



www.ceda.in

Contents

Lesson 1: Building a Design for Simulation	10
Learning Objectives	10
Creating a New Project	10
Project Templates.....	12
New projects may be based on an existing design or a template with Capture	12
The PSpice Project.....	13
Available Libraries.....	13
What Is a "Part" in EDA?.....	14
Placing a Part.....	15
Placing Multiple Instances of Parts.....	17
Rotating and Mirroring Parts	18
Connecting Parts Electrically.....	18
Valid Connections.....	19
Placing Wires	20
Rubber banding	21
Placing Symbols.....	22
Editing Properties.....	23
Creating Custom Filters	23
Assigning a Net Alias	24
Lab 1-1: Creating a RC Circuit.....	25
Lab Objectives	25
Creating a New PSpice Project.....	25
Placing the Voltage Source.....	27
Placing the Resistors	27
Placing the Capacitor	28
Connecting the Components	28
Placing the Analog Ground Symbol	28
Assigning the Capacitor Reference Designator.....	29
Labeling Nets	29
Assigning a DC Value of 10V to the Voltage Source	29
Saving the File.....	29
Lab 1-2: Property Spreadsheet Editor.....	29
Lab Objectives	29
Editing Properties for Multiple Parts	30
Additional Lab: Creating a Custom Filter	30
Lesson 2: DC Bias Point Analysis	31
Lesson Objectives	31
Setting Up a Simulation Profile	32
The Simulation Settings Dialog Box	32
Running a Bias Point Simulation.....	35
Examine the Output File and Bias Display.....	35
Value Multipliers	37

Lab 2-1: Simulating a DC Bias Point	38
Lab Objectives	38
Creating a Bias Point Simulation Profile	38
Running the Simulation	39
Viewing Simulation Output	39
Additional Lab Exercise.....	40
Lesson 3: DC Sweep Analysis	41
Learning Objectives	41
What is DC Sweep Analysis?	41
DC Sweep Settings and Options	41
Data Collection.....	42
Examining Results in Probe.....	44
Setting up Sources.....	44
Markers	46
Customizing the Probe Display.....	46
Multiple Y-axes.....	47
Lab 3-1: Performing a DC Sweep.....	48
Lab Objectives	48
Setting Up the Simulation	48
Setting Up Data Collection and Probe Windows.....	49
Analyzing the Results in the Probe Window	49
Plotting Traces From the Menu	50
Plotting Traces Using Markers	51
Customizing the Probe Display.....	52
Changing Axis and Grid Settings	53
Additional Lab Exercise.....	55
Lesson 4: AC Sweep Analysis	55
Lesson Objectives	55
What Does AC Sweep Analysis Do?.....	55
AC Sweep Simulation Setup.....	55
Configuring Sources	56
Setting up the AC Sweep Simulation Profile.....	57
Viewing Simulation Results in Probe	59
Trace Names.....	61
Functions	61
Lab 4-1: Performing an AC Sweep Analysis.....	62
Lab Objectives	62
Creating a New PSpice Project.....	62
Setting Up and Starting an AC Sweep Analysis.....	63
Running the Simulation and Viewing Results	64
Displaying a Bode Plot	65
Lesson 5: Simulating a Text Netlist.....	67
Learning Objectives	67
Running a Simulation From A Circuit File	67

PSPice Circuit File Format.....	67
Interpretation of Device Types and Pin Order.....	69
Device Instantiations.....	70
Comments in a Circuit File.....	71
Model Statements.....	72
Subcircuit Statements.....	72
Lab 5-1: Creating a Text Netlist.....	73
Lab Objectives.....	73
Creating a PSPice Netlist.....	73
Running PSPice Using the Netlist.....	74
Lesson 6: Stimulus Editor.....	74
Learning Objectives.....	74
Stimulus Editor.....	75
Vstim, Istim, Digstim Symbols.....	76
Defining a Stimulus.....	76
Configuring Stimulus Files.....	79
Where have all the Files Gone?.....	80
Review of Adding a Stimulus.....	80
Lab 6-1: Creating a Sine Wave Stimulus.....	81
Lab Objectives.....	81
Placing the VSTIM Symbol.....	81
Running the Simulation.....	82
Lab 6-2: Creating a Piecewise Linear Stimulus.....	83
Lab Objectives.....	83
Creating a Piece-wise Linear Source.....	84
Correcting Mistakes.....	86
Additional Exercises.....	86
Lesson 7: Transient Analysis.....	86
Learning Objectives.....	86
What is Transient Analysis?.....	87
Setting up a Simulation.....	87
Transient Sources.....	89
PSPice Runtime Settings.....	89
Scheduling Option Values.....	90
Lab 7-1: Transient Analysis on RC.DSN.....	91
Lab Objectives.....	91
Lab 7-2: Transient on BUFFER.DSN.....	94
Lab Objectives.....	94
Editing the Design File.....	94
Verifying Simulation Set Up.....	94
Plotting the Y Axis.....	95
Lab 7-3: Extending a Simulation.....	97
Lab Objectives.....	97
Re-running the Buffer Transient Simulation.....	97

