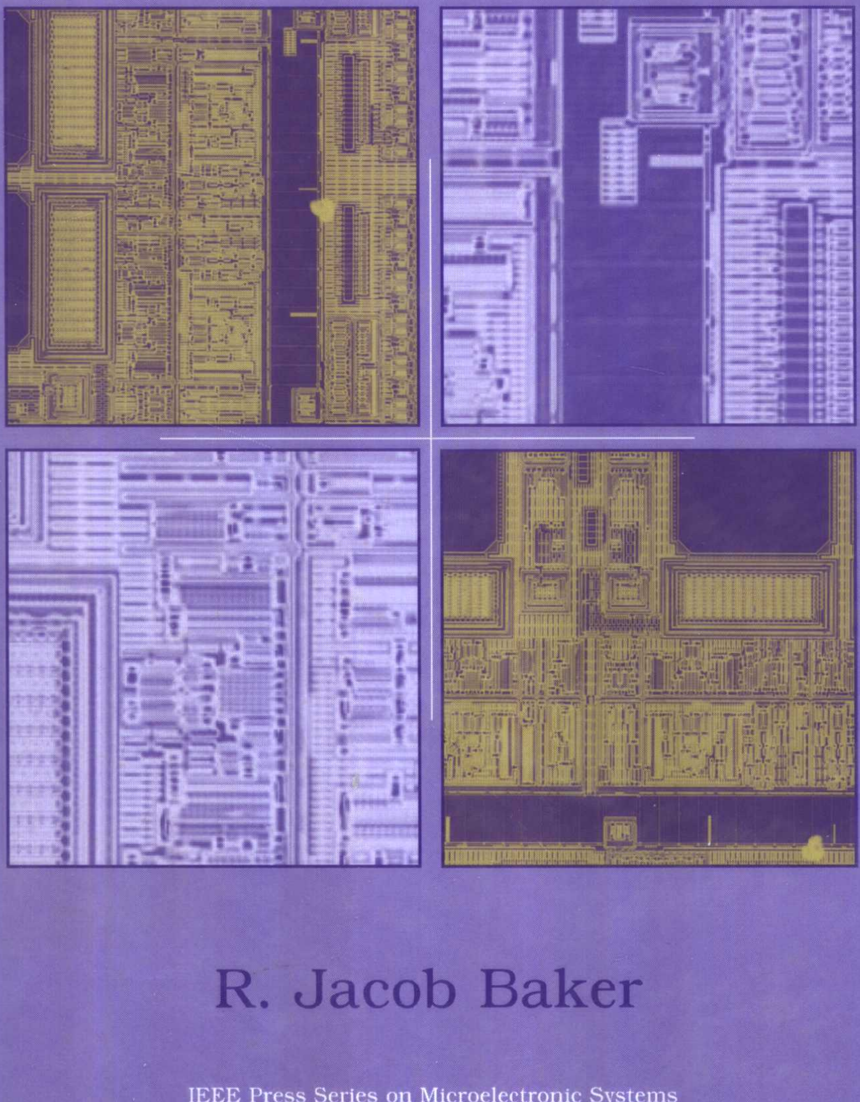
The essential companion to the bestselling
CMOS: Circuit Design, Layout, and Simulation

CMOS

MIXED-SIGNAL CIRCUIT DESIGN



R. Jacob Baker

IEEE Press Series on Microelectronic Systems
Stuart K. Tewksbury and Joe E. Brewer, *Series Editors*

CMOS
Mixed-Signal Circuit Design

R. Jacob Baker

Volume II of CMOS: Circuit Design, Layout, and Simulation

IEEE Press Series on Microelectronic Systems
Stuart K. Tewksbury, *Series Editor*
IEEE Solid-State Circuits Society, *Sponsor*

IEEE
IEEE Press

WILEY-
INTERSCIENCE

A JOHN WILEY & SONS, INC., PUBLICATION

This book is printed on acid-free paper. ♾

Copyright © 2002 by The Institute of Electrical and Electronics Engineers, Inc.

No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, scanning or otherwise, except as permitted under Section 107 or 108 of the 1976 United States Copyright Act, without either the prior written permission of the Publisher, or authorization through payment of the appropriate per-copy fee to the Copyright Clearance Center, 222 Rosewood Drive, Danvers, MA 01923, (978) 750-8400, fax (978) 750-4744. Requests to the Publisher for permission should be addressed to the Permissions Department, John Wiley & Sons, Inc., 605 Third Avenue, New York, NY 10158-0012, (212) 850-6011, fax (212) 850-6008, E-Mail: PERMREQ @ WILEY.COM.

