

PSpice for Circuit Theory and Electronic Devices

© Springer Nature Switzerland AG 2022

Reprint of original edition © Morgan & Claypool 2007

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means—electronic, mechanical, photocopy, recording, or any other except for brief quotations in printed reviews, without the prior permission of the publisher.

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	xiii
1. Introduction to PSpice and Ohm's Law	1
1.1 Laying out A Schematic	1
1.2 Libraries	2
1.2.1 Moving Components Around	4
1.2.2 Display Properties	5
1.3 The DC Circuit	6
1.3.1 New Simulation	7
1.3.2 Main Operational Icons	8
1.3.3 Simulation Settings	9
1.4 Potential Divider	9
1.4.1 Current Divider	11
1.5 Exercises	12
2. The DC Circuit and Kirchhoff's Laws	13
2.1 Maximum Power Transfer	13
2.1.1 Param Part	13
2.1.2 Simulation Settings	13
2.1.3 Trace Expression Box	15
2.2 Changing the X-Axis Variable	16
2.2.1 The Log Command	17
2.3 Mesh Analysis	17
2.4 Nodal Analysis	20
2.5 Exercises	22
3. Transient Circuits and Laplace Transforms	23
3.1 Transient Analysis	23
3.2 Laplace Transform and Capacitance	23
3.3 Inductance	24
3.4 First-Order CR and LR Circuits	24
3.4.1 Solution	25
3.4.2 Partial Fraction Expansion	26

3.4.3	Initial Conditions	27
3.5	Example 2	28
3.5.1	Solution	28
3.6	Example 3	29
3.6.1	Solution	30
3.7	Example 4	32
3.7.1	Solution	33
3.8	Exercise	34
4.	Transfer Functions and System Parameters	35
4.1	Transfer Functions	35
4.2	Butterworth Transfer Functions and the Laplace Part	35
4.2.1	Piece-wise Linear Part (VPWL)	37
4.3	Probe Grid and Cursors Icons	37
4.4	Elaplace Part and the Step Response	38
4.5	Chebyshev Transfer Functions Impulse Response	40
4.5.1	Unsynchronizing Probe Plots	42
4.6	First-Order Low-Pass Filter Step and Impulse Responses	42
4.7	Obtaining the Frequency Response from the Impulse Response	44
4.8	The Low-Pass CR Filter Step Response	45
4.9	Rise Time	46
4.10	Step Response of a Series-Tuned LCR Circuit	48
4.10.1	Overshoot	48
4.11	Exercises	48
5.	AC Circuits and Circuit Theorems	53
5.1	AC Circuit Theory	53
5.2	Capacitors	53
5.2.1	Capacitive Reactance Plot	54
5.2.2	Capacitor Current and Voltage Waveforms	55
5.3	Inductors	56
5.3.1	Inductor Signal Phase Measurement	58
5.4	AC Circuit Theorems	58
5.4.1	Thévenin's Theorem	59
5.4.2	Thévenin Impedance	60
5.4.3	Thévenin Voltage	61
5.5	Norton Equivalent Circuit	61
5.5.1	The Output File	62

5.6	AC Mesh and Nodal Analysis	62
5.7	Exercises	62
6.	Series and Parallel-tuned Resonance.....	65
6.1	Resonance	65
6.2	Series-Tuned Circuit	65
6.2.1	Current Response.....	66
6.3	Example	67
6.3.1	Solution	67
6.4	Q-Factor.....	67
6.4.1	The -3 dB Bandwidth	67
6.5	Voltages Across L and C at Resonance	68
6.6	Universal Response Curve.....	70
6.7	Selectivity of a Series-Tuned Resonant Circuit	70
6.7.1	L/C Ratio and Selectivity	72
6.8	Series-Tuned LCR Phase Response.....	72
6.9	Impedance of a Series-Tuned Circuit.....	73
6.10	Fourier Series.....	74
6.10.1	Series-Tuned Circuit as a Low-Pass Filter	74
6.10.2	The Output File.....	75
6.11	Skip Initial Conditions	76
6.12	Parallel-Tuned LCR Circuit	77
6.12.1	Universal Response Curve	80
6.12.2	Relationship Between the Resonant Frequency and Bandwidth.....	81
6.12.3	Loaded Q -factor.....	81
6.13	Example.....	82
6.13.1	Solution	82
6.14	Problem.....	82
6.15	Frequency Response of a Parallel-Tuned Circuit	83
6.16	Dynamic Impedance.....	84
6.16.1	Loaded and Unloaded Q -factor	84
6.17	Transformers	88
6.17.1	Transformer Parameters	89
6.17.2	Matching Transformer	89
6.18	Power Supplies: Rectification and Regulation	90
6.18.1	Power Supply Waveforms	91
6.18.2	Power Supply Voltage Regulation	91

6.19	Ground Bounce	91
6.20	Power Factor Correction	94
6.20.1	Average Power and Apparent Power	96
6.21	Exercises	97
7.	Semiconductor Devices and Characteristics	101
7.1	Semiconductor Devices	101
7.2	The Forward and Reverse-Biased Diode Characteristic	101
7.3	Diode Parameters	102
7.4	DC Load Line	103
7.5	Voltage Regulation	105
7.5.1	Zener Diode Characteristic	106
7.5.2	Zener Diode Regulation	107
7.6	Silicon-Controlled Rectifier	107
7.7	Triac Controller	109
7.8	The Bipolar Transistor	109
7.8.1	The Input and Output BJT Characteristics	110
7.8.2	The Output Characteristic	112
7.8.3	DC Load Lines	113
7.9	Junction Field-Effect Transistor	114
7.9.1	The Common Source JFET Transistor Input Characteristic	114
7.9.2	Adding a Load Line to the Transfer Characteristic	115
7.9.3	Quiescent DC Operating Point	117
7.9.4	JFET Output Characteristic	117
7.9.5	Effect of Temperature on the JFET Transfer Characteristic	120
7.10	The D Operator	122
7.11	Exercises	122
8.	Operational Amplifier Characteristics	127
8.1	Ideal Operational Amplifiers	127
8.1.1	The Inverting Operational Amplifier and Virtual Earth	127
8.1.2	Slew Rate Limiting	130
8.2	The Noninverting Operational Amplifier	130
8.2.1	Gain-Bandwidth Product	131
8.3	Audio Power Amplifiers	133
8.3.1	The Output File	136
8.4	Mosfet Device Characteristic	138
8.4.1	CMOS Model	139

8.4.2	Nand Gate	142
8.4.3	Nor Gate	143
8.5	Exercises	144
	References	152
	Appendix A: Laplace and z -Transform Table	154
	Index	155
	Author Biography	159

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 four 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.