An Analog Electronics Companion

Basic Circuit Design for Engineers and Scientists

Scott Hamilton

An Analog Electronics Companion

Basic Circuit Design for Engineers and Scientists

Scott Hamilton
Department of Physics and Astronomy
University of Manchester

CAMBRIDGE
UNIVERSITY PRESS

PUBLISHED BY THE PRESS SYNDICATE OF THE UNIVERSITY OF CAMBRIDGE
The Pitt Building, Trumpington Street, Cambridge, United Kingdom

CAMBRIDGE UNIVERSITY PRESS
The Edinburgh Building, Cambridge CB2 2RU, UK
40 West 20th Street, New York, NY 10011–4211, USA
477 Williamstown Road, Port Melbourne, VIC 3207, Australia
Ruiz de Alarcón 13, 28014 Madrid, Spain
Dock House, The Waterfront, Cape Town 8001, South Africa

http://www.cambridge.org

First published 2003

Printed in the United Kingdom at the University Press, Cambridge

Typeface Times NR MT 10.5/14 pt System QuarkXPress™ [SE]

A catalogue record for this book is available from the British Library

Library of Congress Cataloguing in Publication data
Hamilton, Scott, 1932–
An analog electronics companion / Scott Hamilton.
p. cm.
Includes bibliographical references and index.
ISBN 0 521 79838 8
1. Electronics. I. Title.
TK7816 .H35 2001
621.381–dc21 2001043081

ISBN 0 521 79838 8 hardback

Preface

This morning my newspaper contained the obituary of Sir Alan Hodgkin, Nobel Laureate together with Andrew Huxley and John Eccles, in physiology and medicine. What has this to do with our interest in electronics? Well, the prize was awarded for the elucidation of the mechanism of the propagation of electrical impulses along neural fibres, the basis of our own internal electronic system. Before the understanding of these mechanisms the position in this field would have been akin to that of Oersted, Faraday, Ohm, Ampère and Maxwell in trying to understand conduction, since at that time the electron was unknown and, for example, they imagined that an electric field somehow created charge to allow for conduction. The intimate interactions between electrical and biochemical activity are nowadays of great interest with the possibility of constructing electronic–biological systems. The consequences of Maxwell's synthesis of electricity, magnetism and light and the prediction of electromagnetic waves have been immense. Almost everything we shall discuss hinges ultimately on his discoveries and they still stand as a pinnacle in the field of physics:

> If you have bought one of those T-shirts with Maxwell's equations on the front, you may have to worry about its going out of style, but not about its becoming false. We will go on teaching Maxwellian electrodynamics as long as there are scientists.
>
> Steven Weinberg, Physicist, Nobel Laureate (New York Review of books)

Why another book on electronics? Twenty years ago I wrote one prompted by the burgeoning production of integrated circuits and the thought that many, like myself, who were not electronic engineers nevertheless needed to be able to develop circuits for our own use. It has been said that the threat of imminent execution concentrates the mind wonderfully. On a very much lower level, having to present a coherent account of all the various topics one thinks important is a very searching test of one's understanding as one finds all the holes in one's knowledge, so there has been a considerable learning process to go through. Age does have some advantages, one of them being the time to think more deeply, to understand more clearly and to fill in the missing bits. As Kierkegaard observed, 'Life can only be understood backwards; but it must be lived forwards'.

The world's first synthesized drug dates back to Hippocrates, who reported that a willow bark extract relieved aches. On August 10th 1897, Felix Hoffman, a chemist for Bayer, created a synthetic version, now called aspirin. This has alleviated many headaches and one may hope that this book may also.

Now, with the centenary of the discovery of the electron by J. J. Thomson, also in 1897, an essential ingredient in this subject, it seemed appropriate to consider an update. However, in the interval one has become older and more experienced even if not wiser, and one's point of view as to what is important has necessarily changed. This is not a textbook; it is not a serial and coherent treatment of electronics topics; it is essentially a prompt and a companion and a reminder of many things and techniques you may not know or have forgotten (at least those which I find useful and have not forgotten). Experienced engineers will possibly find little new of interest, but I aim more, as before, at the many on the margins or who have not had access or time to learn all they would have liked to. The other very significant development in more recent times has been the use of computer techniques for the simulation of electronic circuits. This has so enabled the analysis of systems compared with what before could reasonably be done by hand, as to make non-access to such a facility a severe disadvantage. Since the software can run successfully on PCs, and the cost is not prohibitive, it allows almost all to make use of it. Again, the book is not intended as a manual on how to use SPICE, the generic form of the software, but rather some indication of how it may be used to help in the design process or to test your more extreme 'what if' ideas. There are of course limitations in relating simulation to actual circuits, but it is my experience that with a little thought in making allowance for 'parasitic' effects it is possible to achieve very close correspondence.