For ordering and customer service, call 1-800-CALL WILEY.

Library of Congress Cataloging-in-Publication Data is available.

Baker, R. Jacob,
  CMOS: Mixed-Signal Circuit Design, volume II/
  R. Jacob Baker
    p. cm.
    Includes bibliographical references and index.
    ISBN 0-471-22754-4 (cloth: alk. paper)

Printed in the United States of America

10 9 8 7 6 5 4 3 2 1

To

Julie, Kyri, and Josh

Preface to Volume II

Why a second volume? Why not write a separate, unassociated, book covering mixed-signal circuit design? The answer to these questions comes from my desire to keep from duplicating material available in *CMOS Circuit Design, Layout, and Simulation* (the, sometimes affectionately called, "purple" book containing the first 29 chapters). My goal, when writing this second volume, was to provide a quality textbook and reference that contains material that isn't covered in other textbooks and to couple this material with online learning aids. The supporting companion website, **http://cmosedu.com**, provides worked solutions to the problems, SPICE simulation netlist examples, and discussions concerning mixed-signal circuit design.

Book Organization

The power of a mixed-signal circuit design, and perhaps the reason they are replacing analog-only designs in the implementation of analog interfaces, comes from the marriage of analog circuits with digital signal processing. Chapter 30 uses the topic of data converter modeling as a vehicle to illustrate some fundamental signal processing topics. In addition, models are developed to aid in the understanding of quantization noise. Chapter 31 discusses signal-to-noise ratio (SNR) and ways of improving SNR using filtering (averaging). The chapter provides a practical discussion of some of the basic tools the mixed-signal designer uses to improve SNR. Decimation, interpolation, and feedback (and the concept of pushing quantization noise to higher-frequencies, i.e., noise-shaping, so that it can be filtered out using a digital filter) are also discussed. Because of the importance of noise-shaping to the design of analog interfaces Ch. 32 is devoted entirely to noise-shaping data converters (often called delta-sigma or sigma-delta data converters). The presentation mixes theory with practical implementations and examples to illustrate the operation of these data converters.

此为试读，需要完整PDF请访问：www.ertongbook.com

Chapter 33 covers circuit design using submicron CMOS devices. This chapter was born, in part, after I was asked, "Why didn't you use a more modern CMOS process in the purple book?" Submicron devices don't follow the square-law MOSFET model and so hand calculations based on equations derived from the square-law models are somewhat meaningless. I quickly realized, however, that many designers don't do hand calculations and so the significant deviations between long- and short-channel behavior aren't readily apparent. Similarly, students are often asked to compare their hand calculations to SPICE simulation results using the level 1 model. Of course, the simulations and hand calculations match but have little, if anything, to do with the performance of the submicron CMOS circuits in actual silicon.

At the risk of stating the obvious, accurate models for submicron CMOS devices are extremely important. In Ch. 33 the EKV model is used. The EKV model does a good job of modeling the device's transition from weak to strong inversion. This is important when simulating modern data converter or mixed-signal circuits where the threshold voltage doesn't scale with the power supply voltage (i.e., the circuitry is operating a larger percentage of the time in the subthreshold region). As seen in Ch. 33, hand calculations can be performed with transconductance or output resistance values read off of plots generated experimentally or from simulations using the accurate EKV model.

Chapter 34 covers the implementation of data converters. Its purpose is to provide ideas and discussions for implementing the data converter topologies discussed in Ch. 29. Matching, offsets, gains, bandwidths, and topologies are discussed to provide insight into the design of Nyquist rate data converters.

Chapter 35 covers the design of integrated filters. This topic is a book in itself so I've picked the most relevant subjects and attempted to provide practical insight useful when implementing the filters in CMOS technology. The material is complete and useful enough so that, once the material in the chapter is understood, the reader should have little difficulty understanding the merits or trade-offs of any filtering topology. A special twist on the material is the way the topic of digital filtering is presented (hopefully, it is very practical).

Significant effort, in these first six chapters, has been put into integrating theory and examples with simulation results (using actual numbers, i.e., not just symbolic representations). The reader serious about learning the material will find the ability to modify a simulation netlist (downloaded from http://cmosedu.com) and look at the resulting output a very useful learning tool. However, while hand calculations and simulations are very important when learning and designing CMOS circuits, equally, if not more, important is the actual building and testing of the mixed-signal circuits. Chapter 36 provides some examples of actual circuits built "at the bench" to provoke thought and interest. The hope is that instead of just simulating a design, the engineer/student may also want to build and test a representation of the circuit using discrete components. While these discrete circuits won't provide an exact representation of the actual integrated circuits, they can provide insight into the limitations of a particular design. They can also be used to get aquainted with the test equipment and the possible loading that might be introduced into the circuit when probing.

