HSPICE®
Quick Reference Guide

U-2003.03, March 2003

Comments?

E-mail your comments about Synopsys documentation to
doc@synopsys.com

Document Order Number: 37919-000 PA
HSPICE Quick Reference Guide, U-2003.03

HSPICE Quick Reference Guide

Table of Contents

Introduction

This Quick Reference Guide is a condensed version of the *HSPICE Simulation and Analysis User Guide* and the *HSPICE Applications Manual*. For more specific details and examples refer to the relevant manual.

Syntax Notation

xxx, yyy, zzz	Arbitrary alphanumeric strings
< ... >	Optional data fields are enclosed in angle brackets < >. All other symbols and punctuation are required.
UPPERCASE	Keywords, parameter names, etc. are represented in uppercase.
lowercase	Variables; should be replaced with a numeric or symbolic value.
...	Any number of parameters of the form shown can be entered.
+	Continuation of the preceding line.

The meaning of a parameter may depend on its location in the statement. Be sure that a complete set of parameters is entered in the correct sequence before running the simulation.

Common Abbreviations

Å	Angstrom
amp	ampere
cm	centimeter
deg	degree Centigrade (unless specified as Kelvin)
ev	electron volt
F	farad
H	Henry
m	meter
s	second
V	volt

Input and Output Files

General Form	/usr/george/mydesign.sp
/usr/george/	The design path.
mydesign	The design name.
mydesign	The design root.
tr0	The suffix.

File Name Suffix

X increments for each .TEMP or .ALTER. If X <= 36, X is one of the characters 0-9, A-Z.If X is > 36, use .OPTION ALT999 or ALT9999, as described in the HSPICE User Guide.

Input:	
input netlist	.sp
design configuration	.cfg
Output	(X is alter number, usually 0) (N is the statement number in one netlist, starting at 0).
graph data	.trX (transient analysis)
	.swX (dc sweep)
	.acX (ac analysis)
	.mtX (tran Measure)
	.msX (dc Measure)
	.maX (ac Measure)
	.pwlN_trX (from .STIM <TRAN> PWL)
	.datN_trX (from .STIM TRAN DATA)
	.datN_acX (from .STIM AC DATA)
	.datN_swX (from .STIM DC DATA)
	.vecN_trX (from .STIM <TRAN> VEC)
hardcopy data	.grX (from .GRAPH)

Input Netlist File

For a complete description of HSPICE installation, system configuration, setup and basic operation, please refer to the *HSPICE Simulation and Analysis User Guide*. HSPICE now accepts input line lengths of 1024 characters.

Sample Input Netlist File Structure

.TITLE	Implicit first line; becomes input netlist file title.
* or $	Comments to describe the circuit.
.OPTION <.TRAN> <.AC> <.DC> <.OP>	Set conditions for simulation analysis.
.TEMPERATURE	Sets the circuit temperatures for the entire circuit simulation.
PRINT/PLOT/ GRAPH/PROBE	Sets print, plot, graph, and probe variables.
.IC or .NODESET	Sets input state; can also be put in initial conditions.
SOURCES	Sets input stimulus.
NETLIST	Circuit description.
.MACRO libraries	.LIBRARY and .INC.
<.PROTECT>	Suppresses the printout of the text from the list file.
<.UNPROTECT>	Restores output printback.
.ALTER	Sequence for inline case analysis.
.PARAMETER	Defines a parameter.
.END	Terminates any ALTERs and the simulation.

Numeric Scale Factors

A number may be an integer, a floating point number, an integer or floating point number followed by an integer exponent, or an integer or floating point number followed by one of the scale factors listed below.

A	=1e-18
F	=1e-15
P	=1e-12
N	=1e-9

U	=1e-6
M	=1e-3
K	=1e3
MEG (or X)	=1e6
MI	=25.4e6
G	=1e9

Algebraic Expressions

In addition to simple arithmetic operations (+, -, *, /), the following quoted string functions may be used:

sin(x)	sinh(x)	abs(x)	cos(x)	cosh(x)
min(x,y)	tan(x)	tanh(x)	max(x,y)	atan(x)
sqrt(x)	exp(x)	db(x)	log(x)	log10(x)
pwr(x,y)	pow(x,y)	(instead of x**y)		

Algebraic Expressions as Input

General Form	'algebraic expression'

Either single (' ') or double (" ") quotes may be used.

Algebraic Expressions as Output

General Form	PAR ('algebraic expression')

The left and right parentheses are mandatory.

Equation Constants

ε_o	Vacuum permittivity=8.854e-12 F/m
εox	3.453143e-11 F/m
εsi	1.0359e-10 *F/m* dielectric constant of silicon
f	Frequency
k	1.38062e-23 - Boltzmann's constant
q	1.60212e-19 - Electron charge
t	Temperature in degrees Kelvin
Δt	t - tnom

tnom	Nominal temperature in degrees Kelvin (user-input in degrees C). Tnom = 273.15 + TNOM
vt(t)	k · t/q Thermal voltage
vt(tnom)	k · tnom/q Thermal voltage

Behavior Macromodeling

HSPICE performs the following types of behavioral modeling.

Subcircuit/Macros

.SUBCKT or .MACRO Statement

General Form	.SUBCKT subnam n1 <n2 n3 …> + <parnam=val …>
Or	.MACRO subnam n1 <n2 n3 …> + <parnam=val …>
n1 …	Node numbers for external reference
parnam	A parameter name set to a value or another parameter
subnam	Reference name for the subcircuit model call

See “.SUBCKT or .MACRO Statement” in the *HSPICE Simulation and Analysis User Guide*.

.ENDS or .EOM Statement

General Form	.ENDS <SUBNAM>
Or	.EOM <SUBNAM>

See “.ENDS or .EOM Statement” in the *HSPICE Simulation and Analysis User Guide*.

Subcircuit Calls

General Form	Xyyy n1 <n2 n3 …> subnam + <parnam=val …> <M=val>
M	Multiplier
n1 …	Node names for external reference
parnam	A parameter name set to a value for use only in the subcircuit
subnam	Subcircuit model reference name
Xyyy	Subcircuit element name

See “Subcircuit Call Statement” in the *HSPICE Simulation and Analysis User Guide*.

Voltage and Current Controlled Elements

HSPICE supports the following voltage and current controlled elements. For detailed information, see "Voltage and Current Controlled Elements" in the *HSPICE Simulation and Analysis User Guide*.

E Elements

Voltage Controlled Voltage Source—VCVS

Linear

General Form	E*xxx* *n+* *n-* <VCVS> *in+* *in-* *gain* + <MAX=*val*> <MIN=*val*> <SCALE=*val*> + <TC1=*val*> <TC2=*val*><ABS=1> + <IC=*val*>

Polynomial

General Form	E*xxx* *n+* *n-* <VCVS> POLY(*NDIM*) *in1+* + *in1-* ... *inndim+* *inndim-* + <TC1=*val*> <TC2=*val*> <SCALE=*val*> + <MAX=*val*> <MIN=*val*> <ABS=1> + *p0* <*p1*…> <IC=*val*>

Piecewise Linear

General Form	E*xxx* *n+* *n-* <VCVS> PWL(1) *in+* + *in-* <DELTA=*val*> <SCALE=*val*> + <TC1=*val*> <TC2=*val*> *x1*,*y1* + *x2*,*y2* ... *x100*,*y100* + <IC=*val*>

Multi-Input Gates

General Form	E*xxx* *n+* *n-* <VCVS> *gatetype*(*k*) + *in1+* *in1-* ... *inj+* *inj-* + <DELTA=*val*> <TC1=*val*> + <TC2=*val*> <SCALE=*val*> + *x1*,*y1* ... *x100*,*y100* + <IC=*val*>

Delay Element

General Form	E*xxx* *n*+ *n*- <VCVS> DELAY *in*+ + *in*- TD=*val* <SCALE=*val*> + <TC1=*val*> <TC2=*val*> + <NPDELAY=*val*>

See “Voltage-Controlled Voltage Source (VCVS)” in the *HSPICE Simulation and Analysis User Guide*.

Behavioral Voltage Source

General Form	E*xxx* *n*+ *n*- VOL=’*equation*’ + <MAX=*val*> <MIN=*val*>

See “Voltage and Current Controlled Elements” in the *HSPICE Simulation and Analysis User Guide*.

Ideal Op-Amp

General Form	`E`*`xxx n`*`+` *`n`*`- OPAMP` *`in`*`+` *`in`*`-`

See “Ideal Op-Amp” in the *HSPICE Simulation and Analysis User Guide*.

Ideal Transformer

General Form	E*xxx* *n*+ *n*- TRANSFORMER *in*+ *in*- *k*

See “Ideal Transformer” in the *HSPICE Simulation and Analysis User Guide*.

E Element Parameters

Parameter	Description
ABS	Output is absolute value if ABS=1.
DELAY	Keyword for the delay element.
DELTA	Controls the curvature of the piecewise linear corners.
Exxx	Voltage-controlled element name.
gain	Voltage gain.
gatetype(k)	Can be AND, NAND, OR, or NOR.
IC	Initial condition.
in +/-	Positive or negative controlling nodes.
k	Ideal transformer turn ratio.
MAX	Maximum output voltage value.

Parameter	Description
MIN	Minimum output voltage value.
n+/-	Positive or negative node of a controlled element.
NDIM	Number of polynomial dimensions.
NPDELAY	Sets the number of data points to use in delay simulations.
OPAMP	Keyword for an ideal op-amp element.
P0, P1…	Polynomial coefficients.
POLY	Polynomial keyword.
PWL	Piecewise linear function keyword.
SCALE	Element value multiplier.
TC1, TC2	First-order and second-order temperature coefficients.
TD	Time (propagation) delay keyword.
TRANSFORMER	Keyword for an ideal transformer.
VCVS	Keyword for a voltage-controlled voltage source.
x1,…	Controlling voltage across the in+ and in- nodes.
y1,…	Corresponding element values of *x*.

See “E Element Parameters” in the *HSPICE Simulation and Analysis User Guide*.

F Elements

Current Controlled Current Sources—CCCS

Linear

General Form	F*xxx n+ n-* <CCCS> *vn1 gain* + <MAX=*val*> <MIN=*val*> + <SCALE=*val*> <TC1=*val*> + <TC2=*val*> <M=*val*> <ABS=1> + <IC=*val*>

Polynomial

General Form	F*xxx n+ n-* <CCCS> POLY(*ndim*) + *vn1* <... *vnndim*> <MAX=*val*> + <MIN=*val*> <TC1=*val*> + <TC2=*val*> <SCALE=*val*> + <M=*val*> <ABS=1> *p0* <*p1*…> + <IC=*val*>

Piecewise Linear

General Form	F*xxx n*+ *n*- <CCCS> PWL(1) *vn1* + <DELTA=*val*> <SCALE=*val*> + <TC1=*val*> <TC2=*val*> <M=*val*> + *x1,y1* ... *x100,y100* + <IC=*val*>

Multi-Input Gates

General Form	F*xxx n*+ *n*- <CCCS> *gatetype*(*k*) + *vn1*, ... *vnk* <DELTA=*val*> + <SCALE=*val*> <TC1=*val*> + <TC2=*val*> <M=*val*> <ABS=1> + *x1,y1* ... *x100,y100* + <IC=*val*>

Delay Element

General Form	F*xxx n*+ *n*- <CCCS> DELAY *vn1* + TD=*val* <SCALE=*val*> + <TC1=*val*> <TC2=*val*> + NPDELAY=*val*

See “Current-Controlled Current Source (CCCS)” in the *HSPICE Simulation and Analysis User Guide*.