It is also my belief that some knowledge of the physical basis and origins of electronics is rather beneficial. The book is divided into five parts. First is a résumé of the general mathematical tools that may be useful in analysing systems. The treatment is on a fairly straightforward level with the emphasis on usability rather than any mathematical rigour – we assume that the mathematicians have sorted out all the difficulties. Second is an introduction to some of the physics underlying the many techniques used. Most electronics books simply state various laws, e.g. Kirchhoff's laws, without any indication as to their origin or validity. With electronics extending now into far-flung areas where applicability may be questioned, it is as well to have some grasp of the underlying physics. Third is a discussion of a number of circuit analysis techniques of general applicability. Fourth is a consideration of some of the most common circuit elements, in particular their deviations from the ideal in so far as this may affect the models that you may use for simulation. Fifth is the use of simulation as an aid to design. I use a particular flavour of SPICE, PSpice, but I hope that most of what is done will be applicable to all the other flavours. There are many, sometimes very large, texts on the format and use of SPICE which should be consulted to learn the techniques. It is slightly unfortunate that most of these date from the time when it was necessary for you to write out the appropriate netlist for the circuit but it is probably useful to know the general techniques and rules involved so you can understand the limitations and sort out some of the difficulties that can arise. Versions of SPICE are now screen based in that you need only draw the circuit schematic and the software will create the required netlist, which saves considerable time and avoids your entry errors. In this part I have chosen a range of circuits many of which have arisen in my own work (and which I hope means that I have had to think much more about and understand better) and which illustrate techniques that could be of use in more general circuits. It is the techniques rather than the applications that are important. Where appropriate I have sought to compare direct analysis, sometimes using Mathcad©, with SPICE results. The aim is also to encourage you to experiment in more unusual ways: modifications are quickly made, signals which in actual circuits may be difficult to measure are readily observed, and if you make a mistake and pass a current of 1000A you do not get a large puff of smoke! Some circuits can take a lot of simulation time so use a fast PC if you can. Nowadays the cost of a high-speed computer is insignificant compared with the time you will save.

Included with the book is a student, or demonstration, copy of the simulation software PSpice on CD-ROM. This is provided by arrangement with Cadence and I must acknowledge their generous assistance and collaboration in this matter. The software includes most of the full version but is limited as to the size of circuits that may be run and the libraries of models that are so essential. The circuits in the book which have been simulated are included on the CD and most, but not all, will run under the demo version of the software. Some additional libraries, made up for the purpose, are also included. The {circuit}.prb files, which determine the form of the simulation to be run and the output display, are also included to assist in the initial running of the circuits.

It will be evident from the book's contents that I do not subscribe to approaches that avoid the use of mathematics at almost any cost. Mathematics is the language of science and you place yourself at a considerable disadvantage if you cannot speak it competently. It provides the path to deeper understanding of how systems behave and, in particular, it allows you to make predictions. Design is in essence prediction since you are expecting the system to meet the requirements.

> Numbers count in every sense. If you know a thing by its quality, you know it only vaguely. If you know it by its quantity, you begin to know it deeply. You have access to power, and the understanding it provides. Being afraid of quantification is tantamount to disenfranchising yourself, giving up one of the most potent prospects for understanding and changing the world.
>
> Carl Sagan, physicist and astronomer

The application of mathematics should not put you off. Like everything else you will make many mistakes but practice is what is required and you can't get that if you never try.

Get it down. Take chances. It may be bad, but that's the only way you can do anything really good.

William Faulkner

Ever tried. Ever failed. No matter. Try again. Fail again. Fail better.

