1103
  offset, adjustment, 922–925
  open-circuit, gain, 51
  open-circuit reverse, ratio, 1262
  output, 1107–1108
  output offset, 917
  pinch-down, 347
  positive, limiter, 1086, 1087
  positive threshold, 1103
  precision peak, detectors, 1081–1082
  quiescent diode, 193
  reference, 208, 1097
  Schmitt triggers with reference, 1106–1107
  small-signal, gain, 15, 47
  switching, 1106
thermal, 186
thermal, drift, 919–922
threshold, 186, 339, 1001
transition, 604
turn-on, 186
upper threshold, 1103
zener, 208
zener, limiter, 1093–1094
Voltage-controlled current source (VCCS), 88, 1122–1123
Voltage-controlled oscillator (VCO), 1120–1126
charging mode of, 1120
circuit implementation of, 1121–1122
discharging mode of, 1121
NE/SE-566, 1123–1125
Voltage-controlled voltage source (VCVS), 51–52, 87
Voltage-current converters, 155–156
Voltage doublers, 284–286
Voltage-feedback ratio, 1262
Voltage follower, 116
Voltage gain, 41–42, 47
Voltage multipliers, 284–287
voltage doublers, 284–286
voltage triplers and quadruplers, 286–287
Voltage reference diode, 20
Voltage regulator, 209
Voltage-sensing/current-comparing feedback, 644. See also Shunt-shunt feedback
Voltage-sensing/voltage-comparing feedback, 653. See also Series-shunt feedback
Voltage-to-frequency (V/F) converters, 1120, 1147–1151, 1155
Voltage transfer characteristic (VTC), 986–987, 1027
Voltage transfer function, 805
Voltage triplers and quadruplers, 286–287
VTC. See Voltage transfer characteristic
Weighted-resistor D/A converter, 1159–1160
Weighted summer circuit, 149
Wideband amplifier, 16
Wide-band-pass filters, 837–840
Wide-band-reject filters, 844–846
Widlar current source, 591–595
Wien-bridge oscillators, 874–877
Wilson current source, 564–565, 596–601
Zener breakdown, 315
Zener diodes, 208–219, 787
data sheet for, 224–225
temperature effects of, 218–219
zener limiters, 214–218
zener regulator, 209–211
Zener effect, 315
Zener hard limiter, 1094, 1096
Zener level shifting, 388, 497
Zener limiters, 214–218
Zener regulator, 209–211
design of, 211–214
Zener resistance, 209
Zener voltage, 208
Zener voltage limiter, 1093–1094
Zero-biased pn junction, 307–314
built-in junction potential, 308–310
electric field distribution, 310–311
junction potential distribution, 311–312
space charge depletion width, 312–314
Zero-crossing detectors, 1100–1101
Zero quality factor, 810
Zero-value method, high-frequency, 82–84