Acknowledgments

Finally, and perhaps most importantly, I would like to thank the reviewers, contributors, colleagues, and (especially!) students who helped make this book possible: Jake Anderson, Brian Bergeson, Tom Bernhard, Kurt Beigel, Jan Bissey, Bill Black, Ken Boorom (special thanks for the very detailed reviews), Dave Boyce, Liz Brauer, Joe Brewer, Curtis Cahoon, Mansun Chan, John Chen, Cathy Faduska, Chris Fayomi, Ed Fong, Dan Foty, Rich Friel, Paul Furth, Randy Geiger, David Goldman, Neil Goldsman, Tyler Gomm, Mike Green, John Griffin, Joe Hartman, Francis Heck, Rick Hilton, Gexin Huang, Glen Hush, Pandurang Irkar, Alok Jain, Brent Keeth, Christy Kuhnen, Harry Li, Yantao Ma, Paul Mason, Mary Miller, Bob Moehrke, Ken Moore, Sugato Mukherjee, Fred Perner, Adrian Ong, Zuxu Qin, Jeremy Rice, Ben Rivera, Brandon Roth, Savang Sengkhamyong, Mir Seyyedy, Brian Shirley, Joseph P. Skudlarek, Jim Slupe, Ken Smith, Mike Smith (special thanks for WinSPICE), Liu Song, Stuart Tewksbury, Lisa Van Horn, Tony VenGraitis, and Tom Voshell.