Additional Exercise	97
Lesson 8: Resolving Simulation Errors	97
Learning Objectives	98
Common Circuit Errors and Solutions.....	98
Multiple Bias Points.....	99
Simulation Accuracy.....	99
Non-convergence	100
Newton-Raphson.....	100
Error Tolerances/Convergence Criteria	100
Common Causes of Convergence Failure.....	101
Troubleshooting DC and Bias Point Convergence Failure.....	101
Troubleshooting Transient Convergence Failure	102
Scheduling Option Values	103
Lab 8-1: Floating Node	104
Lab Objectives	104
Opening the nodcpath Project.....	104
Identifying Errors.....	105
Correcting Errors.....	105
Lab 8-2: Resolution.....	105
Lab Objectives	105
Opening the stpcplg Project.....	105
Setting Up and Running Simulation	106
Correcting Errors.....	106
Lab 8-3: Scheduling Runtime Settings	106
Lab Objectives	106
Opening the stpcplg Circuit	106
Lab 8-4: Convergence Error	107
Lab Objectives	107
Open the Project and Run the Simulation.....	107
Correcting Simulation Errors	109
Lab 8-5: Discontinuities.....	110
Lab Objectives	110
Lab 8-6: No Limit	111
Lab Objectives	111
Lesson 9: Transformers.....	112
Learning Objectives	113
Linear Transformers.....	113
Linear Transformer Circuit	114
Non-linear Transformers.....	114
Coupling Inductors.....	115
Lab 9-1: Linear Transformer.....	116
Lab Objectives	116
Constructing the Circuit	117
Configuring the Coupling Symbol.....	117

Configuring the Simulation.....	118
Lab 9-2: Non-linear Transformers	119
Lab Objectives	119
Create a Non-linear Transformer	119
Additional Exercise	120
Lesson 10: Parametric Analysis.....	120
Learning Objectives	120
What Does Parametric Analysis Do?.....	120
Preparing a Design for Parametric Analysis	121
Parametric Setup	122
Viewing Parametric Results in Probe	123
Using Cursors.....	124
Lab 10-1: Parametric Analysis.....	124
Lab Objectives	124
Create and Define Cval Part.....	125
Using a Global Parameter as a Property Value.....	126
Viewing the Output in the Probe Window.....	127
Additional Exercise	129
Lesson 11: Editing a Model	130
Learning Objectives	130
Creating a Model with the Model Editor	131
Text Mode	131
Graphical Mode.....	132
Local and Global Model Libraries	133
Referencing a Model	134
Library Indexing.....	135
Extracted Parameters.....	136
Lab 11-1: Editing a Model	136
Lab Objectives	136
Creating Project Zener.opj	136
Running the Simulation	138
Lab 11-2: Extracting a Model	139
Lab Objectives	139
Creating a New Library and Model	139
Entering Specification Data	140
Extracting PSpice Model Parameters.....	146
Getting a Symbol.....	148
Configuring New Parts and Models.....	149
Additional Exercise	150
Lesson 12: Adding New Parts.....	150
Learning Objectives	150
Types of Simulation Models	150
Creating Subcircuits from Schematics.....	151
Creating the Subcircuit.....	151

Differences Between Models and Subcircuits	151
Creating Symbols for Simulation Models	152
Creating Symbol Libraries	152
Creating a New Part in a Library	154
Part and Symbol Editor Palette	155
PSpice Template.....	155
Configuring Symbol and Model Libraries.....	156
Lab 12-1: Subcircuit from the Buffer Circuit	157
Lab Objectives	157
Create Buffer2 Project	157
Rename Schematic Folder to Create Subcircuit	158
Create Netlist.....	158
Lab 12-2: Creating a Part with the Wizard	159
Lab Objectives	159
Create Library Part.....	159
View the Part.....	160
Using the New Symbol	160
Lab 12-3: Creating a Part from Scratch	162
Lab Objectives	162
Creating a New Library File	162
Drawing the Graphics	164
Editing the Properties.....	166
Using the Part in a Design.....	166
Lesson 13: Temperature Analysis.....	166
Learning Objectives	166
Temperature Coefficients.....	167
Temperature Attributes	167
Extracting Temperature Models.....	169
Resistors and Temperature.....	170
Temperature in Simulation.....	170
Lab 13-1: Temperature Sweep.....	171
Lab Objectives	171
Create Thermostat Circuit.....	171
Configure Simulation Profile.....	171
Additional Exercise.....	172
Lesson 14: Monte Carlo Analysis.....	173
Learning Objectives	173
What Does Monte Carlo Analysis Do?.....	173
Device and Lot Tolerances	173
Configuring Monte Carlo Analysis.....	174
Options Within the Simulation Settings Dialog Box.....	175
Viewing Output File, Probe Plot, and Histograms	176
Lab 14-1: Performing a Monte Carlo Analysis	176
Lab Objectives	176

