

TABLE OF CONTENTS

<i>Release Notes</i>	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

Spice3 User's Manual

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

Spice3 User's Manual

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. Aspic: 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

Spice3 User's Manual

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

Spice3 User's Manual

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

Spice3 User's Manual

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 explicitly 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.

Spice3 User's Manual

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 inherent 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.