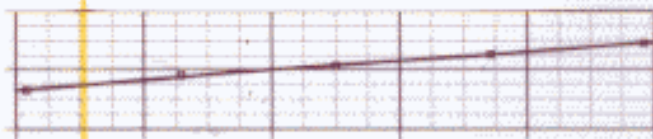
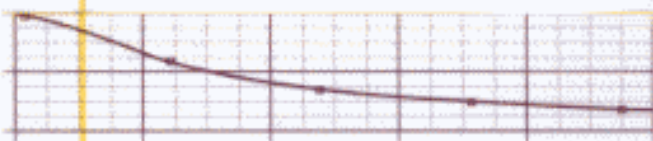


Introduction to **PSpice[®]** Using **OrCAD[®]** for Circuits and Electronics

WITH
CD-ROM
INSIDE

Third Edition



Muhammad H. Rashid



CD-ROM
Included

Contents

Preface	xiii
About the Author	xvii
CHAPTER 1 Introduction	1
1.1 Introduction	1
1.2 Descriptions of Spice	2
1.3 Types of Spice	2
1.4 Types of Analysis	3
1.5 Descriptions of Simulation Software Tools	5
1.6 PSpice Platform	6
1.6.1 PSpice A/D	6
1.6.2 PSpice Schematics	6
1.6.3 OrCAD Capture	7
1.7 PSpice Schematic Versus OrCAD Capture	7
1.8 Limitations of PSpice	9
1.9 SPICE Resources	9
References	12
CHAPTER 2 Circuit Descriptions	14
2.1 Introduction	14
2.2 Input Files	14
2.3 Element Values	16
2.4 Nodes	17
2.5 Circuit Elements	18
2.6 Sources	20
2.7 Types of Analysis	22
2.8 Output Variables	22
2.9 PSpice Output Commands	23
2.10 Format of Circuit Files	23
2.11 Format of Output Files	25
2.12 Examples of SPICE Simulations	26

2.13	OrCAD Capture	32
2.13.1	OrCAD Capture Layout	32
2.13.2	PSpice A/D	35
2.13.3	Probe	37
2.14	Importing Microsim Schematic in OrCAD Capture	37
	References	37
	Problems	38

CHAPTER 3 DC Circuit Analysis **39**

3.1	Introduction	39
3.2	Resistors	40
3.3	Modeling of Elements	41
3.4	Operating Temperature	43
3.5	Independent DC Sources	44
3.5.1	Independent DC Voltage Source	44
3.5.2	Independent DC Current Source	45
3.5.3	Schematic Independent Sources	45
3.6	Dependent Sources	45
3.6.1	Polynomial Source	45
3.6.2	Voltage-Controlled Voltage Source	48
3.6.3	Voltage-Controlled Current Source	49
3.6.4	Current-Controlled Current Source	50
3.6.5	Current-Controlled Voltage Source	51
3.6.6	Schematic Dependent Sources	52
3.7	DC Output Variables	52
3.7.1	Voltage Output	52
3.7.2	Current Output	54
3.8	Types of Output	56
3.8.1	.PRINT (Print Statement)	56
3.8.2	.PLOT (Plot Statement)	56
3.8.3	.PROBE (Probe Statement)	57
3.8.4	Probe Output	57
3.8.5	.WIDTH (Width Statement)	60
3.9	Types of DC Analysis	61
3.9.1	.OP (Operating Point)	61
3.9.2	.TF (Small-Signal Transfer Function)	64
3.9.3	.DC (DC Sweep)	68
3.9.4	.PARAM (DC Parametric sweep)	75
	Summary	79
	References	80
	Problems	80

CHAPTER 4	Transient Analysis	84
4.1	Introduction	84
4.2	Capacitors and Inductors	85
4.2.1	Capacitor	85
4.2.2	Inductor	87
4.3	Modeling of Transient Sources	90
4.3.1	Exponential Source	90
4.3.2	Pulse Source	91
4.3.3	Piecewise Linear Source	93
4.3.4	Single-Frequency Frequency Modulation	93
4.3.5	Sinusoidal Source	95
4.4	Transient Sources	96
4.4.1	Independent Voltage Source	97
4.4.2	Independent Current Source	97
4.5	Transient Output Variables	98
4.6	Transient Output Commands	99
4.7	Transient Response	99
4.7.1	.IC (Initial Transient Conditions)	100
4.7.2	.TRAN (Transient Analysis)	101
4.8	Switches	112
4.8.1	Voltage-Controlled Switch	112
4.8.2	Current-Controlled Switch	117
4.8.3	Time-Dependent Switches	119
	Summary	122
	References	124
	Problems	124
CHAPTER 5	AC Circuit Analysis	131
5.1	Introduction	131
5.2	AC Output Variables	132
5.2.1	Voltage Output	132
5.2.2	Current Output	133
5.3	Independent AC Sources	134
5.4	AC Analysis	135
5.5	Magnetic Elements	145
5.6	Transmission Lines	149
5.7	Multiple Analyses	152
	Summary	154
	References	154
	Problems	155