Samuel Becket

In the mathematical approaches, I have generally tried to give a fairly full account of the sums so that they may be more readily followed, and in many cases you can call on the power of SPICE to validate your conclusions. I have tried to relate the mathematics that has been included to the applications considered later but you should be aware that only a small, but significant, portion of the available techniques is included (a recent handbook runs to 2861 pages: Chen 1995).

I have sought to include a substantial number of references for all the topics referred to so that further information may be readily found. Some will be repetitive but this makes it more likely that you will be able to obtain access. The well-known semiconductor manufacturers provide many models for their products and these are generally accessible on websites if not included in your SPICE. The availability of good models is crucial to the process of simulation but it must be remembered that they are mostly functional rather than transistor level models and do not cover every aspect of the device. Some devices are too difficult to model satisfactorily, especially with acceptable simulation times, and some classes of device still appear to be unmodelled, but there is a great deal that can be achieved.

I hope of course that you will find at least something useful in these pages and that they may prompt you to further investigation. As to errors, I would be most grateful if you would bring these to my attention and I would be happy to discuss as far as I am able any matters that may be of mutual interest. My thanks to my present and past colleagues and to all the correspondents from whom I have received such willing help. In deference to market forces and to the entreaties of the publisher I have used analog rather than analogue both in the title and the text. My apologies to any readers affronted by this craven act.

Technical volumes are generally rather dour affairs with little recourse to levity. As the title of the present volume includes the term companion, as in bedside companion, I feel less constrained and have included a range of quotations, some directly relevant and others that I simply liked. The publisher protests that they may confuse the argument but I hope that they will somewhat lighten the approach.

During the writing of this book MicroSim were subsumed by Orcad and shortly afterwards both became part of Cadence. References should therefore be interpreted in the light of this and enquiries directed appropriately. May I acknowledge the considerable help provided by the above companies over the years and more particularly the assistance of Patrick Goss of MicroSim and Dennis Fitzpatrick of Cadence in dealing with my many queries and observations. The development of the PSpice simulations was primarily carried out using Version 8 of the software. To avoid possible additional errors, and to maintain close corespondence, it is this version that is provided on the CD. It should be noted that the latest issue is several versions ahead, which should be borne in mind if you migrate. The new versions are considerably enhanced but for the purposes of the present applications you are not at a disadvantage. The schematics from Version 8 must be 'imported' into the later versions with possibly some minor adjustments required.

Scott Hamilton
Department of Physics and Astronomy, University of Manchester
Manchester M13 9PL. 21st September 2000. Scott.Hamilton@man.ac.uk

List of symbols and abbreviations

a	prefix atto, $\times 10^{-18}$
A	ampere, unit of current
A	gain of amplifier
A_0	zero frequency gain of amplifier
ABM	analog behavioural model
B	susceptance
$\mathbf{B}$	magnetic flux
$B, \mathcal{B}$	bandwidth
C	capacitance
C	coulomb, unit of charge
c	speed of light, $2.998 \times 10^8\ \mathrm{m\,s^{-1}}$
CCCS	current-controlled current source
CCVS	current-controlled voltage source
C_{GD}	FET gate–drain capacity
C_{GS}	FET gate–source capacity
C_{iss}	FET common-source input capacity
C_{oss}	FET common-source output capacity
C_{rss}	FET common-source reverse transfer capacity
CMRR	common-mode rejection ratio
D	diode
dB	decibel
d.c.	direct current or zero frequency (z.f.)
E	prefix exa, $\times 10^{18}$
$\mathbf{E}$	electric field
F	farad, unit of capacity
f	frequency
f	prefix femto, $\times 10^{-15}$
f_c	corner frequency
f_T	transition frequency
G	prefix giga, $\times 10^9$
G	conductance
G	gain
G	FET gate
$G(0)$	gain at zero frequency
G_0	gain at zero frequency
G_{fs}	transconductance
G_m	transconductance
$G(s)$	gain at frequency s
$G(\infty)$	gain at infinite frequency
G_∞	gain at infinite frequency
$\mathbf{H}$	magnetic field
H	henry, unit of inductance
h	Planck constant, $6.63 \times 10^{-34}\ \mathrm{J\,s}$
$H(s)$	transfer function
Hz	hertz, unit of frequency
I_B	base current
I_C	collector current
I_{DSS}	zero-bias saturation current, gate tied to source
I_E	emitter current
I_{GSS}	gate leakage current, source tied to drain

