TABLE OF CONTENTS

Release Notes ix
1. INTRODUCTION 1
1. TYPES OF ANALYSIS 1
1. DC Analysis 1
2. AC Small-Signal Analysis 2
3. Transient Analysis 2
4. Pole-Zero Analysis 2
5. Small-Signal Distortion Analysis 2
6. Sensitivity Analysis 3
7. Noise Analysis 3
2. ANALYSIS AT DIFFERENT TEMPERATURES 4
3. CONVERGENCE 6
2. CIRCUIT DESCRIPTION 7
1. GENERAL STRUCTURE AND CONVENTIONS 7
2. TITLE LINE, COMMENT LINES AND .END LINE 8
1. Title Line 8
2. .END Line 8
3. Comments 8
3. DEVICE MODELS 9
4. SUBCIRCUITS 10
1. .SUBCKT Line 10
2. .ENDS Line 10
3. Subcircuit Calls 11
5. COMBINING FILES: .INCLUDE LINES 12
3. CIRCUIT ELEMENTS AND MODELS 13
1. ELEMENTARY DEVICES 13
1. Resistors 13
2. Semiconductor Resistors 13
3. Semiconductor Resistor Model (R) 14
4. Capacitors 15
5. Semiconductor Capacitors 15
6. Semiconductor Capacitor Model (C) 16
7. Inductors 16
8. Coupled (Mutual) Inductors 17
9. Switches 17
10. Switch Model (SW/CSW) 18
2. VOLTAGE AND CURRENT SOURCES 19
1. Independent Sources 19
1. Pulse 20

2. Sinusoidal 21
3. Exponential 21
4. Piece-Wise Linear 22
5. Single-Frequency FM 23
2. Linear Dependent Sources 23
1. Linear Voltage-Controlled Current Sources 24
2. Linear Voltage-Controlled Voltage Sources 24
3. Linear Current-Controlled Current Sources 24
4. Linear Current-Controlled Voltage Sources 25
3. Non-linear Dependent Sources 25
3. TRANSMISSION LINES 27
1. Lossless Transmission Lines 27
2. Lossy Transmission Lines 27
3. Lossy Transmission Line Model (LTRA) 28
4. Uniform Distributed RC Lines (Lossy) 29
5. Uniform Distributed RC Model (URC) 30
4. TRANSISTORS AND DIODES 31
1. Junction Diodes 31
2. Diode Model (D) 32
3. Bipolar Junction Transistors (BJTs) 33
4. BJT Models (NPN/PNP) 33
5. Junction Field-Effect Transistors (JFETs) 35
6. JFET Models (NJF/PJF) 36
7. MOSFETs 37
8. MOSFET Models (NMOS/PMOS) 38
9. MESFETs 43
10. MESFET Models (NMF/PMF) 43
4. ANALYSES AND OUTPUT CONTROL 45
1. SIMULATOR VARIABLES (.OPTIONS) 45
2. INITIAL CONDITIONS 49
1. .NODESET: Specify Initial Node Voltage Guesses 49
2. .IC: Set Initial Conditions 49
3. ANALYSES 50
1. .AC: Small-Signal AC Analysis 50
2. .DC: DC Transfer Function 50
3. .DISTO: Distortion Analysis 51
4. .NOISE: Noise Analysis 52
5. .OP: Operating Point Analysis 53
6. .PZ: Pole-Zero Analysis 53
7. .SENS: DC or Small-Signal AC Sensitivity Analysis 54
8. .TF: Transfer Function Analysis 54
9. .TRAN: Transient Analysis 55

