

Contents

Preface10

Chapter 1 • Introduction11

 1.1 Why simulate?11

 1.2 Electronic Simulation13

 1.3 The LTspice Simulator13

 1.3.1 Schematic capture13

 1.3.2 DC analysis14

 1.3.3 AC analysis14

 1.3.4 Transient analysis14

 1.3.5 Plotting the results14

 1.3.6 Transient noise analysis14

 1.3.7 Fourier analysis (FFT)14

 1.3.8 Digital simulation14

 1.3.9 Laplace transform based analysis14

 1.3.10 Expression editor15

 1.3.11 Parameter sweeping15

 1.3.12 AC sweep15

 1.3.13 DC sweep15

 1.3.14 SPICE models15

 1.3.15 Netlist15

 1.3.16 Online help15

Chapter 2 • Getting Started with LTspice16

 2.1 Overview16

 2.2 Installing LTspice on Windows16

 2.3 Running LTspice16

 2.4 Creating a New Schematic17

 2.5 Simulator Settings26

 2.6 Example Schematics26

 2.7 Online Help26

Chapter 3 • Basic DC Circuit Analysis and Simulation	27
3.1 Overview	27
3.2 Simple 4-Resistor Circuit	27
3.3 A More Complex Resistor Circuit	30
3.4 Resistive Potential Divider–Sweep a DC Voltage	33
3.5 Resistive Potential Divider–Sweep a Resistor Value	35
3.6 Power Dissipation in a DC Circuit.	37
3.7 Resistor-Capacitor (RC) Transient Circuit	39
3.8 Resistor-Inductor-Capacitor (RLC) Transient Circuit	41
3.9 Input Resistance, Output Resistance, and Transfer Function	46
Chapter 4 • Basic AC Circuit Analysis and Simulation	49
4.1 Overview	49
4.2 Resistor – Capacitor Circuit.	49
4.3 Resistor – Inductor Circuit	50
4.4 Resistor – Inductor – Capacitor Circuit	51
4.5 Thevenin’s Theorem–AC Circuit Analysis	54
4.6 Three-Phase Circuits	56
4.6.1 Star with resistive loads	57
4.6.2 Star with resistive and inductive loads.	59
4.7 Mutual Inductance	60
4.7.1 Circuit with mutual inductance	61
4.8 Using a Transformer.	63
4.9 Frequency Sweeping	63
4.10 Fast Fourier Transform (FFT).	65
4.11 Input & Output Impedance, and Gain.	66
Chapter 5 • Diode Circuits	74
5.1 Overview	74
5.2 Half-Wave Rectifier	74
5.3 Full-Wave Rectifier.	76
5.4 Full-Wave Rectifier with a Transformer	78
5.5 Zener Diode Voltage Regulator	79

5.6 Diode Clamp81

5.7 Voltage Multiplier.82

5.8 I-V Characteristic of a Diode85

Chapter 6 • Transistor Analysis, Design and Simulation87

6.1 Overview87

6.2 Bipolar Junction Transistor (BJT) I-V Curve87

6.3 Common-Emitter Transistor Amplifier–Analysis.88

6.4 Common-Emitter Transistor Amplifier–Design.91

6.5 Audio Power Amplifiers.93

6.5.1 Class A audio power amplifier simulation94

6.5.2 Class AB audio power amplifier.96

6.6 BJT Switch98

6.7 JFET Common-Source Amplifier.100

6.8 MOSFET I-V Characteristic Curve.102

Chapter 7 • Thyristors, Diacs and Triacs–Importing a SPICE model.105

7.1 Overview105

7.2 Thyristors, diacs, and triacs105

7.2.1 Triac–diac phase control106

7.3 Inserting a Potentiometer–Using a Potentiometer Model109

Chapter 8 • Oscillators113

8.1 Overview113

8.2 Transistor Sinewave Oscillators113

8.2.1 Colpitts oscillator113

8.2.2 Hartley oscillator.115

8.3 Sine Wave Oscillators with Operational Amplifiers116

8.3.1 Phase-shift oscillator117

8.3.2 Wien bridge oscillator120

8.3.3 Colpitts oscillator122

8.5.4 Hartley oscillator.124

8.4 Op-Amp-Based Square Wave Oscillator126

8.5 The 555 Timer.127

8.5.1 The 555 astable multivibrator.	127
8.5.2 Blinking an LED	128
8.5.3 Metronome	130
8.5.4 Parametric sweep	131
Chapter 9 • Simulating Other Analog Circuits	134
9.1 Overview	134
9.2 The LT1086 Voltage Regulator.	134
9.3 10 V Output	135
9.4 DC Boost Converter	135
9.5 Optocoupler Simulation	137
9.6 Operational Amplifier with Single Supply	138
9.7 Sweeping a Resistor Value	139
Chapter 10 • Filters – Hierarchical Blocks	141
10.1 Overview	141
10.2 Low-Pass Filter	141
10.3 Filter Design Tool.	142
10.3.1 Low-pass active filters	142
10.4 Hierarchical Blocks – 2 nd -Order Low-Pass Filter Block	146
10.4.1 Using the filter block	147
10.5 Passing Parameters to Hierarchical Blocks.	149
10.5.1 Using the programmable block	149
10.6 Create Your Own Symbol	151
10.7 LTspice Filter Library.	152
Chapter 11 • Digital Logic	154
11.1 Overview	154
11.2 Digital Components	154
11.3 Half Adder Using Logic Gates	154
11.4 Full Adder from Two Half Adders	156
11.5 Importing the 74HC Family	158
11.6 Two-Input Multiplexer.	161
11.7 4-Bit Binary Counter	162

11.8 More Inputs	164
11.9 Full Adder	165
11.10 Monostable Multivibrator.	166
11.11 Square Wave Oscillator.	168
Chapter 12 • SPICE models	170
12.1 Overview	170
12.2 The Structure of SPICE Models	170
12.3 Passive Components	170
12.4 Active Components	172
12.4.1 Diodes.	172
12.4.2 Bipolar transistors.	174
12.4.3 MOSFETs	175
12.4.4 Creating a new diode model	176
12.5 Logic Gates	178
12.6 SPICE Directives	178
Chapter 13 • Some Useful Tools and Functions	180
13.1 Overview	180
13.2 Generating Triangular and Sawtooth Waveforms	180
13.3 Generating a Periodic Arbitrary Waveform.	181
13.4 The Voltage-Controlled Switch.	184
13.5 Laplace-Transform Based Simulation	185
Appendix A – Third-Party SPICE Models	189
Appendix B – Useful Resources	191
Index	193