F Element Parameters

Parameter	Heading
ABS	Output is absolute value if ABS=1.
CCCS	Keyword for current-controlled current source.
DELAY	Keyword for the delay element.
DELTA	Controls the curvature of piecewise linear corners.
Fxxx	Current-controlled current source element name.
gain	Current gain.
gatetype(k)	Can be AND, NAND, OR, or NOR.
IC	Initial condition (estimate).
M	Number of element replications, in parallel.
MAX	Maximum output current value.
MIN	Minimum output current value.
n+/-	Positive or negative controlled source connecting nodes.

Parameter	Heading
NDIM	Number of polynomial dimensions. Must be a positive number. Default=one dimension.
NPDELAY	Number of data points to use in delay simulations.
P0, P1…	Polynomial coefficients.
POLY	Polynomial keyword.
PWL	Piecewise linear function keyword.
SCALE	Element value multiplier.
TC1, TC2	First-order and second-order temperature coefficients.
TD	Time (propagation) delay keyword.
vn1…	Names of voltage sources, through which the controlling current flows.
x1,…	Controlling current, through the *vn1* source.
y1,…	Corresponding output current values of *x*.

See “F Element Parameters” in the *HSPICE Simulation and Analysis User Guide*.

G Elements

Voltage Controlled Current Source—VCCS

Linear

General Form	G*xxx* *n*+ *n*- <VCCS> *in*+ *in*- + transconductance <MAX=*val*> + <MIN=*val*> <SCALE=*val*> + <M=*val*> <TC1=*val*> <TC2=*val*> + <ABS=1> <IC=*val*>

Polynomial

General Form	G*xxx* *n*+ *n*- <VCCS> POLY(*NDIM*) + *in1*+ *in1*- ... + <*inndim*+ *inndim*-> MAX=*val*> + <MIN=*val*> <SCALE=*val*> + <M=*val*> <TC1=*val*> <TC2=*val*> + <ABS=1> P0<P1…> <IC=*vals*>

Piecewise Linear

General Form	G*xxx* *n+* *n-* <VCCS> PWL(1) *in+* + *in-* <DELTA=*val*> <SCALE=*val*> + <M=*val*> <TC1=*val*> <TC2=*val*> + *x1*,*y1* *x2*,*y2* ... *x100*,*y100* + <IC=*val*> <SMOOTH=*val*>
Or	G*xxx* *n+* *n-* <VCCS> NPWL(1) *in+* + *in-* <DELTA=*val*> <SCALE=*val*> + <M=*val*> <TC1=*val*><TC2=*val*> + *x1*,*y1* *x2*,*y2* ... *x100*,*y100* + <IC=*val*> <SMOOTH=*val*>
Or	G*xxx* *n+* *n-* <VCCS> PPWL(1) *in+* + *in-* <DELTA=*val*> <SCALE=*val*> + <M=*val*> <TC1=*val*> <TC2=*val*> + *x1*,*y1* *x2*,*y2* ... *x100*,*y100* + <IC=*val*> <SMOOTH=*val*>

Multi-Input Gates

General Form	G*xxx* *n+* *n-* <VCCS> *gatetype*(*k*) + *in1+* *in1-* ... *ink+* *ink-* + <DELTA=*val*> <TC1=*val*> + <TC2=*val*> <SCALE=*val*> + <M=val> *x1*,*y1* ... + *x100*,*y100*<IC=*val*>

Delay Element

General Form	G*xxx* *n+* *n-* <VCCS> DELAY *in+* + *in-* TD=*val* <SCALE=*val*> + <TC1=*val*> <TC2=*val*> + NPDELAY=*val*

See “Voltage-Controlled Current Source (VCCS)” in the *HSPICE Simulation and Analysis User Guide*.

Behavioral Current Source

General Form	G*xxx* *n+* *n-* CUR=’*equation*’ +<MAX>=*val*> <MIN=*val*> <M=*val*> +<SCALE=*val*>

See “Behavioral Current Source” in the *HSPICE Simulation and Analysis User Guide*.

Voltage Controlled Resistor—VCR

Linear

General Form	G*xxx n+ n-* VCR *in+ in-* + transfactor <MAX=*val*> + <MIN=*val*> <SCALE=*val*> + <M=*val*> <TC1=*val*> <TC2=*val*> + <IC=*val*>

Polynomial

General Form	G*xxx n+ n-* VCR POLY(NDIM) *in1+* + *in1-* ... <*inndim+ inndim-*> + <MAX=*val*> <MIN=*val*> + <SCALE=*val*> <M=*val*> + <TC1=*val*> <TC2=*val*> + P0 <P1…> <IC=*vals*>

Piecewise Linear

General Form	G*xxx n+ n-* VCR PWL(1) *in+ in-* + <DELTA=*val*> <SCALE=*val*> + <M=*val*> <TC1=*val*> <TC2=*val*> + *x1*,*y1 x2*,*y2* ... *x100*,*y100* + <IC=*val*> <SMOOTH=*val*>
Or	G*xxx n+ n-* VCR NPWL(1) *in+ in-* + <DELTA=*val*> <SCALE=*val*> + <M=*val*> <TC1=*val*> <TC2=*val*> + *x1*,*y1 x2*,*y2* ... *x100*,*y100* + <IC=*val*> <SMOOTH=*val*>
Or	G*xxx n+ n-* VCR PPWL(1) *in+ in-* + <DELTA=*val*> <SCALE=*val*> + <M=*val*> <TC1=*val*> <TC2=*val*> + *x1*,*y1 x2*,*y2* ... *x100*,*y100* + <IC=*val*> <SMOOTH=*val*>

Multi-Input Gates

General Form	G*xxx n+ n-* VCR *gatetype*(*k*) + *in1+ in1-* ... *ink+ ink-* + <DELTA=*val*> <TC1=*val*> + <TC2=*val*> <SCALE=*val*> + <M=*val*> *x1*,*y1* ... *x100*,*y100* + <IC=*val*>

See “Voltage-Controlled Resistor (VCR)” in the *HSPICE Simulation and Analysis User Guide*.

Voltage Controlled Capacitors—VCCAP

General Form	G*xxx* *n+* *n-* VCCAP PWL(1) *in+* + *in-* <DELTA=*val*> + <SCALE=*val*> <M=*val*> + <TC1=*val*> <TC2=*val*> + *x1*,*y1* *x2*,*y2* ... *x100*,*y100* + <IC=*val*> <SMOOTH=*val*>

See “Voltage-Controlled Capacitor (VCCAP)” in the *HSPICE Simulation and Analysis Manual*.

G Element Parameters

Parameter	Description
ABS	Output is absolute value, if ABS=1.
CUR= equation	Current output which flows from n+ to n-.
DELAY	Keyword for the delay element.
DELTA	Controls the curvature of the piecewise linear corners.
Gxxx	Voltage-controlled element name.
gatetype(k)	Can be AND, NAND, OR, or NOR.
IC	Initial condition.
in +/-	Positive or negative controlling nodes.
M	Number of element replications in parallel.
MAX	Maximum current or resistance value.
MIN	Minimum current or resistance value.
n+/-	Positive or negative node of the controlled element.
NDIM	Number of polynomial dimensions.
NPDELAY	Sets the number of data points to use in delay simulations.
NPWL	Models the symmetrical bidirectional switch or transfer gate, NMOS.
p0, p1 …	Polynomial coefficients.
POLY	Polynomial keyword.
PWL	Piecewise linear function keyword.
PPWL	Models the symmetrical bidirectional switch or transfer gate, PMOS.
SCALE	Element value multiplier.
SMOOTH	For piecewise-linear, dependent-source elements, SMOOTH selects curve smoothing.

Parameter	Description
TC1,TC2	First- and second-order temperature coefficients.
TD	Time (propagation) delay keyword.
transconduct-ance	Voltage-to-current conversion factor.
transfactor	Voltage-to-resistance conversion factor.
VCCAP	Keyword for voltage-controlled capacitance element.
VCCS	Keyword for voltage-controlled current source.
VCR	Keyword for the voltage controlled resistor element.
x1, ...	Controlling voltage, across the *in*+ and *in*- nodes.
y1, ...	Corresponding element values of *x*.

See “G Element Parameters” in the *HSPICE Simulation and Analysis User Guide*.

H Elements

Current Controlled Voltage Source—CCVS

Linear

General Form	H*xxx n+ n-* <CCVS> *vn1* + transresistance <MAX=*val*> + <MIN=*val*> <SCALE=*val*> + <TC1=*val*><TC2=*val*> <ABS=1> + <IC=*val*>

Polynomial

General Form	H*xxx n+ n-* <CCVS> POLY(*NDIM*) + *vn1* <... *vnndim*> <MAX=*val*> + <MIN=*val*> <TC1=*val*> + <TC2=*val*> <SCALE=*val*> + <ABS=1> *P0* <*P1*…> <IC=*val*>

Piecewise Linear

General Form	Hxxx n+ n- <CCVS> PWL(1) vn1 + <DELTA=val> <SCALE=val> + <TC1=val> <TC2=val> x1,y1 ... + x100,y100 <IC=val>

Multi-Input Gates

General Form	H*xxx n+ n- gatetype(k)* + *vn1*, ... *vnk* <DELTA=*val*> + <SCALE=*val*> <TC1=*val*> + <TC2=*val*> *x1*,*y1* ... + *x100*,*y100* <IC=*val*>

Delay Element

General Form	H*xxx n+ n-* <CCVS> DELAY *vn1* + TD=*val* <SCALE=*val*><TC1=*val*> + <TC2=*val*> <NPDELAY=*val*>

See “Current-Controlled Voltage Source (CCVS)” in the *HSPICE Simulation and Analysis User Guide*.

H Element Parameters

Parameter	Description
ABS	Output is absolute value if ABS=1.
CCVS	Keyword for current-controlled voltage source.
DELAY	Keyword for the delay element.
DELTA	Controls the curvature of piecewise linear corners.
gatetype(k)	Can be AND, NAND, OR, or NOR.
Hxxx	Current-controlled voltage source element name.
IC	Initial condition.
MAX	Maximum voltage value.
MIN	Minimum voltage value.
n+/-	Positive or negative controlled source connecting nodes.
NDIM	Number of polynomial dimensions.
NPDELAY	Number of data points to use in delay simulations.
P0, P1…	Polynomial coefficients.
POLY	Polynomial dimension.
PWL	Piecewise linear function keyword.
SCALE	Element value multiplier.
TC1, TC2	First-order and second-order temperature coefficients.
TD	Time (propagation) delay keyword.
trans-resistance	Current-to-voltage conversion factor.

Parameter	Description
vn1…	Names of voltage sources, through which the controlling current flows.
x1,…	Controlling current, through the *vn1* source.
y1,…	Corresponding output voltage values of *x*.

See “H Element Parameters” in the *HSPICE Simulation and Analysis User Guide*.

Op-Amp Element Statement

COMP=0	xa1 in- in+ out vcc vee modelname AV=val
Or	
COMP=1	xa1 in- in+ out comp1 comp2 vcc vee modelname AV=val
in+	Noninverting input
in-	Inverting input
modelname	Subcircuit reference name
out	Output, single ended
vcc	Positive supply
vee	Negative supply

See “Op-Amp Element Statement Format” in the *HSPICE Applications Manual*.