J	joule, unit of energy
K	kelvin, unit of absolute temperature
k_B	Boltzmann constant, 1.38×10^{-23} J K^{-1}
k	prefix kilo, $\times 10^3$
k	coupling factor
L	Inductance
L	load
$\mathcal{L}$	Laplace transform operator
LHP	left half-plane (of complex plane)
M	magnetization
M	prefix mega, $\times 10^6$
m	prefix milli, $\times 10^{-3}$
m	metre, unit of length
m_e	electron rest mass, 9.11×10^{-31} kg
N	Poynting vector
N, n	turns
n	prefix nano, $\times 10^{-9}$
n	refractive index
N_A	Avogadro number, 6.02×10^{23} mol^{-1}
P	power
p	prefix pico, $\times 10^{-12}$
P	prefix peta, $\times 10^{15}$
Q	quality factor
Q	charge
Q	transistor
q_e	electronic charge, 1.602×10^{-19} C
R	resistor
$R_{DS(on)}$	FET on resistance
RHP	right half-plane (of complex plane)
S	source
S	FET source
S	signal
S	siemen, unit of conductance
s	second, unit of time
s	complex frequency
SRF	self-resonant frequency
T	time constant
T	time interval or delay
T	prefix tera, $\times 10^{12}$
T	degree absolute or kelvin
t_p	pulse width
t_r	risetime
t_{rr}	reverse recovery time
V	volt, unit of potential
V_B	base voltage
V_{BE}	base–emitter voltage
V_C	collector voltage
VCC	supply voltage
VCCS	voltage-controlled current source
V_{CE}	collector–emitter voltage
VCVS	voltage-controlled voltage source
V_D	drain voltage
V_{DS}	drain–source voltage
V_E	emitter voltage
V_G	gate voltage
V_{GS}	gate–source voltage
V_J	p-n junction voltage
V_{oc}	open circuit voltage
V_S	source voltage
V_{th}	FET threshold voltage
W	watt, unit of power
X	reactance

Y	admittance
Y	prefix yotta, $\times 10^{24}$
Z	impedance
Z	prefix zetta, $\times 10^{21}$
z.f.	zero frequency (d.c.)
α	coefficient of resistance
α_0	attenuation factor
β	beta, feedback factor
β	beta, transistor current gain
β_0	d.c. current gain
χ_B	magnetic susceptibility
γ	gamma
$\delta(t)$	delta function
δ	skin depth
∂	partial differentiation
ε	epsilon, permittivity
ε_0	permittivity of free space, $8.85 \times 10^{-12}\ \mathrm{F\,m^{-1}}$
γ	magnetogyric ratio
Γ	contour length
θ	theta, angle
λ	lambda, wavelength
μ	mu, relative permeability
μ_0	permeability of free space, $4\pi \times 10^{-7}\ \mathrm{H\,m^{-1}}$
$\mathrm{\mu}$	prefix micro, $\times 10^{-6}$
ν	nu, frequency
Φ	magnetic flux
ρ	rho, density, resistivity
σ	sigma, conductivity
τ	tau, time constant
ϕ	phi, angle
ψ	psi, angle
ω	omega, angular frequency
ω_c	corner angular frequency
ω_T	transition angular frequency
Ω	ohm, unit of resistance
$\bullet$	vector dot product
$\times$	vector cross product
$\propto$	proportional to
$>$	greater than
$\gg$	much greater than
$\geq$	greater than or equal to
$<$	less than
$\ll$	much less than
$\leq$	less than or equal to
$=$	equals
$\equiv$	identically equal to
$\cong$	approximately or very nearly equals
$\approx$	of the order of
*	multiplication in SPICE expressions
$*$	convolution symbol
/	division in SPICE expressions
$\langle\ \rangle$	average value of
$\|\ \|$	modulus or absolute value
$\parallel$	parallel
$\Leftrightarrow$	Fourier pair
∂	partial differential
δ	a small increment
∇	del
Δ	a small change or increment
exp	exponential
j	square root of -1
$\mathscr{Im}$	imaginary part of a complex number
$\mathscr{Re}$	real part of a complex number
Ln	logarithm to base e
Log	logarithm to base 10

Contents

Part 5 SPICE circuit applications

Part 1

Mathematical techniques

Philosophy is written in this grand book – I mean the universe – which stands continually open to our gaze, but it cannot be understood unless one first learns to comprehend the language and interpret the characters in which it is written. It is written in the language of mathematics, and its characters are triangles, circles, and other geometrical figures, without which it is humanly impossible to understand a single word of it; without these one is wandering about in a dark labyrinth.