Creating the Pressure Project	177
Setting Up the Analysis.....	178
Examining the Results	180
Lab 14-2: Viewing Monte Carlo Histograms	182
Lab Objectives	182
Preparing the Circuit	183
Setting up the Analysis	183
Creating the Histogram	183
Changing the Number of Histogram Divisions	184
Viewing the Center Frequency Distribution of the Filter	186
Additional Exercise	186
Lesson 15: Hierarchical Blocks and Symbols	187
Learning Objectives	187
Pros & Cons of Hierarchical Blocks	187
Pros & Cons of Hierarchical Symbols	188
Connectors and Ports	188
Creating Hierarchical Blocks	189
Navigating the Hierarchy	190
Hierarchical Netlist	191
Lab 15-1: Hierarchical Block.....	192
Lab Objectives	192
Creating the Top Level Design	192
Creating the Lower Level Design	195
Simulating and Viewing Results.....	196
Lab 15-2: Hierarchical Symbols	197
Lab Objectives	197
Preparing a Design to be Symbolized.....	197
Creating a Hierarchical Symbol.....	198
Setting up the Project to Use Hierarchy.....	200
Using the New Symbol in a Design.....	202
Lab 15-3: Creating a Hierarchical Netlist.....	204
Lab Objectives	204
Open Hier_clip Design.....	204
Creating a Hierarchical Netlist.....	207
Lesson 16: Analog Behavioral Modeling	208
Learning Objectives	208
Analog Behavioral Modeling Defined.....	208
Using ABM Devices	209
Using ABM Devices by Reference	210
Lab 16-1: ABM Evaluate with Wired Inputs.....	210
Lab Objectives	210
Define Symbol Properties	211
Lab 16-2: ABM Evaluate by Reference	212
Lab Objectives	212

Editing the Design.....	212
Lab 16-3: Frequency Table	213
Lab Objectives	213
Placing and Defining Symbols.....	213
Lab 16-4: Voltage-Controlled Oscillator	215
Lab Objectives	215
Entering a Circuit into a New Project	215
Lesson 17: Simulating Digital Circuits.....	216
Learning Objectives	216
How Digital Components are Modeled	217
Placing Digital Parts.....	217
Wiring Digital Parts	218
Digital Power and Ground	218
Setting up Digital Simulations	220
Viewing Digital Results	221
Lab 17-1: Counter Example.....	223
Lab Objectives	223
Creating the Circuit.....	223
Creating the Simulation Profile	223
Running the Simulation	224
Viewing the Results in the Probe Window	224
Lab 17-2: Counter with Bus.....	225
Lab Objectives	225
Placing the Bus.....	226
Connecting the Bus	226
Running the Simulation	226
Lesson 18: Mixed Analog and Digital	227
Learning Objectives	227
Mixed Simulation.....	227
Lab 18-1: Mixed A/D Oscillator.....	228
Lab Objectives	228
Opening the A/D Circuit.....	228
Additional Exercise.....	229
Lesson 19: Measurements	229
Learning Objectives	229
How Measurements Work	229
Measurement Format	230
Measurement Example.....	232
Plot Window Templates.....	234
Lab 19-1: Using the Risetime Measurement	234
Lab Objectives	234
Creating the Goals Project	235
Configuring a Transient Analysis	235
Running the Simulation	235

Configuring and Running the Performance Analysis	235
Lab 19-2: Creating a Measurement.....	236
Lab Objectives	236
Opening the Buffer Project	236
Adding a Trace	237
Defining the Measurement.....	237
Using the New Goal Function.....	238
Lab 19-3: Creating Plot Window Templates	239
Lab Objectives	239
Opening the Buffer Circuit	239
Using the New Plot Window Template	241
Lesson 20: Worst Case Analysis.....	242
Learning Objectives	242
Configuring the Analysis	244
Lab 20-1: Biased BJT Amplifier - Working WC	246
Lab Objectives	246
Creating the Circuit.....	246
Creating the Simulation Profile	246
Running the Simulation	248
Analyzing the Results	249
Lab 20-2: Mised Worst-Case.....	251
Lab Objectives	251
Editing the Rbmod Model.....	251
Running the Worst Case Analysis	251
What We Are Seeing.....	252
Lesson 21: Noise Analysis.....	253
Lesson Objectives	253
What Is Noise Analysis?.....	253
Definitions.....	253
Calculating Output and Input Noise	253
Setting up Noise Analysis	253
Lab 21-1: Noise Analysis.....	257
Lab Objectives	257
Configuring and Running Noise Analysis	257
Examining the Results in Probe	258

Feedback

Toc280986771	261
Frequently Asked Questions	277