Boise, ID, January 2002 R. Jacob (Jake) Baker

Contents

Preface to Volume II xv

Chapter 30 Data Converter Modeling 1

30.1 Sampling and Aliasing: A Modeling Approach 2

30.1.1 Impulse Sampling 2

A Note Concerning the AAF and RCF 4

Time-Domain Description of Reconstruction 9

Using SPICE for Spectral Analysis (Looking at the Spectrum of a Signal) 13

Representing the Impulse Sampler's Output in the Z-Domain 16

An Important Note 18

30.1.2 The Sample and Hold 19

SPICE Modeling the Sample and Hold 19

S/H Spectral Response 20

Circuit Concerns for Implementing the S/H 25

30.2 SPICE Models for DACs and ADCs 26

30.2.1 The Ideal DAC 26

SPICE Modeling Approach 28

30.2.2 The Ideal ADC 32

Summary 35

30.3 Quantization Noise 35

30.3.1 Viewing the Quantization Noise Spectrum Using Simulations 37

An Important Note 39

RMS Quantization Noise Voltage 40

Treating Quantization Noise as a Random Variable 42

Calculating RMS Quantization Noise Voltage from a Spectrum 43

The DFT's Relationship to the Continuous Time Fourier Transform 51

30.3.2 Quantization Noise Voltage Spectral Density 52

Reducing Quantization Noise Using Averaging 54

The Noise Spectral Density View of Averaging 55

An Important Note 56

Practical Implementation of Averaging in ADCs 57

Chapter 31 Data Converter SNR 63

31.1 Data Converter SNR: An Overview 64

31.1.1 Effective Number of Bits 64

Signal-to-Noise Plus Distortion Ratio 66

Spurious-Free Dynamic Range 68

Dynamic Range 68

Specifying SNR and SNDR 69

31.1.2 Clock Jitter 70

Using Oversampling to Reduce Sampling Clock Jitter Stability Requirements 72

A Practical Note 74

Modeling Clock Jitter with SPICE 74

Using Our SPICE Jitter Model 78

31.1.3 A Tool: The Spectral Density 79

The Spectral Density of Deterministic Signals: An Overview 79

The Spectral Density of Random Signals: An Overview 82

Specifying Phase Noise from Measured Data 86

31.2 Improving SNR Using Averaging 87

31.2.1 Using Averaging to Improve SNR 87

Spectral Density View of Averaging Revisited 88

An Important Observation 92

Jitter and Averaging 93

Relaxed Requirements Placed on the Antialiasing Filter 94

Data Converter Linearity Requirements 95

Adding a Noise Dither to the ADC Input 97

The Z-Plane 99

31.2.2 Decimating Filters for ADCs 106
The Accumulate and Dump 106
Averaging without Decimation 111
Relaxed Requirements Placed on the Antialiasing Filter Revisited 113
Implementing Averaging Filters 114
Aliasing Concerns When Using Decimation 120
A Note Concerning Stability 122
Decimating Down to 2*B* 124
31.2.3 Interpolating Filters for DACs 126
The Dump and Interpolate 127
Practical Implementation of Interpolators 128
31.2.4 Bandpass and Highpass Sinc Filters 132
Canceling Zeroes to Create Highpass and Bandpass Filters 132
Frequency Sampling Filters 135
31.3 Using Feedback to Improve SNR 136
31.3.1 The Discrete Analog Integrator 136
A Note Concerning Block Diagrams 138
31.3.2 Modulators 140

Chapter 32 Noise-Shaping Data Converters 149

32.1 Noise-Shaping Fundamentals 149
32.1.1 SPICE Models 149
Nonoverlapping Clock Generation and Switches 150
Op-Amp Modeling 151
SPICE Modeling a 1-Bit ADC 153
32.1.2 First-Order Noise-Shaping 154
A Digital First-Order NS Demodulator 157
Modulation Noise in First-Order NS Modulators 158
RMS Quantization Noise in a First-Order Modulator 162
Decimating and Filtering the Output of a NS Modulator 164
Implementing the Sinc Averaging Filter Revisited 167
Analog Sinc Averaging Filters using SPICE 169
Using our SPICE Sinc Filter Model 171
Analog Implementation of the First-Order NS Modulator 172
The Feedback DAC 174
Understanding Averaging and the Use of Digital Filtering with the Modulator 174
Pattern Noise from DC Inputs (Limit Cycle Oscillations) 179

Integrator and Forward Modulator Gain 182
Comparator Gain, Offset, Noise, and Hysteresis 186
Op-Amp Gain (Integrator Leakage) 186
Op-Amp Settling Time 188
Op-Amp Offset 189
Op-Amp Input Referred Noise 190
Practical Implementation of the First-Order NS Modulator 191
Fully-Differential Modulator with a Single-Ended Input 193
32.1.3 Second-Order Noise-Shaping 194
Second-Order Modulator Topology 195
Integrator Gain 198
Implementing Feedback Gains in the DAI 202
Using Two Delaying Integrators to Implement the Second-Order Modulator 205
Selecting Modulator (Integrator) Gains 207
Understanding Modulator SNR 208
32.2 Noise-Shaping Topologies 210
32.2.1 Higher-Order Modulators 210
M^{th}-Order Modulator Topology 211
Decimating the Output of an M^{th}-Order NS Modulator 212
Implementing Higher-Order, Single-Stage, Modulators 213
32.2.2 Multibit Modulators 215
Simulating a Multibit NS Modulator Using SPICE 215
Multibit Demodulator (Used in a NS DAC) Implementation (Error Feedback) 217
Implementation Concerns 220
32.2.3 Cascaded Modulators 221
Second-Order (1-1) Modulators 221
Third-Order (1-1-1) Modulators 223
Third-Order (2-1) Modulators 223
Implementing the Additional Summing Input 226
32.2.4 Bandpass Modulators 228
Implementing a Bandpass Modulator 229
Chapter 33 Submicron CMOS Circuit Design 235
33.1 Submicron CMOS: Overview and Models 236
33.1.1 CMOS Process Flow 236
33.1.2 Capacitors and Resistors 239
Using a MOSFET as a Capacitor 239
Using a Native or Natural MOSFET Capacitor 239

此为试读，需要完整PDF请访问：www.ertongbook.com

The Floating MOS Capacitor 239
Metal Capacitors 241
An Important Note 244
Resistors 245
33.1.3 SPICE MOSFET Modeling 248
Model Selection 248
Model Parameters 252
An Important Note 254
A Note Concerning "Long L MOSFETs" 255
33.2 Digital Circuit Design 255
33.2.1 The MOSFET Switch 256
Bidirectional Switches 258
A Clocked Comparator 261
Common-Mode Noise Elimination 264
33.2.2 Delay Elements 271
33.2.3 An Adder 274
33.3 Analog Circuit Design 276
33.3.1 Biasing 276
Selecting the Excess Gate Voltage 276
Selecting the Channel Length 277
Small-Signal Transconductance, g_m 279
MOSFET Transition Frequency, f_T 279
The Beta Multiplier Self-Biased Reference 280
33.3.2 Op-Amp Design 284
Output Swing 284
Slew-rate Concerns 287
Differential Output Op-Amp 295
33.3.3 Circuit Noise 298
Thermal Noise 298
The Spectral Characteristics of Thermal Noise 299
Noise Equivalent Bandwidth 301
MOSFET Noise 303
Noise Performance of the Source-Follower 304
Noise Performance of a Cascade of Amplifiers 305
DAI Noise Performance 306
Chapter 34 Implementing Data Converters 311
34.1 R-2R Topologies for DACs 312
34.1.1 The Current-Mode R-2R DAC 312