4. BATCH OUTPUT 56
1. .SAVE Lines 56
2. .PRINT Lines 56
3. .PLOT Lines 57
4. .FOUR: Fourier Analysis of Transient Analysis Output 58
5. INTERACTIVE INTERPRETER 59
1. EXPRESSIONS, FUNCTIONS, AND CONSTANTS 60
2. COMMAND INTERPRETATION 64
3. COMMANDS 65
1. Ac*: Perform an AC, small-signal frequency response analysis 65
2. Alias: Create an alias for a command 65
3. Alter*: Change a device or model parameter 65
4. Asciiplot: Plot values using old-style character plots 66
5. Aspice: Asynchronous spice run 66
6. Bug: Mail a bug report 66
7. Cd: Change directory 66
8. Destroy: Delete a data set 67
9. Dc*: Perform a DC-sweep analysis 67
10. Define: Define a function 67
11. Delete*: Remove a trace or breakpoint 68
12. Diff: Compare vectors 68
13. Display: List known vectors and types 68
14. Echo: Print text 68
15. Edit*: Edit the current circuit 69
16. Fourier: Perform a fourier transform 69
17. Hardcopy: Save a plot to a file for printing 69
18. Help: Print summaries of Spice3 commands 70
19. History: Review previous commands 70
20. Iplot*: Incremental plot 70
21. Jobs: List active asynchronous spice runs 70
22. Let: Assign a value to a vector 70
23. Linearize*: Interpolate to a linear scale 71
24. Listing*: Print a listing of the current circuit 71
25. Load: Load rawfile data 71
26. Op*: Perform an operating point analysis 72
27. Plot: Plot values on the display 72
28. Print: Print values 72
29. Quit: Leave Spice3 or Nutmeg 73
30. Rehash: Reset internal hash tables 73
31. Reset*: Reset an analysis 73
32. Reshape: Alter the dimensionality or dimensions of a vector 74
33. Resume*: Continue a simulation after a stop 74

34. Rspice: Remote spice submission 74
35. Run*: Run analysis from the input file 75
36. Rusage: Resource usage 76
37. Save*: Save a set of outputs 77
38. Sens*: Run a sensitivity analysis 77
39. Set: Set the value of a variable 77
40. Setcirc*: Change the current circuit 78
41. Setplot: Switch the current set of vectors 78
42. Settype: Set the type of a vector 78
43. Shell: Call the command interpreter 78
44. Shift: Alter a list variable 79
45. Show*: List device state 79
46. Showmod*: List model parameter values 80
47. Source: Read a Spice3 input file 80
48. Status*: Display breakpoint information 80
49. Step*: Run a fixed number of timepoints 81
50. Stop*: Set a breakpoint 81
51. Tf*: Run a Transfer Function analysis 81
52. Trace*: Trace nodes 82
53. Tran*: Perform a transient analysis 82
54. Transpose: Swap the elements in a multi-dimensional data set 82
55. Unalias: Retract an alias 83
56. Undefine: Retract a definition 83
57. Unset: Clear a variable 83
58. Version: Print the version of Spice 83
59. Where: Identify troublesome node or device 84
60. Write: Write data to a file 84
61. Xgraph: use the xgraph(1) program for plotting. 84

4. CONTROL STRUCTURES 85
1. While — End 85
2. Repeat — End 85
3. Dowhile — End 85
4. Foreach — End 86
5. If — Then — Else 86
6. Label 86
7. Goto 87
8. Continue 87
9. Break 87

5. VARIABLES 88
6. MISCELLANEOUS 92
7. BUGS 94

6. BIBLIOGRAPHY 97

A. APPENDIX A: EXAMPLE CIRCUITS 99
1. Circuit 1: Differential Pair 99
2. Circuit 2: MOSFET Characterization 99
3. Circuit 3: RTL Inverter 100
4. Circuit 4: Four-Bit Binary Adder 100
5. Circuit 5: Transmission-Line Inverter 102
B. APPENDIX B: MODEL AND DEVICE PARAMETERS 103
1. URC: Uniform R.C. line 104
2. ASRC: Arbitrary Source 104
3. BJT: Bipolar Junction Transistor 105
4. BSIM1: Berkeley Short Channel IGFET Model 108
5. BSIM2: Berkeley Short Channel IGFET Model 110
6. Capacitor: Fixed capacitor 114
7. CCCS: Current controlled current source 115
8. CCVS: Linear current controlled current source 115
9. CSwitch: Current controlled ideal switch 115
10. Diode: Junction Diode model 116
11. Inductor: Inductors 117
12. mutual: Mutual inductors 118
13. Isource: Independent current source 118
14. JFET: Junction Field effect transistor 119
15. LTRA: Lossy transmission line 120
16. MES: GaAs MESFET model 121
17. Mos1: Level 1 MOSfet model with Meyer capacitance model 123
18. Mos2: Level 2 MOSfet model with Meyer capacitance model 126
19. Mos3: Level 3 MOSfet model with Meyer capacitance model 129
20. Mos6: Level 6 MOSfet model with Meyer capacitance model 132
21. Resistor: Simple linear resistor 135
22. Switch: Ideal voltage controlled switch 135
23. Tranline: Lossless transmission line 136
24. VCCS: Voltage controlled current source 137
25. VCVS: Voltage controlled voltage source 137
26. Vsource: Independent voltage source 138

RELEASE NOTES

Spice3f is the six major release of Spice3. This release incorporates new features not available in Spice 3c or 3d, as well as several performance improvements. All of the features described here are believed to be fully functional. The development of SPICE and its algorithms is ongoing at Berkeley, and therefore not all of the intended capabilities of the program have been implemented in full yet.