Op-Amp .MODEL Statement

General Form	.MODEL mname AMP parameter=value …
AMP	Identifies an amplifier model
mname	Model name. Elements reference the model by this name.
parameter	Any model parameter described below
value	Value assigned to a parameter

See “Op-Amp .MODEL Statement Format” in the *HSPICE Applications Manual*.

Controlling Input

For complete definitions, see the *HSPICE Simulation and Analysis User Guide*, “Specifying Simulation Input and Controls.”

.OPTION Statement

General Form	.OPTION *opt1 <opt2 opt3 …>*
opt1 …	Specifies any input control options.

See “.OPTION Statement” in the *HSPICE Simulation and Analysis User Guide*.

General Control (I/O) Options

Option	Description
ACCT	Reports job accounting and runtime statistics, at the end of the output listing.
ACOUT	AC output calculation method, for the difference in values of magnitude, phase, and decibels for prints and plots.
ALT999, ALT9999	Generates up to 1000 (`ALT999`) or 10,000 (`ALT9999`) unique output files from `.ALTER` simulation runs.
ALTCHK	Disables topology checking in elements redefined by the `.ALTER` statement.
BEEP	`BEEP=1` sounds an audible tone when simulation returns a message, such as *info: hspice job completed*. `BEEP=0` turns off the audible tone.
BINPRINT	Outputs binning parameters of the CMI MOSFET model. Currently available only for Level 57.
BRIEF, NXX	Stops print back of data file until HSPICE finds an `.OPTION BRIEF = 0`, or the `.END` statement.
CO = x	Sets the number of columns for printout: *x* can be either `80` (for narrow printout) or `132` (for wide carriage printouts).
INGOLD = x	Specifies the printout data format.
LENNAM = x	Maximum length of names, in the printout of operating point analysis results.

Option	Description
LIST, VERIFY	Produces an element summary of the input data to print.
MEASDGT = x	Formats the `.MEASURE` statement output, in both the listing file and the `.MEASURE` output files (*.ma0*, *.mt0*, *.ms0*, and so on).
NODE	Prints a node cross reference table.
NOELCK	Bypasses element checking, to reduce pre-processing time for very large files.
NOMOD	Suppresses printout of model parameters
NOPAGE	Suppresses page ejects for title headings
NOTOP	Suppresses topology checks, to increase speed for pre-processing very large files
NUMDGT = x	Number of significant digits to print, for output variable values.
NXX	Same as BRIEF. See BRIEF.
OPTLST = x	Outputs additional optimization information: 0 No information (default). 1 Prints parameter, Broyden update, and bisection results information. 2 Prints gradient, error, Hessian, and iteration information. 3 Prints all of the above, and Jacobian.
OPTS	Prints the current settings, for all control options.
PATHNUM	Prints subcircuit path numbers, instead of path names
PLIM = x	Specifies plot size limits, for current and voltage plots.
POSTTOP=*n*	Outputs instances, up to *n* levels deep. `.OPTION POST` saves all nodes, at all levels of hierarchy. `.OPTION POSTTOP` or `.OPTION POSTTOP=1` saves only the `TOP` node. `.OPTION POSTTOP=2` saves only nodes at the top two levels.
POST_VERSION = x	Sets the post-processing output version with values `x=9601`, `9007`, or `2001`..
STATFL	Controls if HSPICE creates a `.st0` file. `statfl=0` (default) outputs a `.st0` file. `statfl=1` suppresses the `.st0` file.
SEARCH	Search path for libraries and included files.

Option	Description
VERIFY	Same as LIST. See LIST.

See “General Control Options” in the *HSPICE Simulation and Analysis User Guide*.

IBIS PKG and EDB Simulation Input

Option	Description
EBDMAP	Name of a map file, which lists the relationship between HSPICE sub-circuit names and: IBIS board-level module. *X* element name in the sub-circuit. On-board component.
EBDTYPE	Type of elements to use, to represent board-level pin connected traces.
PKGMAP:	Name of EBD map file. This file lists the relationship between the HSPICE subcircuit and the IBIS component.
PKGTYPE	Types of elements to use, to represent the package effect.

See “Using PKG and EBD Simulation” in the *HSPICE User Guide*.

CPU Options

Option	Description
CPTIME = x	Maximum CPU time, in seconds, allotted for this simulation job.
EPSMIN = x	Smallest number that a computer can add or subtract, a constant value.
EXPMAX = x	Largest exponent that you can use for an exponential, before overflow occurs.
LIMTIM = x	Amount of CPU time reserved to generate prints and plots, if a CPU time limit (`CPTIME =` *x*) terminates simulation.

See “CPU Options” in the *HSPICE Simulation and Analysis User Guide*.

Interface Options

Option	Description
ARTIST = x	`ARTIST = 2` enables Cadence Analog Artist interface. Requires a specific license.
CDS, SDA	`CDS = 2` produces a Cadence WSF (ASCII format) post-analysis file for Opus™. Requires a specific license.
CSDF	Selects Common Simulation Data Format (Viewlogic-compatible graph data file).
DLENCSDF	How many digits to use with Viewlogic-compatible graph data file format.
MEASOUT	Outputs `.MEASURE` statement values and sweep parameters into an ASCII file, for post-analysis processing using AvanWaves or other analysis tools.
MENTOR = x	`MENTOR = 2` enables the Mentor MSPICE-compatible (ASCII) interface. Requires a specific license.
MONTECON	Continues Monte Carlo analysis. Retrieves next random value, even if non-convergence occurs.
POST = x	Stores simulation results for analysis, using AvanWaves interface or other methods. `POST = 1` saves results in binary. `POST = 2` saves results in ASCII. `POST = 3` saves results in New Wave binary format.
post_version =2001	Sets the post-processing output version with a value of 2001.
PROBE	Limits post-analysis output to only variables specified in `.PROBE`, `.PRINT`, `.PLOT`, and `.GRAPH` statements.
PSF = x	Specifies if HSPICE outputs binary or ASCII data from the Parameter Storage Format.
SDA	Same as CDS. See CDS.
ZUKEN = x	If x is 2, enables Zuken interactive interface. If x is 1 (default), disables this interface.

See “Interface Options” in the *HSPICE Simulation and Analysis User Guide*.

Analysis Options

Option	Description
ASPEC	Sets HSPICE to ASPEC-compatibility mode.
FFTOUT	Prints out 30 harmonic fundamentals, sorted by size, THD, SNR, and SFDR.
LIMPTS = x	Number of points to print or plot in AC analysis.
PARHIER	Selects parameter-passing rules that control evaluation order of subcircuit parameters.
SPICE	Makes HSPICE compatible with Berkeley SPICE.
SEED	Starting seed for a random-number generator, for Monte Carlo analysis.

See “Analysis Options” in the *HSPICE Simulation and Analysis User Guide*.

Error Options

Option	Description
BADCHR	Generates a warning, when it finds a non-printable character in an input file.
DIAGNOSTIC	Logs negative model conductances.
NOWARN	Suppresses all warning messages, except those generated from statements in .ALTER blocks.
WARNLIMIT = x	Limits how many times certain warnings appear in the output listing. This reduces the output listing file size.

See “Error Options” in the *HSPICE Simulation and Analysis User Guide*.

Version Options

Option	Description
H9007	Sets default values for general-control options, to correspond to the values for HSPICE Release H9007D.

See “Version Options” in the *HSPICE Simulation and Analysis User Guide*.

Model Analysis Options

See “Model Analysis Options” in the *HSPICE Simulation and Analysis User Guide*.

General Options

Option	Description
DCAP	Selects equations, to calculate depletion capacitance for LEVEL 1 or 3 diodes, BJTs.
hier_scale	Defines how HSPICE interprets the *S* parameter as a user-defined parameter or an HSPICE scale parameter.
MODSRH	If MODSRH=1, HSPICE does not load or reference a model described in a `.MODEL` statement, if the netlist does not use that model. This option can shorten simulation run time. Default is `MODSRH=0`.
SCALE	Element scaling factor.
TNOM	Reference temperature for simulation.
MODMONTE	If `MODMONTE=1`, then each device receives a different random value for its Monte Carlo parameters. If `MODMONTE=0` (default), then each device receives the same random value for its Monte Carlo parameters.

MOSFET Control Options

Option	Description
CVTOL	Changes the number of numerical integration steps, when calculating gate capacitor charge for a MOSFET, using CAPOP = 3.
DEFAD	Default value for MOSFET drain diode area.
DEFAS	Default value for MOSFET source diode area.
DEFL	Default value for MOSFET channel length.
DEFNRD	Default number of squares for drain resistor on a MOSFET.
DEFNRS	Default number of squares for source resistor on a MOSFET.
DEFPD	Default MOSFET drain diode perimeter.
DEFPS	Default MOSFET source diode perimeter.
DEFW	Default MOSFET channel width.
SCALM	Model scaling factor.

Option	Description
WL	Reverses specified order in the VSIZE MOS element. Default order is length-width; changes order to width-length.

See “MOSFET Control Options” in the *HSPICE Simulation and Analysis User Guide*.

Inductor Options

You can use the following inductor options in HSPICE:

GENK	Automatically computes second-order mutual inductance, for several coupled inductors.
KLIM	Minimum mutual inductance, below which automatic second-order mutual inductance calculation no longer proceeds.

BJT and Diode Options

EXPLI	Current-explosion model parameter. PN junction characteristics above explosion current are linear.

DC Solution Control Options

Option	Description
ABSH = x	Sets the absolute current change, through voltage-defined branches (voltage sources and inductors).
ABSI = x	Sets the absolute branch current error tolerance in diodes, BJTs, and JFETs during DC and transient analysis.
ABSMOS = x	Current error tolerance (for MOSFET devices), in DC or transient analysis.
ABSTOL = x	ABSTOL is an alias for ABSI. See ABSI.
ABSVDC = x	Sets the absolute minimum voltage, for DC and transient analysis.
DI = x	Sets the maximum iteration-to-iteration current change, through voltage-defined branches (voltage sources and inductors).
KCLTEST	Starts KCL (Kirchhoff’s Current Law) test.
MAXAMP = x	Sets the maximum current, through voltage-defined branches (voltage sources and inductors).

Option	Description
RELH = x	Relative current tolerance, through voltage-defined branches (voltage sources and inductors).
RELI = x	Relative error/tolerance change, from iteration to iteration. Determines convergence for all currents in diode, BJT, and JFET devices.
RELMOS = x	Sets error tolerance (percent) for drain-to-source current, from iteration to iteration. Determines convergence for currents in MOSFET devices.
RELV = x	Relative error tolerance for voltages.
RELVDC = x	Relative error tolerance for voltages.

See “DC Operating Point, DC Sweep, and Pole/Zero Options” in the *HSPICE Simulation and Analysis User Guide*.

Matrix Options

ITL1 = x	Maximum DC iteration limit.
ITL2 = x	Iteration limit for the DC transfer curve.
NOPIV	Prevents HSPICE from automatically switching to pivoting matrix factors.
PIVOT = x	Selects a pivot algorithm.
PIVREF	Pivot reference.
PIVREL = x	Maximum/minimum row/matrix ratio.
PIVTOL = x	Absolute minimum value for which HSPICE or accepts a matrix entry as a pivot.
SPARSE = x	Same as PIVOT.