34.1.2 The Voltage-Mode R-2R DAC 314
34.1.3 A Wide-Swing Current-Mode R-2R DAC 316
  DNL Analysis 317
  INL Analysis 318
  Switches 319
  Experimental Results 319
  Improving DNL (Segmentation) 321
  Trimming DAC Offset 323
  Trimming DAC Gain 325
  Improving INL by Calibration 326
34.1.4 Topologies Without an Op-Amp 328
  The Voltage-Mode DAC 328
  Two Important Notes Concerning Glitches 331
  The Current-Mode (Current Steering) DAC 332
34.2 Op-Amps in Data Converters 334
  Gain Bandwidth Product of the Noninverting Op-Amp Topology 335
  Gain Bandwidth Product of the Inverting Op-Amp Topology 336
34.2.1 Op-Amp Gain 337
34.2.2 Op-Amp Unity Gain Frequency 339
34.2.3 Op-Amp Offset 339
  Adding an Auxiliary Input Port 339
34.3 Implementing ADCs 343
34.3.1 Implementing the S/H 343
  A Single-Ended to Differential Output S/H 345
34.3.2 The Cyclic ADC 351
  Comparator Placement 352
  Implementing Subtraction in the S/H 354
  Understanding Output Swing 357
34.3.3 The Pipeline ADC 359
  Using 1.5 Bits/Stage 361
  Capacitor Error Averaging 368
  Comparator Placement 374
  Clock Generation 377
  Offsets and Alternative Design Topologies 378
  Dynamic CMFB 382
  Layout of Pipelined ADCs 385

Chapter 35 Integrator-Based CMOS Filters 393

35.1 Integrator Building Blocks 393

35.1.1 Lowpass Filters 393

35.1.2 Active-RC Integrators 395

Effects of Finite Op-Amp Gain Bandwidth Product 398

Active-RC SNR 403

35.1.3 MOSFET-C Integrators 404

Why use an Active Circuit (an Op-Amp) 406

35.1.4 g_m-C (Transconductor-C) Integrators 407

Common-Mode Feedback Considerations 410

A High-Frequency Transconductor 411

35.1.5 Discrete-Time Integrators 411

An Important Note 415

Exact Frequency Response of a First-Order Discrete- Time Digital (or Ideal SC) Filter 416

35.2 Filtering Topologies 418

35.2.1 The Bilinear Transfer Function 418

Active-RC Implementation 419

Transconductor-C Implementation 419

Switched-Capacitor Implementation 420

Digital Filter Implementation 421

The Canonic Form (or Standard Form) of a Digital Filter 425

35.2.2 The Biquadratic Transfer Function 429

Active-RC Implementation 431

Switched-Capacitor Implementation 437

High Q 438

Q Peaking and Instability 443

Transconductor-C Implementation 445

The Digital Biquad 446

35.3 Filters using Noise-Shaping 448

Removing Modulation Noise 448

Implementing the Multipliers 451

Chapter 36 At the Bench 459

36.1 A Push-Pull Amplifier 459

Deadbug Prototyping 460

Probing 462

Testing the Circuit 463

36.2 A First-Order Noise-Shaping Modulator 467
Prototyping the Modulator 469
36.3 Measuring 1/f Noise 472
MOSFET Noise 474
Input-Referred Noise Voltage 475
Chopper Stabilization 476
36.4 A Discrete Analog Integrator 479
Clock Generation 480
Prototyping the Filter 482
35.5 Quantization Noise 485
Prototyping the ADC Circuit 488
Index **493**
About the Author **502**

Chapter 30

Data Converter Modeling

In this chapter we continue our discussion of data converters by discussing methods to model ideal data converters and their components using SPICE. The main goal of this chapter is to provide tools for evaluating mixed-signal designs with large complexity, which can be used in design evaluation and later in the book. In particular, we will generate SPICE models, using behavioral elements, for ideal analog-to-digital converters (ADCs) and digital-to-analog converters (DACs) blocks. This allows us to analyze the performance of a mixed-signal circuit block in a SPICE simulation within a reasonable amount of time. For example, if we have designed a DAC at the transistor level and want to use SPICE to simulate its operation, under various temperatures and matching conditions, we may apply a digital input code generated from our ideal ADC with a sinewave input as seen in Fig. 30.1. Similarly, given a digital signal processing (DSP) system, we can drop our ideal DAC into the simulation at any point where there is a digital word and get an analog waveform output.

