PSpice for Circuit Theory and Electronic Devices

PSpice for Circuit Theory and Electronic Devices
Paul Tobin

ISBN: 978-3-031-79754-5 paperback
ISBN: 978-3-031-79755-2 ebook

DOI: 10.1007/978-3-031-79755-2

A Publication in the Springer series
SYNTHESIS LECTURES ON DIGITAL CIRCUITS AND SYSTEMS #7

Lecture #7
Series Editor: Mitchell A. Thornton, Southern Methodist University

Library of Congress Cataloging-in-Publication Data

Series ISSN: 1932-3166 print
Series ISSN: 1932-3174 electronic

First Edition
10 9 8 7 6 5 4 3 2 1

PSpice for Circuit Theory and Electronic Devices

Paul Tobin
School of Electronic and Communications Engineering
Dublin Institute of Technology
Ireland

SYNTHESIS LECTURES ON DIGITAL CIRCUITS AND SYSTEMS #7

ABSTRACT

PSpice for Circuit Theory and Electronic Devices is one of a series of five PSpice books and introduces the latest Cadence Orcad PSpice version 10.5 by simulating a range of DC and AC exercises. It is aimed primarily at those wishing to get up to speed with this version but will be of use to high school students, undergraduate students, and of course, lecturers. Circuit theorems are applied to a range of circuits and the calculations by hand after analysis are then compared to the simulated results. The Laplace transform and the s-plane are used to analyze CR and LR circuits where transient signals are involved. Here, the Probe output graphs demonstrate what a great learning tool PSpice is by providing the reader with a visual verification of any theoretical calculations. Series and parallel-tuned resonant circuits are investigated where the difficult concepts of dynamic impedance and selectivity are best understood by sweeping different circuit parameters through a range of values.

Obtaining semiconductor device characteristics as a laboratory exercise has fallen out of favour of late, but nevertheless, is still a useful exercise for understanding or modelling semiconductor devices. Inverting and non-inverting operational amplifiers characteristics such as gain-bandwidth are investigated and we will see the dependency of bandwidth on the gain using the performance analysis facility. Power amplifiers are examined where PSpice/Probe demonstrates very nicely the problems of cross-over distortion and other problems associated with power transistors. We examine power supplies and the problems of regulation, ground bounce, and power factor correction. Lastly, we look at MOSFET device characteristics and show how these devices are used to form basic CMOS logic gates such as NAND and NOR gates.

KEYWORDS

Cadence Orcad PSpice V10.5, Ohm's law, Kirchhoff's laws, Thévenin and Norton theorems, Mesh and nodal analysis, Laplace, transients, transfer functions, resonance, transformers, power supplies, ground bounce, operational amplifiers, power amplifiers.

I would like to dedicate this book to my wife and friend, Marie and sons Lee, Roy, Scott and Keith and my parents (Eddie and Roseanne), sisters, Sylvia, Madeleine, Jean, and brother, Ted.

Contents

Preface

Many years ago, I discovered how electronic simulation helped students come to grips with difficult engineering concepts. Earlier simulation software used cumbersome circuit netlists but nevertheless showed me how it helped students gain an intuitive circuit design sense. PSpice evolved along with the Windows environment to produce, in my opinion, a very powerful teaching and learning tool for accessing a whole range of difficult areas such as circuit theory, electronics, telecommunications and digital signal processing (DSP). This book, and my other fours books, grew from laboratory exercises and projects given to my student over the last twenty years.

An unfortunate trend in engineering education throughout the world has been to reduce analogue circuit design and circuit theory when considering new course syllabi. This is due, in part, to the ever-growing software-based technology such as the Open Systems Interconnection (OSI) networking model and associated protocols, C, C++, Java etc. Something has to go and unfortunately it seems to be some important basic principles. Students find digital circuits and DSP much easier to understand than analogue circuits and hence students tend to 'cherry pick' the easier topics ending up with a poorer overall understanding of engineering design. This is leaving the engineering recruitment market suffering from a lack of analogue design engineers. Good analogue circuit design is a combination of circuit analysis, an intuitive feel for electronic design and engineering problem solving obtained from experience. PSpice comes to the rescue with all these problems and helps students develop an intuitive design sense in a much shorter time.

This book is a combination of textbook and laboratory manual and contains worked examples with sufficient theory to enable the reader to compare simulation results to hand calculations. Exercises at the end of each chapter are partly worked to encourage the student to finish to completion. Lecturers should find the book as a valuable source for examination questions (loud groan from all), laboratory work, student projects and lecture material. It should also be very useful to second-level high school teachers where electronic technology has been introduced into the curriculum for some years. The book contains eight chapters covering topics from DC, AC and electronic devices. Chapter 1 introduces PSpice version 10.5 using a very simple DC circuit. Chapter 2 examines fundamental electric circuit principles and circuit theorems applied to DC and AC networks. In chapter 3, we look at the Laplace transform applied to first–order CR and LR switching circuits where the simulation outputs of currents and voltage at different times may be compared to hand calculations. Chapter 4 continues with more s-plane circuits and examines Butterworth and Chebychev transfer function. Chapters 5 and 6 analyses and simulates, AC circuits and applies circuit theorems such as Thévenin's theorem, mesh and nodal analysis to a range of circuits, including series and parallel resonant circuits. In Chapters 7 we plot electronic device characteristics in order to design circuits using measured device parameters from the characteristics. In the last chapter we examine operational and power amplifiers and a brief visit to CMOS devices and logic gates.

ACKNOWLEDGMENTS

I was introduced to circuit theory and electronics when I attended, many years ago, a very comprehensive series of lectures on these topics given by a fine lecturer and retired head of our department, Chris Cowley, so my thanks to him now many years later. I should also thank my students, past and present for inadvertently proof reading my books.