Pole/Zero I/O Options

CAPTAB	Prints table of single-plate node capacitance for diodes, BJTs, MOSFETs, JFETs, and passive capacitors at each operating point.
DCCAP	Generates C-V plots, and prints capacitance values of a circuit (both model and element), during a DC analysis.
VFLOOR = x	Minimum voltage to print in output listing.

Convergence Options

CONVERGE	Invokes different methods to solve non-convergence problems
CSHDC	The same option as CSHUNT; use only with the CONVERGE option.
DCFOR = x	Use with DCHOLD and .NODESET, to enhance DC convergence.
DCHOLD = x	Use DCFOR and DCHOLD together, to initialize a DC analysis.
DCON = X	If a circuit cannot converge, HSPICE or automatically sets DCON = 1.
DCSTEP = x	Converts DC model and element capacitors to a conductance, to enhance DC convergence properties.
DCTRAN	DCTRAN is an alias for CONVERGE. See CONVERGE.
DV = x	Maximum iteration-to-iteration voltage change, for all circuit nodes, in both DC and transient analysis.
GMAX = x	Conductance, in parallel with a current source, for .IC and .NODESET initialization circuitry.
GMINDC = x	Conductance in parallel to all pn junctions and all MOSFET nodes, for DC analysis.
GRAMP = x	HSPICE sets this value during autoconvergence.
GSHUNT	Conductance, added from each node to ground. Default=0.
ICSWEEP	Saves current analysis result of parameter or temperature sweep, as the starting point in the next analysis in the sweep.
ITLPTRAN	Controls the iteration limit used in the final try of the pseudo-transient method, in OP or DC analysis.
NEWTOL	Calculates one more iterations past convergence, for every calculated DC solution and timepoint circuit solution.
OFF	For all active devices, initializes terminal voltages to zero, if you did not initialize them to other values.
RESMIN = x	Minimum resistance for all resistors, including parasitic and inductive resistances.

Pole/Zero Control Options

Option	Description
CSCAL	Sets the capacitance scale. HSPICE multiplies capacitances by `CSCAL`.
FMAX	Sets the maximum frequency of angular velocity, for poles and zeros.
FSCAL	Sets the frequency scale, by which HSPICE or multiplies the frequency.
GSCAL	Sets the conductance scale.
LSCAL	Sets the inductance scale.
PZABS	Absolute tolerances, for poles and zeros.
PZTOL	Relative error tolerance, for poles or zeros.
RITOL	Minimum ratio for (real/imaginary), or (imaginary/real) parts of poles or zeros.
(X0R,X0I), (X1R,X1I), (X2R,X2I)	The three complex starting points, in the Muller pole/zero analysis algorithm.

See “Pole/Zero Control Options” in the *HSPICE Simulation and Analysis User Guide*.

Transient and AC Control Options

Option	Description
ABSH = x	Sets the absolute current change, through voltage-defined branches (voltage sources and inductors).
ABSV = x	Same as `VNTOL`. See `VNTOL`.
ACCURATE	Selects a time algorithm; uses `LVLTIM=3` and `DVDT = 2` for circuits such as high-gain comparators. Default is `0`.
ACOUT	AC output calculation method, for the difference in values of magnitude, phase, and decibels. Use this option for prints and plots. Default is `1`.
CHGTOL = x	Sets a charge error tolerance if you set `LVLTIM=2`. Default=`1e-15` (coulomb).
CSHUNT	Capacitance, added from each node to ground. Default is 0.
DI = x	Maximum iteration-to-iteration current change, through voltage-defined branches (voltage sources and inductors). Default is `0.0.`
GMIN = x	Minimum conductance added to all PN junctions, for a time sweep in transient analysis. Default is `1e-12`.

Option	Description
GSHUNT	Conductance, added from each node to ground. Default is zero.
MAXAMP = x	Maximum current, through voltage-defined branches (voltage sources and inductors). If current exceeds the `MAXAMP` value, HSPICE issues an error. Default=`0.0`.
RELH = x	Relative current tolerance, through voltage-defined branches (voltage sources and inductors). Default is `0.05`.
RELI = x	Relative error/tolerance change, from iteration to iteration. Default is `0.01` for `KCLTEST=0`, or `1e-6` for `KCLTEST=1`.
RELQ = x	Used in timestep algorithm, for local truncation error (`LVLTIM=2`). Default=`0.01`.
RELTOL, RELV	Relative error tolerance for voltages. Default is `1e-3`.
RISETIME	Smallest risetime of a signal, `.OPTION RISETIME = x`.
TRTOL = x	Used in timestep algorithm for local truncation error (`LVLTIM=2`). Default=`7.0`.
VNTOL = x, ABSV	Absolute minimum voltage, for DC and transient analysis. Default=`50` (microvolts).

See “Transient and AC Small Signal Analysis Options” in the *HSPICE Simulation and Analysis Manual*.

Speed Options

AUTOSTOP	Stops transient analysis, after calculating all `TRIG-TARG` and `FIND-WHEN` measure functions. To use `AUTOSTOP` with measure functions (such as , `AVG`, `RMS`, `MIN`, `MAX`, `PP`, `ERR`, `ERR1,2,3`, or `PARAM`), set `.OPTION autostop` from_to or `.OPTION autostsop from_to=1`.
BKPSIZ = x	Size of breakpoint table. Default=`5000`.
BYPASS	To speed-up simulation, does not update status of latent devices. Default is `1`.
BYTOL = x	Voltage tolerance, at which a MOSFET, MESFET, JFET, BJT, or diode becomes latent. Default is `MBYPASSxVNTOL`.
FAST	To speed-up simulation, does not update status of latent devices. Default is `0`.
ITLPZ	Sets the iteration limit for pole/zero analysis. Default is `100`.

MBYPASS = x	Computes default of BYTOL control option. Default is 1, for DVDT = 0, 1, 2, or 3. Default is 2, for DVDT = 4.
TRCON	Controls automatic convergence, and the speed of large non-linear circuits with large TSTOP/TSTEP values. Default=1.

Timestep Options

ABSVAR = x	Absolute limit for the maximum voltage change, from one time point to the next. Default is 0.5 (volts).
DELMAX = x	Maximum Delta of the internal timestep. HSPICE automatically sets the DELMAX value.
DVDT	Adjusts the timestep, based on rates of change for node voltage. Default=4. 0 - original algorithm 1 - fast 2 - accurate 3,4 - balance speed and accuracy
FS = x	Decreases Delta (internal timestep) by the specified fraction of a timestep (TSTEP), for the first time point of a transient. Default=0.25.
FT = x	Decreases Delta (the internal timestep), by a specified fraction of a timestep (TSTEP), for an iteration set that does not converge. Default is 0.25.
IMIN = x, ITL3 = x	Minimum number of iterations. Required to obtain convergence at a timepoint in transient analysis simulations. Determines internal timestep. Default is 3.0.
IMAX = x, ITL4 = x	Maximum number of iterations to obtain convergence at a timepoint in transient analysis. Determines internal timestep. Default is 8.0.
ITL5 = x	Iteration limit, for transient analysis output. Default is 0.0.
RELVAR = x	Used with ABSVAR, and DVDT timestep option. Sets relative voltage change, for LVLTIM=1 or 3. Default is 0.30 (30%).

RMAX = x	`TSTEP` multiplier, controls maximum value (`DELMAX`) to use for internal timestep Delta. Default is `5`, when `dvdt=4`, and `lvltim=1`. Otherwise, default=`2`. Maximum is `1e+9`, minimum is `1e-9`. Recommend maximum=`1e+5`.
RMIN = x	Sets the minimum value of Delta (internal timestep). Default=`1.0e-9`.
SLOPETOL = x	Minimum value, for breakpoint table entries in a piecewise linear (PWL) analysis. Default is `0.5`.
TIMERES = x	Minimum separation between breakpoint values, for breakpoint table. Default=`1` ps.

Algorithm Options

DVTR	Limits voltage in transient analysis. Default is `1000`.
IMAX = x, ITL4 = x	Maximum number of iterations to obtain convergence at a timepoint in transient analysis. Determines internal timestep. Default is `8.0`.
IMIN = x, ITL3 = x	Minimum number of iterations. Required to obtain convergence at a timepoint in transient analysis simulations. Determines internal timestep. Default is 3.0.
LVLTIM = x	Selects the timestep algorithm, for transient analysis. If `LVLTIM = 1` (default), HSPICE uses the DVDT timestep algorithm. If `LVLTIM = 2`, HSPICE uses the timestep algorithm for the local truncation error. If `LVLTIM = 3`, HSPICE uses the DVDT timestep algorithm, with timestep reversal.
MAXORD = x	Maximum order of integration, for the `GEAR` method (see `METHOD`).
METHOD = name	Sets numerical integration method, for a transient analysis, to `GEAR` or `TRAP`.
PURETP	Sets the integration method to use, for the reversal time point. Default = `0`.
MU = x	Coefficient, for trapezoidal integration. Range for `MU` is 0.0 to 0.5. Default=`0.5`.

TRCON	Controls the automatic convergence (*autoconvergence*) process. `TRCON=1` (default) enables autoconvergence, if the previous simulation run fails. To disable autoconvergence, set `TRCON=0` or `TRCON=-1`.

Input and Output Options

INTERP	Limits output for post-analysis tools, such as Cadence or Zuken, to only `.TRAN` timestep intervals.
ITRPRT	Prints output variables, at their internal timepoints.
MEASFAIL	If `measfail=0`, outputs `0` into the `.mt#`, `.ms#`, or `.ma#` file, and prints *failed* to the listing file. If `measfail=1` (default), prints *failed* into the `.mt#`, `.ms#`, or `.ma#` file, and into the listing file.
MEASSORT	Automatically sorts large numbers of `.measure` statements. `.OPTION MEASSORT=0` (default; does not sort `.MEASURE` statements). `.OPTION MEASSORT=1` (internally sorts `.MEASURE` statements).
PUTMEAS	Controls the output variables, listed in the `.MEASURE` statement. Default = `1`.
UNWRAP	Displays phase results from AC analysis, in *unwrapped* form (continuous phase plot).

Statements

HSPICE supports the following statements.

.ALTER Statement

General Form	.ALTER <title_string>

See ".ALTER Statement" in the *HSPICE Simulation and Analysis User Guide*.

Comments

General Form	**<Comment on a line by itself>*
Or	*<HSPICE statement>* $*<comment following HSPICE input>*

.ALIAS Statement

You can alias one model name to another:

`.alias pa1 par1`

During simulation, this `.alias` statement indicates to use the `par1` model, in place of a reference to a previously-deleted `pa1` model. See ".ALIAS Statement" in the *HSPICE Simulation and Analysis User Guide*.

.CONNECT Statement

Connects two nodes in your HSPICE netlist, so that simulation evaluates the two nodes as only one node. Both nodes must be at the same level in the circuit design that you are simulating: you cannot connect nodes that belong to different subcircuits.

Syntax

```
.connect node1 node2
```

where:

node1	Name of the first of two nodes to connect to each other.
node2	Name of the second of two nodes to connect to each other. The first node replaces this node in the simulation.

.DATA Statement

See ".DATA Statement" in the *HSPICE Simulation and Analysis User Guide*.