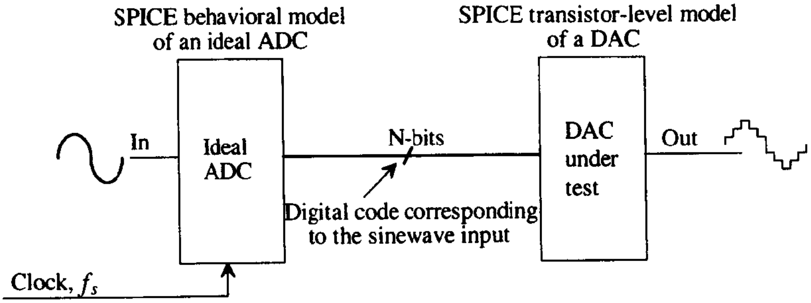


Figure 30.1 Generating the sinewave digital code for DAC simulation with an ideal ADC.

Also, in this chapter we look at how the analog-to-digital and digital-to-analog conversion process affects the signals in the system. Figure 30.2 shows the basic conversion process. We will make extensive use of the spectral analysis capability (discrete fourier transform or DFT) available in SPICE to look at the digital data (and analog signals) in the frequency domain.

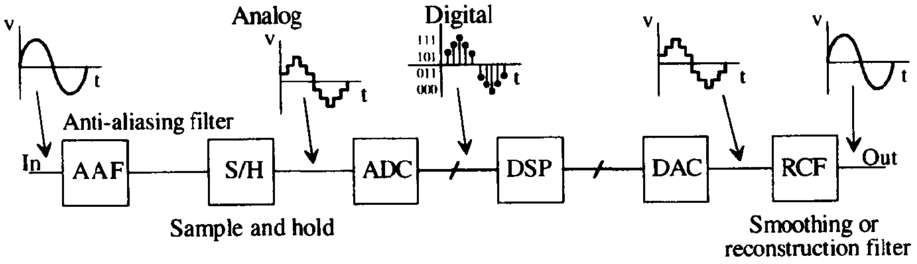


Figure 30.2 Signals resulting from A/D and D/A conversion in a mixed-signal system.

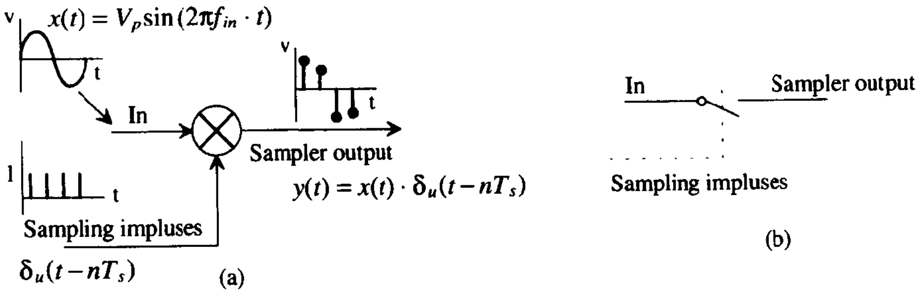
30.1 Sampling and Aliasing: A Modeling Approach

In this section we discuss how sampling a signal changes the signal's spectrum. We also discuss how to model the sampling process in SPICE.

30.1.1 Impulse Sampling

Consider the simple sampling gate shown in Fig. 30.3a. Let's assume we apply a sinewave input, $x(t)$, to this sampling gate of the form, $V_p \sin(2\pi f_{in} \cdot t)$ (for the moment, a single frequency input). The output of the sampling gate (a.k.a. sampler), $y(t)$, is the product of the input and a sampling unit impulse signal or

$$y(t) = \sum_{n=-\infty}^{\infty} V_p \sin(2\pi f_{in} \cdot nT_s) \cdot \delta_u(t - nT_s) \qquad (30.1)$$



(a) (b)

Figure 30.3 (a) Simple sampling gate and (b) SPICE implementation of a sampling gate.

此为试读，需要完整PDF请访问：www.ertongbook.com