CHAPTER 6	Advanced SPICE Commands and Analysis	159
6.1	Introduction	159
6.2	Behavioral Modeling	160
6.2.1	Value	160
6.2.2	Table	162
6.2.3	Laplace	163
6.2.4	FREQ	164
6.3	.SUBCKT (Subcircuit)	164
6.4	.ENDS (End of Subcircuit)	165
6.5	.FUNC (Function)	166
6.6	.GLOBAL (Global)	167
6.7	.INC (Include File)	167
6.8	.LIB (Library File)	168
6.9	.NODESET (Nodeset)	169
6.10	Options	169
6.11	.PARAM (Parameter)	172
6.12	Fourier Analysis	174
6.13	Noise Analysis	177
6.14	.SENS (Sensitivity Analysis)	180
6.15	.STEP (Parametric Analysis)	182
6.16	.DC (DC Parametric Sweep)	186
6.17	Monte Carlo Analysis	189
6.18	DEV/LOT Device and Lot Tolerances	191
6.19	Sensitivity/Worst-Case Analysis	195
	Summary	197
	References	198
	Problems	199
CHAPTER 7	Semiconductor Diodes	203
7.1	Introduction	203
7.2	Diode Characteristics	203
7.2.1	Forward-Bias Region	205
7.2.2	Reverse-Bias Region	205
7.2.3	Breakdown Region	206
7.3	Analysis of Diode Circuits	206
7.3.1	DC Analysis	206
7.3.2	Small-Signal AC Analysis	208
7.4	Diode Model	209
7.5	Diode Statement	211
7.6	Diode Parameters	212
7.7	Examples of DC Analysis	215
7.8	Examples of Transient and AC Analysis	223

	Summary	232
	References	232
	Problems	233
CHAPTER 8	Bipolar Junction Transistors	237
	8.1 Introduction	237
	8.2 BJT Model	237
	8.3 BJT Statements	240
	8.4 BJT Parameters	242
	8.5 Examples of BJT Circuits	248
	Summary	283
	References	283
	Problems	284
CHAPTER 9	Field-Effect Transistors	289
	9.1 Introduction	289
	9.2 Junction Field-Effect Transistors	289
	9.3 JFET Parameters	293
	9.4 Examples of JFET Amplifiers	295
	9.5 Metal Oxide Silicon Field-Effect Transistors	307
	9.6 MOSFET Parameters	312
	9.7 Examples of MOSFET Amplifiers	318
	9.8 Gallium Arsenide MESFETs	329
	Summary	334
	References	334
	Problems	335
CHAPTER 10	Op-Amp Circuits	340
	10.1 Introduction	340
	10.2 DC Linear Models	340
	10.3 AC Linear Model	341
	10.4 Nonlinear Macromodel	342
	10.5 Examples of OP-AMP Circuits	344
	References	367
	Problems	368
CHAPTER 11	Difficulties	373
	11.1 Introduction	373
	11.2 Large Circuits	374
	11.3 Running Multiple Circuits	374
	11.4 Large Outputs	375
	11.5 Long Transient Runs	375

11.6	Convergence Problems	376
11.6.1	DC Sweep	376
11.6.2	Bias Point	378
11.6.3	Transient Analysis	379
11.7	Analysis Accuracy	380
11.8	Negative Component Values	381
11.9	Power-Switching Circuits	382
11.10	Floating Nodes	385
11.11	Nodes with Less than two Connections	389
11.12	Voltage Source and Inductor Loops	389
11.13	Running A PSpice File on Spice	390
11.14	Running a Spice File on PSpice	390
11.15	Running an Earlier Version of Schematic	391
	References	391
	Problems	392
Appendix A	Drawing in PSpice Schematics	393
A.1	Learning Objectives	393
A.2	Installing the Software	393
A.3	Overview	393
A.4	The Circuit Analysis Process	395
A.5	Drawing the Circuit	395
A.6	Copying and Capturing Schematics	405
Appendix B	Drawing in OrCAD Capture Lite	406
B.1	Learning Objectives	406
B.2	Installing the Software	406
B.3	Overview	408
B.4	The Circuit Analysis Process	409
B.5	Drawing the Circuit	410
B.6	Copying and Capturing Schematics	422
Appendix C	Creating Input Circuit File	424
Appendix D	DOS Commands	427
Appendix E	Noise Analysis	429
E.1	Thermal Noise	429
E.2	Shot Noise	429
E.3	Flicker Noise	430
E.4	Burst Noise	430
E.5	Avalanche Noise	431

E.6	Noise in Diodes	431
E.7	Noise in Bipolar Transistors	432
E.8	Noise in Field-Effect Transistors	432
E.9	Equivalent Input Noise	433

Appendix F	Nonlinear Magnetic Model	437
-------------------	---------------------------------	------------

Appendix G	PSpice Default Symbol Libraries	440
-------------------	--	------------

Bibliography		442
---------------------	--	------------

Index		444
--------------	--	------------