Inline .DATA Statement

General Form	.DATA *datanm pnam1 <pnam2* + *pnam3 …pnamxxx >* + *pval1<pval2 pval3 …* + *pvalxxx> pval1' <pval2'* + *pval3' …pvalxxx'>* .ENDDATA

External File .DATA Statement

General Form	.DATA *datanm* + MER FILE = ‘*filename1*’ + *pname1=colnum* + <*panme2=colnum* …> + <FILE = ‘*filename2*’ + *pname1 = colnum* + <*pname2 = colnum* …>> … + <OUT = ‘*fileout*’> .ENDDATA

Column Laminated .DATA Statement

General Form	.DATA *datanm* + LAM FILE=‘*filename1*’ + *pname1=colnum* + <*panme2=colnum* …> + <FILE=‘*filename2*’ + *pname1=colnum* + <*pname2=colnum* …>>… + <OUT = ‘*fileout*’> .ENDDATA
datanm	Specifies the data name referred to in the `.TRAN`, `.DC`, or `.AC` statement.
LAM	Specifies column-laminated (parallel merging) data files to use.
filenamei	Specifies the name of the data file to read.
MER	Specifies concatenated (series merging) data files to use.
pnami	Specifies the parameter names used for source value, element value, device size, model parameter value, and so on.
colnum	Specifies the column number in the data file, for the parameter value.
fileout	Specifies the name of the data file to write, with all of the data concatenated.
pvali	Specifies the parameter value.

See “Column Laminated .DATA Statement” in the *HSPICE User Guide*.

.DEL LIB Statement

General Form	.DEL LIB '<*filepath*>*filename*' + *entryname* .DEL LIB *libnumber entryname*
entryname	Entry name, used in the library call statement to delete.
filename	Name of a file to delete from the data file.
filepath	Path name of a file, if the operating system supports tree-structured directories.
libnumber	Library number, used in the library call statement to delete.

See ".DEL LIB Statement" in the *HSPICE Simulation and Analysis User Guide*.

Element Statements

General Form	elname <node1 node2 … nodeN> + <mname> <pname1 = val1> + <pname2 = val2> <M = val>
Or	elname <node1 node2 … nodeN> + <mname> <pname = 'expression'> + <M = val>
Or	elname <node1 node2 … nodeN> + <mname> <val1 val2 … valn>
	B IBIS buffer C Capacitor D Diode E,F,G,H Dependent current and voltage sources I Current source J JFET or MESFET K Mutual inductor L Inductor M MOSFET Q BJT R Resistor S S element T,U,W Transmission line V Voltage source X Subcircuit call
expression	Any mathematical expression containing values or parameters, i.e., param1 * val2.

elname	Element name that cannot exceed 1023 characters, and must begin with a specific letter for each element type.
M = val	Element multiplier.
mname	Model reference name is required for all elements except passive devices.
node1 …	Node names are identifiers of the nodes to which the element is connected.
pname1 …	Element parameter name used to identify the parameter value that follows this name.
val1…	Value assigned to the parameter pname1 or to the corresponding model node.

See “Element and Source Statements” in the *HSPICE Simulation and Analysis User Guide*.

.END Statement

General Form	.END <comment>
comment	Any comment, normally the name of the data file being terminated.

See “.END Statement” in the *HSPICE Simulation and Analysis User Guide*.

.GLOBAL Statement

General Form	.GLOBAL node1 node2 node3 …

See “.GLOBAL Statement” in the *HSPICE Simulation and Analysis User Guide*.

.IC/.DCVOLT Initial Condition Statement

General Form	.IC *v*(*node1*)=*val* 1 v(*node2*)= + *val* 2 …
Or	.DCVOLT V(*node1*)=*val* 1 + V(*node2*)=*val* 2

See “.IC and .DCVOLT Initial Condition Statements” in the *HSPICE Simulation and Analysis User Guide*.

.IF-.ELSEIF-.ELSE-.ENDIF Statements

You can use this `if-else` structure to change the circuit topology, expand the circuit, set parameter values for each device instance, or select different model cards in each `if-else` block.

.if (*condition1*)
<statement_block1>

{ .elseif (*condition2*)
<statement_block2>
}

[.else (*condition3*)
<statement_block3>
]

.endif

.INCLUDE Statement

General Form	.INCLUDE ‘<filepath> filename’

See “.INCLUDE Statement” in the *HSPICE Simulation and Analysis User Guide*.

.LIB Library Call Statement

General Form	.LIB ‘<filepath> filename’ entryname
entryname	Entry name for the section of the library file to include.
filename	Name of a file to include in the data file.
filepath	Path to a file.

See “.LIB Library Call Statement” in the *HSPICE Simulation and Analysis User Guide*.

.LIB Library File Definition Statement

General Form

```
.LIB entryname1
.
. $ ANY VALID SET OF HSPICE
+ STATEMENTS
.
.ENDL entryname1
.LIB entryname2
.
. $ ANY VALID SET OF HSPICE
+ STATEMENTS
.
.ENDL entryname2
.LIB entryname3
.
. $ ANY VALID SET OF HSPICE
+ STATEMENTS
.
.ENDL entryname3
```

The text following a library file entry name must consist of valid HSPICE statements. See ".LIB Library File Definition Statement" in the *HSPICE Simulation and Analysis User Guide*.

.LIB Nested Library Calls

Library calls may be nested in other libraries provided they call different files. Library calls may be nested to any depth. See ".LIB Nested Library Calls" in the *HSPICE Simulation and Analysis User Guide*.

.MALIAS Statement

You can use the `.MALIAS` statement to assign an alias (another name) to a diode, BJT, JFET, or MOSFET model that you defined in a `.MODEL` statement. See .MALIAS Statement in the *HSPICE Simulation and Analysis User Guide*.

The syntax of the `.MALIAS` statement is:

.MALIAS *model_name*=*alias_name1* <*alias_name2* . . .>

.MODEL Statement

General Form	.MODEL mname type + <VERSION = version_number> + <pname1 = val1 pname2 = val2 …>
VERSION	HSPICE version number, used to allow portability of the BSIM (LEVEL=13), BSIM2 (LEVEL = 39) models between HSPICE releases. Version parameter also valid for LEVEL 49, 53, 54, 57, and 59.
mname	Model name reference.
pname1 …	Parameter name.
type	Selects the model type, which must be one of the following: For HSPICE: AMP operational amplifier model C capacitor model CORE magnetic core model D diode model L magnetic core mutual inductor NJF n-channel JFET model NMOS n-channel MOSFET model NPN npn BJT model OPT optimization model PJF p-channel JFET model PLOT plot model for `.GRAPH` statement PMOS p-channel MOSFET model PNP pnp BJT model R resistor model U lossy transmission line (lumped) W lossy transmission line model S S model SP Frequency table model

See ".MODEL Statement" in the *HSPICE Simulation and Analysis User Guide*.

.NODESET Statement

General Form	.NODESET V(node1) = val1 + <V(node2) = val2 …>
Or	.NODESET node1 val1 <node2 val2>
node1…	Node numbers or node names can include full path names or circuit numbers
val1	Specifies voltage.

See “.NODESET Statement” in the *HSPICE Simulation and Analysis User Guide*.

.PARAM Statement

General Form	.PARAM <ParamName>=<RealNumber>

See “.PARAM Statement” in the *HSPICE Simulation and Analysis User Guide*.

Algebraic Format

General Form	.PARAM <ParamName>=<*AlgebraicExpression*> .PARAM<*ParamName1*>=<*ParamName2*>

Quotes around the algebraic expression are mandatory.

See “Algebraic Parameter (Equation)” in the *HSPICE Simulation and Analysis User Guide*.

Optimization Format

General Form	OPTIMIZE=opt_pav_fun
Or (element or model keyname)	.PARAM <ParamName>=<OptParamFunc> (<Init>, <LoLim>, <Hi Lim>, <Inc>)
paramname 1 …	Parameter names are assigned to values
OptParmFunc	Optimization parameter function (string)
Init	Initial value of parameter (real)
LoLim	Lower limit for parameter (real)
HiLim	Upper limit for parameter (real)
Inc	Rounds to nearest <Inc> value (real)

A parameter can be used in an expression only if it is defined.

.PROTECT Statement

General Form	.PROTECT

The .PROTECT command suppresses the print back of text. See ".PROTECT Statement" in the *HSPICE Simulation and Analysis User Guide*.

.TITLE Statement

General Form	Any string of up to 72 characters
Or	.Title "any string"
Title	The first line of the simulation is always the title.

See "Title of Simulation and .TITLE Statement" in the *HSPICE Simulation and Analysis User Guide*.

.UNPROTECT Statement

General Form	.UNPROTECT

The .UNPROTECT command restores normal output functions from a .PROTECT command. See ".UNPROTECT Statement" in the *HSPICE Simulation and Analysis User Guide*.

.WIDTH Statement

General Form	.WIDTH OUT={80|132}
OUT	The output print width. Permissible values are 80 and 132.

See ".WIDTH Statement" in the *HSPICE Simulation and Analysis User Guide*.

Analyzing Data

You can perform several types of analysis with HSPICE.

DC Analysis

HSPICE can perform the following types of DC analysis.

.DC Statement—DC Sweep

See “.DC Statement—DC Sweeps” in the *HSPICE Simulation and Analysis User Guide*.

Sweep or Parameterized Sweep

General Form	.DC *var1 start1 stop1 incr1* + <SWEEP *var2 type np* + *start2 stop2*>
Or	.DC *var1 start1 stop1 incr1* + <*var2 start2 stop2 incr2*>

Data-Driven Sweep

General Form	.DC *var1 type np start1 stop1* + <SWEEP DATA = *datanm*>
Or	.DC DATA = *datanm* + <SWEEP *var2 start2 stop2* + *incr2*>
Or	.DC DATA = datanm

Monte Carlo

General Form	.DC *var1 type np start1 stop1* + <SWEEP MONTE = *val*>
Or	.DC MONTE = *val*

Optimization

General Form	.DC DATA = *datanm* OPTIMIZE = + *opt_par_fun* RESULTS = + *measnames* MODEL = *optmod*
Or	.DC var1 start1 stop1 SWEEP + OPTIMIZE = OPT*xxx* + RESULTS = *measname* + MODEL = *optmod*

DC analysis statement	.DC <DATA=*filename*> SWEEP + OPTIMIZE=OPT*xxx* + RESULTS=*ierr1* ... *ierrn* + MODEL=*optmod*
DATA=datanm	*Datanm* is the reference name of a `.DATA` statement.
incr1 …	Voltage, current, element, model parameters, or temperature increment values.
MODEL	Optimization reference name, used in the `.MODEL OPT` statement.
MONTE=val	Produces a number (*val*) of randomly generated values, which select parameters from a distribution.
np	Number of points per decade (or depending on the preceding keyword).
OPTIMIZE	Specifies the parameter reference name used in the `.PARAM` statement.
RESULTS	Specifies the measure name used in the `.MEASURE` statement.
start1 …	Starting voltage, current, element, model parameters, or temperature values.
stop1 …	Final voltage, current, any element, model parameter, or temperature values.
SWEEP	Indicates that a second sweep has a different variation type (`DEC`, `OCT`, `LIN`, `POI`, `DATA` statement, or `MONTE = val`).
TEMP	Indicates a temperature sweep.
type	Can be any of the following keywords: `DEC`, `OCT`, `LIN`, `POI`.
var1 …	Name of an independent voltage or current source, any element or model parameter, or the `TEMP` keyword.

.OP Statement—Operating Point

General Form	.OP *<format> <time> <format> <time>*
format	Any of the following keywords: `ALL`, `BRIEF`, `CURRENT`, `DEBUG`, `NONE`, `VOLTAGE`.
time	Parameter after `ALL`, `VOLTAGE`, `CURRENT`, or `DEBUG` to specify the time at which the report is printed.