Galileo Galilei (1564–1642)

As indicated in the preface, this book is substantially about design and hence prediction. The tools that allow us to extrapolate to create a new design are an understanding of the physical characteristics and limitations of components, mathematical techniques that allow us to determine the values of components and responses to input signals, and of course as much experience as one can get. The latter of course includes making as much use as possible of the experience of others either by personal contact or by consulting the literature.

This part covers much of the basic mathematics that is generally found useful in analysing electronic circuits. There is a fairly widely propagated view that you can get by without much mathematical knowledge but I evidently do not subscribe to this. Many do indeed do very well without recourse to mathematics but they could do so much better with some knowledge, and this book is, in part, an attempt to persuade them to make the effort. We do not present a course on these techniques as that would expand the book far beyond an acceptable size, but rather provide an indication and reminder of what we think is important and useful. Much of the reluctance in this direction is possibly caused by the unattractiveness of heavy numerical computation but this is nowadays generally unnecessary since we have the assistance of many mathematical computational packages and, in our case, the enormous power and convenience of electronic simulation software. With the spread of the ubiquitous PC it is now uncommon for an electronicist to be without access to one.

When carrying out algebraic analysis it is all too easy to make mistakes and great care must be taken when writing out equations. It is often of assistance to check your units to see that they are consistent as this can often be of great use in catching errors. You also need to be prepared to make approximations as the equations for even quite simple circuits become more complex than can be analysed. SPICE can be of considerable assistance in that you may use it to determine at least approximate values for parameters that then allow you to determine the relative magnitudes of terms and hence which may be neglected without serious error. You can then check your final result against SPICE which is able to carry out the analysis without significant approximation. The benefit of the algebraic analysis is that it makes the function of each component evident and provides parameterized design formulae.

Though some of the topics may at first sight seem unexpected, I hope that as you progress through later sections you will come to appreciate their relevance. Some are treated in terms of simply a reminder and some are delved into in a little more detail. As far as possible references to further sources of information are provided.

1.1 Trigonometry

The power of instruction is seldom of much efficacy except in those happy dispositions where it is almost superfluous.

Edward Gibbon

It may seem unexpected to find a section on trigonometry, but in electronics you cannot get away from sine waves. The standard definitions of sine, cosine and tangent in terms of the ratio of the sides of a right-angled triangle are shown in Fig. 1.1.1 and Eq. (1.1.1).

For angle θ and referring to the sides of the triangle as opposite (o), adjacent (a) and hypotenuse (h) we have:

$$\sin\theta=\frac{o}{h},\qquad \cos\theta=\frac{a}{h},\qquad \tan\theta=\frac{o}{a} \tag{1.1.1}$$

A common way to represent a sinusoidal wave is to rotate the phasor OA around the origin O at the appropriate rate ω (in radians per second) and take the projection of OA as a function of time as shown in Fig. 1.1.2.

The corresponding projection along the x-axis will produce a cosine wave. This allows us to see the values of the functions at particular points, e.g. at $\omega t=\pi/2$, π, $3\pi/2$ and 2π as well as the signs in the four quadrants (Q). These are summarized in Table 1.1.1.

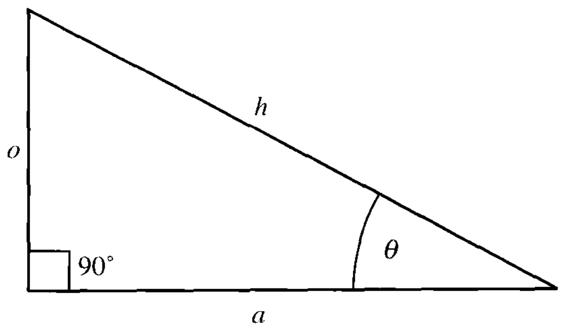

Fig. 1.1.1 Right-angled triangle.