Bugs in 3f2 fixed in 3f3:

```
Ascii (printer) plots in spice3f2 did not print bode plots
vs log of the frequency by default, as in spice2.
You had to explicity request the x-axis to be log; either
"plot vdb(2) xlog" (best) or "plot vdb(2) vs log10(frequency)" will do.
Now simply "plot vdb(2)" will work.

The on-line documentation has been brought up to date by converting this
manual into a format readable on-line.

Significant problems with AC sensitivities in 3f2 only have been fixed.

Multiple analyses and plots in spice2 emulation mode under 3f2 and earlier
generated misleading error messages.
This no longer happens in 3f3.
```

New features and bug fixes incorporated in Spice3f (the current release of Spice3):

```
Sensitivity analysis.
Added a parameter for fitting JFET models (parameter "B").
Fixed a discontinuity problem in MOS level 3
      (related to the "kappa" parameter).
Working "alter" command.
Improved "show" and "showmod" commands for operating
      point summary tables (like Spice2).
Working "trace" command.
Interactive "set" variable values now the same as ".options" settings.
Improved plotting, including implicitly transforming data for smith plots.
Added the function "deriv( )" (derivative) to the front-end.
Corrected an error affecting the specified initial conditions for
      some circuits.
Small signal behaviour of non-linear dependant sources fixed.
Miscellaneous bug fixes in the front end.
```

New features and bug fixes incorporated in Spice3e (the *previous* release of Spice3):

```
Lossy Transmission Lines.
Proper calculation of sheet resistance in the MOS models.
A new command ("where") to aid in debugging troublesome circuits.
Smith-chart plots working (see the "plot" command).
Arbitrary sources in subcircuits handled correctly.
Arbitrary source reciprocal calculations and DC biasing fixed.
Minor bug-fixes to the Pole-Zero analysis.
Miscellaneous bug fixes in the front end.
```

Some common problems remaining in Spice3f (note that this list is not complete):

```
Models defined within subcircuits are not always handled correctly.
If you have trouble, move the model definition outside of ".subckt"
and ".ends" lines.
```

```
Batch run data is not compacted if a "rawspice" data file is generated,
resulting in excessively large output files for some difficult inputs.

Sufficient detail is sometimes not preserved in transient analysis.
Providing a small value for the "TMAX" parameter (the fourth argument)
in the transient run command will solve this problem.

Convergence problems can sometimes be worked around by relaxing the
transient "TMAX" parameter.

The substrate node of the bipolar transistor (BJT) is modelled
incorrectly (this may actually be due to inherant numerical
problems with the model).  Do not use the substrate node; use
a semiconductor capacitor to model substrate effects.

Charge is not conserved in MOS devices based on the Meyer model.

Transient simulation of strictly resistive circuits (typical for
first runs or tests) allow a time step that is too large (e.g. a
sinusoidal source driving a resistor).  There is no integration error
to restrict the time step.  Use the "TMAX" parameter or include reactive
elements.

Deep nesting of subcircuits may exceed internal static buffers.

The PZ analysis can not be interrupted;  the sensitivity analysis
can not be continued (the interactive "resume" command) once interrupted.

There are many other small bugs, particularly in the front end.
```

Bug reporting:

Please send bug reports or suggested enhancements to Spice3 via electronic mail to:

```
spice@Berkeley.EDU  (Internet)
ucbvax!spice  (UUCP-net)
```

or by physical mail to "Spice3 Bug Report" at:

```
EECS/ERL Industrial Support Office
205 Cory Hall
U.C. Berkeley
Berkeley, CA   94720
```

Information on ordering Spice3 or other software distributed through the EECS/ERL Industrial Support Office should be directed to the above physical mail address or to "ilpsoftware@berkeley.edu" via electronic mail.

Include the version of Spice used, the system type and operating system version, the smallest input file demonstrating the problem, and the resulting (incorrect) output.