See “.OP Statement — Operating Point” in the *HSPICE Simulation and Analysis User Guide*.

.PZ Statement—Pole/Zero Analysis

General Form	`.PZ ov srcnam`
ov	Output variable: a node voltage `V(`*`n`*`)`, or branch current `I(`*`element`*`)`
srcnam	Input source: an independent voltage or current source name

See “.PZ Statement— Pole/Zero Analysis” in the *HSPICE Simulation and Analysis User Guide*.

.SENS Statement—DC Sensitivity Analysis

General Form	.SENS *ov1 <ov2* ...>
ov1 ov2 …	Branch currents, or nodal voltage, for DC component sensitivity analysis.

See “.SENS Statement — DC Sensitivity Analysis” in the *HSPICE Simulation and Analysis User Guide*.

.TF Statement—DC Small-Signal Transfer Function Analysis

General Form	`.TF` *`ov srcnam`*
ov	Small-signal output variable
srcnam	Small-signal input source

See “.TF Statement — DC Small-Signal Transfer Function Analysis” in the *HSPICE Simulation and Analysis User Guide*.

AC Analysis

.AC Statement

Single/Double Sweep

General Form	.AC *type np fstart fstop*
Or	.AC *type np fstart fstop* + <SWEEP *var* <*START=>start* + <STOP=>*stop* <STEP=>*incr*>
Or	.AC *type np fstart fstop* <SWEEP *var* *type np start stop*>
Or	.AC *type np fstart fstop* + <SWEEP *var* + START="*param_expr1*" + STOP="*param_expr2*" + STEP="*param_expr3*">
Or	.AC *type np fstart fstop* + <SWEEP *var start_expr* + *stop_expr step_expr*>

See “.AC Statement” in the *HSPICE Simulation and Analysis User Guide*.

Parameterized Sweep

General Form	.AC *type np fstart fstop* <SWEEP DATA = *datanm*>
Or	.AC DATA = *datanm*
Or	.AC DATA = *datanm* <SWEEP *var* + <START=>start <STOP=>stop + <STEP=>*incr*>
Or	.AC DATA = *datanm* <SWEEP *var* + *type np start stop*>
Or	.AC DATA = *datanm* <SWEEP *var* + START="*param_expr1*" + STOP="*param_expr2*" + STEP="*param_expr3*">
Or	.AC DATA = *datanm* <SWEEP *var* + *start_expr stop_expr* + *step_expr*>

Optimization

General Form	.AC DATA = *datanm* + OPTIMIZE = *opt_par_fun* + RESULTS = *measnames* + MODEL = *optmod*
AC analysis statement	.AC <DATA=*filename*> SWEEP + OPTIMIZE=OPT*xxx* + RESULTS=*ierr1* ... *ierrn* + MODEL=*optmod*

Random/Monte Carlo

General Form	.AC *type np fstart fstop* + <SWEEP MONTE = *val*>
DATA=*datanm*	Data name referenced in the `.AC` statement.
fstart	Starting frequency. If you use `POI` (list of points) type variation, use a list of frequency values, not `fstart fstop`.
fstop	Final frequency.
incr	Increment value of the voltage, current, element, or model parameter. If you use *type* variation, specify the *np* (number of points) instead of *incr*.
MONTE = *val*	Produces a number (*val*) of randomly-generated values. HSPICE uses these values to select parameters from a distribution, either *Gaussian*, *Uniform*, or *Random Limit*.
np	Number of points, or points per decade or octave, depending on which keyword precedes it.
start	Starting voltage or current, or any parameter value for an element or a model.
stop	Final voltage or current, or any parameter value for an element or a model.
SWEEP	This keyword indicates that the `.AC` statement specifies a second sweep.
TEMP	This keyword indicates a temperature sweep
type	Can be any of the following keywords: `DEC` – decade variation. `OCT` – octave variation. `LIN` – linear variation. `POI` – list of points.

var	Name of an independent voltage or current source, element or model parameter, or the `TEMP` (temperature sweep) keyword.

.DISTO Statement—AC Small-Signal Distortion Analysis

General Form	`.DISTO` *`Rload <inter <skw2`* `+` *`<refpwr <spwf>>>>`*
inter	Interval at which HSPICE prints a distortion-measure summary.
refpwr	Reference power level, used to compute the distortion products.
Rload	Element name of the output load resistor, into which the output power feeds.
skw2	Ratio of the second frequency (F2) to the nominal analysis frequency (F1).
spwf	Amplitude of the second frequency (F2).

See “.DISTO Statement — AC Small-Signal Distortion Analysis” in the *HSPICE Simulation and Analysis User Guide*.

.NOISE Statement—AC Noise Analysis

General Form	.NOISE ovv srcnam inter
inter	Interval at which HSPICE prints a noise analysis summary; inter specifies how many frequency points to summarize in the AC sweep.
ovv	Nodal voltage output variable, defining the node at which HSPICE sums the noise.
srcnam	Name of the independent voltage or current source, to use as the noise input reference.

See “.NOISE Statement — AC Noise Analysis” in the *HSPICE Simulation and Analysis User Guide*.

.SAMPLE Statement—Noise Folding Analysis

General Form	.SAMPLE FS = freq <TOL = val> + <NUMF = val> <MAXFLD = val> + <BETA = val>
BETA	Integrator duty cycle; specifies an optional noise integrator at the sampling node.

FS = freq	Sample frequency, in Hertz.
MAXFLD	Maximum number of folding intervals.
NUMF	Maximum allowed number of frequencies that you can specify.
TOL	Sampling error tolerance.

See “.SAMPLE Statement — Noise Folding Analysis” in the *HSPICE Simulation and Analysis User Guide*.

Small-Signal Network Analysis

.NET Statement—AC Network Analysis

One-port network

General Form	`.NET` *`input`* `<RIN =` *`val`*`>`
Or	.NET *input* <*val*>

Two-port network

General Form	.NET *output input* + <ROUT = *val*> <RIN = *val*>
input	Name of the voltage or current source for AC input.
output	Output port. It can be: An output voltage, `V(`*`n1,n2`*`)`. An output current, `I(`*`source`*`)`, or `I(`*`element`*`)`.
RIN	Keyword, for input or source resistance. The `RIN` value calculates output impedance, output admittance, and scattering parameters. The default `RIN` value is `1` ohm.
ROUT	Keyword, for output or load resistance. The `ROUT` value calculates input impedance, admittance, and scattering parameters. The default `ROUT` value is `1` ohm.

See “.NET Statement - AC Network Analysis” in the *HSPICE Simulation and Analysis User Guide*.

AC Network Analysis—Output Specification

General Form	Xij(z), ZIN(z), ZOUT(z), YIN(z), YOUT(z)
ij	Identifies which matrix parameter to print.
X	Specifies Z for impedance, Y for admittance, H for hybrid, or S for scattering.
YIN	Input admittance.
YOUT	Output admittance.
z	Output type: R, I, M, P, DB, T.
ZIN	Input impedance.
ZOUT	Output impedance.

See “AC Network Analysis - Output Specification” in the *HSPICE Simulation and Analysis User Guide*.

Temperature Analysis

.TEMP Statement

General Form	.TEMP t1 <t2 <t3 ...>>
t1 t2 …	Temperatures, in °C, at which HSPICE simulates the circuit.

See “.TEMP Statement” in the *HSPICE Simulation and Analysis User Guide*.

Transient Analysis

.TRAN Statement

See “Using the .TRAN Statement” in the *HSPICE Simulation and Analysis User Guide*.

Single-Point Analysis

```
.TRAN tincr1 tstop1 <tincr2
      tstop2 ...tincrN tstopN>
+ <START = val> <UIC>
```

Double-Point Analysis

```
.TRAN tincr1 tstop1 <tincr2
    tstop2 ...tincrN tstopN>
+ <START = val> <UIC>
+ <SWEEP var type np pstart pstop>
```

or

```
.TRAN tincr1 tstop1 <tincr2
    tstop2 ...tincrN tstopN>
+ <START = val> <UIC> <SWEEP var
+ START="param_expr1" STOP="param_expr2"
+ STEP="param_expr3">
```

or

```
.TRAN tincr1 tstop1 <tincr2 tstop2 ...
    tincrN tstopN>
+ <START=val> <UIC> <SWEEP var start_expr
+ stop_expr step_expr>
```

Data-Driven Sweep

General Form (data-driven sweep)	.TRAN DATA = datanm
Or	TRAN tincr1 tstop1 <tincr2 tstop2...tincrN + tstopN> <START = val> <UIC> + <SWEEP DATA = datanm>
Or	.TRAN DATA = datanm <SWEEP var + <START=>pstart <STOP=>pstop + <STEP=>pincr>
Or	.TRAN DATA = datanm <SWEEP var + type np pstart pstop>
Or	.TRAN DATA = datanm <SWEEP var + START="param_expr1" + STOP="param_expr2" + STEP="param_expr3">
Or	.TRAN DATA = datanm <SWEEP var + start_expr stop_expr step_expr>

Monte Carlo Analysis

General Form	.TRAN tincr1 tstop1 <tincr2 tstop2 + ...tincrN tstopN> + <START = val> + <UIC><SWEEP MONTE = val> >

Optimization

General Form	.TRAN DATA = datanm OPTIMIZE = + opt_par_fun RESULTS = measnames + MODEL = optmod
TRAN analysis statement	.TRAN <DATA=*filename*> SWEEP + OPTIMIZE=OPT*xxx* + RESULTS=*ierr1* ... *ierrn* + MODEL=*optmod*
DATA = datanm	Data name referenced in the .TRAN statement.
MONTE = val	Produces a number *val* of randomly-generated values used to select parameters from a distribution.
np	Number of points per decade (or depending on the preceding keyword).
param_expr...	User-specified expressions.
pincr	Voltage, current, element, or model parameter, or temperature increment value.
pstart	Starting voltage, current, temperature, any element, or model parameter value.
pstop	Final voltage, current, temperature, any element, or model parameter value.
START	Time at which printing/plotting begins.
SWEEP	Indicates a second sweep is specified on the .TRAN statement.
tincr1…	Printing/plotting increment for printer output, and the suggested computing increment for the postprocessor.
tstop1…	Time at which the transient analysis stops incrementing by tincr1.
type	Specifies any of the following keywords: DEC, OCT, LIN, POI.
UIC	Causes HSPICE to use the nodal voltages specified in the .IC statement (or by the “IC = ” parameters in the various element statements) to calculate the initial transient conditions, rather than solving for the quiescent operating point.

var	Name of an independent voltage or current source, any element or model parameter, or the keyword TEMP.

.BIASCHK Statement

General Form	.biaschk *type* terminal1=*t1* + terminal2=*t2* limit=*lim* + <noise=*ns*><name=*devname1*> + <name=*devname2*>... + <mname=*modelname1*> + <mname=*modelname2*> ...
type	Element type to check.
terminal 1, terminal2	Terminals, between which HSPICE checks (checks between terminal1 and terminal2)
limit	Biaschk limit that you define.
noise	Biaschk noise that you define. The default is 0.1v.
name	Element name to check.
mname	Model name. HSPICE checks elements of this model, for bias.

You can use a wild card, to describe *name* and *mname*, in the `biaschk` card.

? stands for one character.

 * stands for 0 or more characters.

Options for the .biaschk Command

Output file defined option:

General Form	.option biasfile=biaschk/mos.bias

Warning message turn off (on) option:

General Form (on)	.option biawarn=1
General Form (off, default)	.option biawarn=0

Numerical Integration Algorithm Controls

See “Numerical Integration Algorithm Controls (HSPICE)” in the *HSPICE Simulation and Analysis User Guide*.

Gear Algorithm

General Form	.OPTION METHOD=GEAR

Backward-Euler

General Form	.OPTION METHOD=GEAR MU = 0

Trapezoidal Algorithm

General Form	.OPTION METHOD=TRAP

FFT Analysis

.FFT Statement

General Form	.FFT *output_var* <START = *value*> + <STOP = *value*> <NP = *value*> + <FORMAT = *keyword*> + <WINDOW = *keyword*> + <ALFA = *value*> <FREQ = *value*> + <FMIN = *value*> <FMAX = *value*>
ALFA	Parameter used in GAUSS and KAISER windows to control the highest side-lobe LEVEL, bandwidth, and so on.
FMAX	Maximum frequency for which HSPICE prints FFT output into the listing file. THD calculations also use this frequency.
FMIN	Minimum frequency for which HSPICE prints FFT output into the listing file. THD calculations also use this frequency.
FORMAT	Output format. NORM= normalized magnitude UNORM=unnormalized magnitude
FREQ	Frequency to analyze.
FROM	An alias for START.
NP	Number of points to use in FFT analysis.

output_var	Any valid output variable, such as voltage, current, or power.
START	Beginning of the output variable waveform to analyze.
STOP	End of the output variable waveform to analyze.
TO	An alias for STOP.
WINDOW	Window type to use: RECT, BART, HANN, HAMM, BLACK, HARRIS, GAUSS, KAISER.

See ".FFT Statement" in the *HSPICE Applications Manual*.

Worst Case Analysis

See "Worst Case Analysis" in the *Simulation and Analysis User Guide*.

Sigma Deviations

Type	Param	Slow	Fast
NMOS	XL	+	-
	RSH	+	-
	DELVTO	+	-
	TOX	+	-
	XW	-	+
PMOS	XL	+	-
	RSH	+	-
	DELVTO	-	+
	TOX	+	-
	XW	-	+

Monte Carlo Analysis

HSPICE statements needed to set up a Monte Carlo analysis are:

- .PARAM statement.
- .DC, .AC, or .TRAN analysis—enable MONTE.
- .MEASURE statement.

See “Monte Carlo Analysis” in the *HSPICE Simulation and Analysis User Guide*. For details about the syntax for these statements, see “Analysis Syntax” in the *HSPICE Simulation and Analysis User Guide*.

Operating Point

General Form	.DC MONTE=val

DC Sweep

General Form	.DC vin 1 5 .25 SWEEP MONTE=val

AC Sweep

General Form	.AC dec 10 100 10meg SWEEP + MONTE=val

TRAN Sweep

General Form	.TRAN 1n 10n SWEEP MONTE=val

.PARAM Distribution Function Syntax

General Form	.PARAM xx=UNIF(nominal_val, + rel_variation <, multiplier>)
Or	.PARAM xx=AUNIF(nominal_val, + abs_variation <,multiplier>)
Or	.PARAM xx=GAUSS(nominal_val, + rel_variation, sigma <,multiplier>)
Or	.PARAM xx=AGAUSS(nominal_val, + abs_variation, sigma <,multiplier>)
Or	.PARAM xx=LIMIT(nominal_val, + abs_variation)
abs_variation	AUNIF and AGAUSS vary the nominal_val by +/- abs_variation.
AGAUSS	Gaussian distribution function, using absolute variation.
AUNIF	Uniform distribution function, using absolute variation.
GAUSS	Gaussian distribution function, using relative variation.
LIMIT	Random limit distribution function, using absolute variation.

multiplier	If you do not specify a multiplier, the default is 1.
nominal_val	Nominal value for Monte Carlo analysis, and default value for all other analyses.
rel_variation	UNIF and GAUSS vary the nominal_val, by +/- (nominal_val · rel_variation).
sigma	Specifies *abs_variation* or *rel_variation* at the *sigma* level.
UNIF	Uniform distribution function, using relative variation.
xx	Distribution function calculates the value of this parameter.

Optimizing Data

This chapter briefly describes how to optimize your design data.

Analysis Statement (.DC, .TRAN, .AC) Syntax

General Form	.DC <DATA=*filename*> SWEEP + OPTIMIZE=OPT*xxx* + RESULTS=*ierr1* ... + ierr*n* MODEL=*optmod*
DATA	In-line file of parameter data to use in the optimization.
MODEL	The optimization reference name (also specified in the `.MODEL` optimization statement).
OPTIMIZE	Indicates the analysis is for optimization.
Or	.AC <DATA=filename> SWEEP + OPTIMIZE=OPTxxx + RESULTS=ierr1 ... + ierr*n* MODEL=optmod
Or	.TRAN <DATA=filename> SWEEP + OPTIMIZE=OPTxxx + RESULTS=ierr1 ... + ierr*n* MODEL=optmod
RESULTS	The measurement reference name (also specified in the `.MEASURE` optimization statement).

See “Analysis Statement (.DC, .TRAN, .AC)” in the *HSPICE Simulation and Analysis User Guide*.

.PARAM Statement Syntax

General Form	.PARAM parameter=OPTxxx + (initial_guess, low_limit, upper_limit)
Or	.PARAM parameter=OPTxxx + (initial_guess, low_limit, upper_limit, + delta)
delta	The final parameter value is the initial guess $\pm$ (*n*·delta).

OPTxxx	Optimization parameter reference name. The associated optimization analysis references this name.
parameter	Parameter to be varied, the initial value estimate, the lower limit, and the upper limit allowed for the parameter.

See ".PARAM Statement" in the *HSPICE Simulation and Analysis User Guide*.

.MODEL Statement Syntax

General Form	.MODEL *mname* OPT <parameter = val + ...>
CENDIF	Point at which more accurate derivatives are required.
CLOSE	Initial estimate of how close parameter initial value estimates are to final solution.
CUT	Modifies CLOSE, depending on how successful the iterations toward the solution become.
DIFSIZ	Determines the increment change in a parameter value for gradient calculations ($\Delta x = DIFSIZ \cdot max(x,$ PARMIN)).
GRAD	Possible convergence, when gradient of RESULTS function is less than GRAD.
ITROPT	Sets the maximum number of iterations.
LEVEL	Selects an optimizing algorithm.
MAX	Sets the upper limit on CLOSE.
mname	Model name.
PARMIN	Allows better control of incremental parameter changes during error calculations.
RELIN	Relative input parameter variation for convergence.
RELOUT	Relative output RESULTS function variance for convergence.

See ".MODEL Statement" in the *HSPICE Simulation and Analysis User Guide*.

Filters and Systems

To optimize filters and systems, use Pole Zero analysis. See “.PZ Statement— Pole/Zero Analysis” in the *HSPICE User Guide*.

Laplace Transforms

See “Laplace Transform (LAPLACE) Function” and “Laplace Transform” in the *HSPICE User Guide*.

Transconductance H(s)

General Form	Gxxx n_+ n_- LAPLACE in_+ in_- k_0, k_1, ..., k_n + / d_0, d_1, ..., d_m <SCALE=val> <TC1=val> + <TC2=val> <M=val>

Voltage Gain H(s)

General Form	Exxx n_+ n_- LAPLACE in_+ in_- k_0, k_1, ..., k_n + / d_0, d_1, ..., d_m <SCALE=val> <TC1=val> + <TC2=val>

Output Format

For a detailed description of graphing with HSPLOT and GSI, see the *HSPICE Simulation and Analysis User Guide* “Graphing.”

Graphing Results in AvanWaves

The .option POST must be placed in the HSPICE netlist input file.

- POST or POST=1 creates a binary file.
- POST=2 creates an ascii file, portable to all supported machines.

Limiting the Size of the Graph Data File

The option PROBE limits the number of curves stored to those nodes specified in the HSPICE input file’s .PRINT, .PLOT, .OPTION PROBE, and .GRAPH statements. The option INTERP (for transient analysis only) limits the number of points stored. The option INTERP preinterpolates the output to the interval specified on the .TRAN statement.

Automatic Hardcopy During HSPICE Run

A .GRAPH statement will automatically produce a hardcopy plot. A .TITLE placed before each .GRAPH will set the graph title. Otherwise, the simulation title will be used. The option POST in conjunction with .GRAPH will create a graph data file.

Starting AvanWaves—Command line

AvanWaves’ command line definition is:

```
awaves [[-d] <path><design-name> [-c
    + <config_name>]
    [laf(windows|openlook|motif)]
```

-d	The name of the design to be opened on invoking AvanWaves
-c	Specifies that a previously saved configuration for the current design is to be used upon the initialization of AvanWaves.
-laf [windows| openlook| motif]	Specifies the window manager style to be used. The default is Motif.

Setup Commands

Cmd	Default	Description
I	--	Name input file.
XMIN, XMAX, YMIN, YMAX	X=LIM Y=AUTO default 0.0	Set range defaults for all panels.
XSCAL	1.0	Scale for X axis.
YSCAL	1.0	Scale for Y axis.
XS, YS	LIN	Set x or y scale.
P	1	Set number of panels.
F	NONE	Set the frequency of symbols.
T	ON	Set/Toggle ticks.
M	NO	Monotonic. Set/Toggle for family of curves.
XG, YG	ON	Set/Toggle x or y grids.
D	--	Reinitialize all Setup menu values.

Accessible Menus From Setup

G	Bring up the Graph window.
N	Bring up the Node window.
Q	Exit the program.

Node Menu Prompts

-Panel	Each panel prompts for one x-axis parameter and any number of y-axis curves.
-X-axis	Any node may be chosen as the x-axis for a panel.
-Y-axis	Any listed node name or function, or algebraic expression can be entered at the y-axis prompt.

Node Menu Commands

$P	Remove all curves in present panel.
$A	Remove all curves from all panels.
$Q	Exit the program.
MORE	Display next/previous page of nodes.

/BACK	These commands appear only when the node list spans more than one page.
$S	Bring up the Setup menu.

AC Analysis

*R	Draw the Real component of the data.
*I	Draw the Imaginary component of the data.
*M	Calculate and draw the Magnitude.
*P	Calculate and draw the Phase.

Graph Commands

A, D	Add or Delete curves or expressions.
X, Y	Change the view on some panels or all panels.
Q	Exit the program.

Accessible Menus from Graph Menu

N	Bring up the Node window
P	Bring up the Print menu
S	Bring up the Setup menu

Print Menu

The Print menu lists printers and /or plotters at your site on which you may create a hardcopy plot.

Screensave Option

The SCREENSAVE function produces a file that can later be displayed on the terminal. The function is useful for making video slides.

Print Commands

<CR>	Print with the default printer.
1…n-1	Print with one of printer options.
n	Save the screen into a preview file.

.PRINT Statement

General Form	.PRINT antype ov1 <ov2 ... ov*n*>

See “.PRINT Statement” in the *HSPICE Simulation and Analysis User Guide*.

.PLOT Statement

General Form	.PLOT antype ov1 <(plo1,phi1)> . . . + <ov*n*> <(ploon,phin)>

See “.PLOT Statement” in the *HSPICE Simulation and Analysis User Guide*.

.PROBE Statement

General Form	.PROBE antype ov1 … <ov*n*>

See “.PROBE Statement” in the *HSPICE Simulation and Analysis User Guide*.

.GRAPH Statement

General Form	.GRAPH antype <MODEL = mname> + <unam1 = > ov1, <unam2 = >ov2, … + <unam*n* = > ov*n* (plo,phi)
antype	Type of analysis for outputs: DC, AC, TRAN, NOISE, or DISTO.
mname	Plot model name referenced in .GRAPH.
ov1 …ov*n*	Output variables to print or plot.
plo, phi …	Lower and upper plot limits.
unam1…	User-defined output names.

See “.GRAPH Statement” in the *HSPICE Simulation and Analysis User Guide*.

.MODEL Statement for .GRAPH

General Form	.MODEL mname PLOT (pnam1 = val1 + pnam2 = val2….)
mname	Plot model name referenced in .GRAPH statement.
PLOT	Keyword for a .GRAPH statement model.
pnam1=val1…	Each .GRAPH statement model includes several model parameters.

See “.MODEL Statement for .GRAPH” in the *HSPICE Simulation and Analysis User Guide*.

.MEASURE Statement: Rise, Fall, and Delay

General Form	.MEASURE <DC|AC|TRAN> result + TRIG … TARG … <GOAL=val> + <MINVAL=val> <WEIGHT=val>
<DC|AC|TRAN>	Analysis type of the measurement. If omitted, assumes the last analysis mode requested.
GOAL	Desired measure value in optimization.
MEASURE	Specifies measurements.
MINVAL	If the absolute value of GOAL is less than MINVAL, then MINVAL replaces the GOAL value in the denominator of the ERRfun expression.
TRIG…, TARG …	Identifies the beginning of trigger and target specifications, respectively.
WEIGHT	The calculated error is multiplied by the weight value.

See “.MEASURE Statement: Rise, Fall, and Delay” in the *HSPICE Simulation and Analysis User Guide*.

Trigger

General Form	TRIG trig_var VAL=trig_val + <TD=time_delay> <CROSS=c> + <RISE=r> <FALL=f>
Or	TRIG AT=val
result	Name associated with the measured value in the HSPICE output.

Target

General Form	TARG targ_var VAL = targ_val + <TD = time_delay> <CROSS = c | LAST> + <RISE = r | LAST> <FALL = f | LAST>
AT = val	Special case for trigger specification.
CROSS = c RISE = r FALL = f	Numbers indicate which occurrence of a CROSS, FALL, or RISE event to measure.
LAST	HSPICE measures when the last CROSS, FALL, or RISE event occurs.

TARG	Beginning of the target signal specification.
targ_val	Value of the *targ_var*, which increments the counter for crossings, rises, or falls, by one.
targ_var	Name of the output variable, at which HSPICE determines the propagation delay with respect to the *trig_var*.
time_delay	Amount of simulation time that must elapse, before HSPICE enables the measurement.
TRIG	Beginning of the trigger specification.
trig_val	Value of *trig_var* at which the counter for crossing, rises, or falls increments by one.
trig_var	Name of the output variable, that determines the logical beginning of a measurement.

Average, RMS, MIN, MAX, and Peak to Peak

General Form	.MEASURE <DC|AC|TRAN> + *result func out_var* + <FROM = *val*> <TO = *val*> + <GOAL = *val*> + <MINVAL = *val*> <WEIGHT = *val*>
or	.MEASURE < TRAN > *out_var* + func *var* FROM = *start* + TO = *end*
<DC|AC|TRAN>	Analysis type of the measurement. If omitted, HSPICE assumes the last analysis mode requested.
FROM	Initial value for the “func” calculation.
func	Type of the measure statement: AVG (average) MAX (maximum) MIN (minimum) PP (peak-to-peak) RMS (root mean squared) INTEG (integral)
GOAL	Desired .MEASURE value.
MINVAL	If the absolute value of GOAL is less than MINVAL, then MINVAL replaces the GOAL value in the denominator of the ERRfun expression.

out_var	Name of any output variable whose function the simulation measures.
result	Name of the measured value in the HSPICE output.
TO	End of the “func” calculation.
WEIGHT	Multiplies the calculated error, by the weight value.
start	Starting time of the measurement period.
end	Ending time of the measurement period.

Equation Evaluation

General Form	.MEASURE <DC|TRAN|AC> result + PARAM = ‘equation’ <GOAL = val> + <MINVAL = val>

See “Equation Evaluation” in the *HSPICE User Guide*.

ERROR Function

General Form	.MEASURE <DC|AC|TRAN> result + ERRfun meas_var calc_var + <MINVAL = val> < IGNORE | + YMIN = val> <YMAX = val> + <WEIGHT = val> <FROM = val> + <TO = val>

<DC|AC|TRAN>	Analysis type of the measurement. If omitted, assumes the last analysis mode requested.
calc_var	Name of the simulated output variable or parameter in the .MEASURE statement to compare with *meas_var*.
ERRfun	ERRfun indicates which error function to use: ERR, ERR1, ERR2, or ERR3.
FROM	Beginning of the ERRfun calculation.
IGNOR|YMIN	If the absolute value of *meas_var* is less than the IGNOR value, the ERRfun calculation does not consider this point.
meas_var	Name of any output variable or parameter in the data statement.
MINVAL	If the absolute value of *meas_var* is less than MINVAL, then MINVAL replaces the *meas_var* value in the denominator of the ERRfun expression.
result	Name of measured result in the output.
TO	End of the ERRfun calculation.

WEIGHT
Multiplies the calculated error by the weight value.

YMAX
If the absolute value of *meas_var* is greater than the YMAX value, then the ERRfun calculation does not consider this point.

Find and When Functions

General Form

```
.MEASURE <DC|TRAN| AC> result
+ WHEN out_var = val <TD = val>
+ < RISE = r | LAST > <FALL = f |
+ LAST > <CROSS = c | LAST >
+ <GOAL = val> <MINVAL = val>
+ <WEIGHT = val>
```

Or

```
.MEASURE <DC|TRAN|AC> result
+ WHEN out_var1 = out_var2
+ < TD = val > < RISE = r | LAST >
+ <FALL = f | LAST > < CROSS = c|
+ LAST > <GOAL = val>
+ <MINVAL = val> <WEIGHT = val>
```

Or

```
.MEASURE <DC|TRAN|AC> result
+ FIND out_var1 WHEN out_var2 = val
+ < TD = val > < RISE = r | LAST >
+ <FALL = f | LAST > < CROSS = c|
+ LAST > <GOAL = val>
+ <MINVAL = val> <WEIGHT = val>
```

Or

```
.MEASURE <DC|TRAN|AC> result
+ FIND out_var1 WHEN
+ out_var2 = out_var3 <TD = val >
+ < RISE = r | LAST > <FALL = f |
+ LAST ><CROSS = c | LAST>
+ <GOAL = val> <MINVAL = val>
+ <WEIGHT = val>
```

Or

```
.MEASURE <DC|TRAN|AC> result
+ FIND out_var1 AT = val
+ <GOAL = val> <MINVAL = val>
+ <WEIGHT = val>
```

<DC|AC|TRAN>
Analysis type for the measurement. If omitted, HSPICE assume the last analysis type requested.

CROSS = c
RISE = r
FALL = f
Numbers indicate which occurrence of a CROSS, FALL, or RISE event starts measuring.

FIND
Selects the FIND function.

GOAL
Desired .MEASURE value.

LAST
Starts measurement at the last CROSS, FALL, or RISE event.

MINVAL	If the absolute value of GOAL is less than MINVAL, then MINVAL replaces GOAL value in ERRfun expression denominator.
out_var(1,2,3)	Establish conditions to start measuring.
result	Name associated with a measured value in HSPICE output.
TD	Time at which measurement starts.
WEIGHT	Multiplies calculated error by weight value.
WHEN	Selects the WHEN function.

.DOUT Statement

```
.DOUT nd VTH ( time state
< time state > )
```

where:

- *nd* is the node name.
- *VTH* is the single voltage threshold.
- *time* is an absolute time-point.
- *state* is one of the following expected conditions of the *nd* node at the specified *time*:
  - 0 expect ZERO,LOW.
  - 1 expect ONE,HIGH.
  - else Don’t care.

```
.DOUT nd VLO VHI ( time state
< time state > )
```

where:

- *nd* is the node name.
- *VLO* is the voltage of the logic low state.
- *VHI* is the voltage of the logic high state.
- *time* is an absolute time-point.
- *state* is one of the following expected conditions of the *nd* node at the specified *time*:
  - 0 expect ZERO,LOW.
  - 1 expect ONE,HIGH.
  - else Don’t care.

See “.DOUT Statement: Expected State of Digital Output Signal” in the *HSPICE Simulation and Analysis User Guide.*

.STIM Statement

You can use the `.STIM` statement to reuse the results (output) of one simulation, as input stimuli in a new simulation.

The `.STIM` statement specifies:

- Expected stimulus (PWL Source, DATA CARD, or VEC FILE).
- Signals to transform.
- Independent variables.

One `.STIM` statement produces one corresponding output file.

Syntax

Brackets [ ] enclose comments, which are optional.

```
.stim <tran|ac|dc> PWL|DATA|VEC
<filename=output_filename> ...
```

DC and Transient Output

See “DC and Transient Output Variables” in the *HSPICE Simulation and Analysis User Guide*.

Nodal Voltage

General Form	V (n1<,n2>)
n1, n2	Defines nodes between which the voltage difference (n1-n2) is to be printed/plotted.

See “Nodal Voltage Syntax” in the *HSPICE Simulation and Analysis User Guide*.

Current: Voltage Sources

General Form	I (Vxxx)
Vxxx	Voltage source element name.

See “Current: Voltage Sources” in the *HSPICE Simulation and Analysis User Guide*.

Current: Element Branches

General Form	In (Wwww)
or	Iall (Www)
n	Node position number, in the element statement.
Wwww	Element name.
Iall (Www)	An alias just for diode, BJT, JFET, and MOSFET devices.

See “Current: Element Branches” in the *HSPICE User Guide*.

Power Output

See “Power Output” in the *HSPICE Simulation and Analysis User Guide*.

Print/Plot Power

General Form	.PRINT <DC|TRAN> P(element_or_subcircuit_name) POWER
Or	.PLOT <DC|TRAN> P(element_or_subcircuit_name) POWER
antype	Type of analysis for the specified plots: DC, AC, TRAN, NOISE, or DISTO.
ov1 …	Output variables to plot.
plo1,phi1 …	Lower and upper plot limits.

Power calculation is associated only with transient and DC sweep analyses. The .MEASURE statement may be used to compute the average, rms, minimum, maximum, and peak to peak value of the power. POWER invokes the total power dissipation output. See “Print or Plot Power” in the *HSPICE User Guide*.

AC Analysis Output

See “AC Analysis Output Variables” in the *HSPICE Simulation and Analysis User Guide*.

Nodal Voltage

General Form	Vz (n1,<,n2>)
z	Voltage output type.
DB	Decibel
I	Imaginary Part
M	Magnitude
P	Phase
R	Real Part
T	Group Delay
n1, n2	Node names. If you omit n2, HSPICE assumes ground (node 0).

See “Nodal Voltage” in the *HSPICE Simulation and Analysis User Guide*.

Current: Independent Voltage Sources

General Form	Iz (Vxxx)
Vxxx	Voltage source element name. If an independent power supply is within a subcircuit, then to access its current output, append a dot and the subcircuit name to the element name.
z	Current output type. See Nodal Voltage in Chapter 8 of the *HSPICE User Guide* for specific output types.

See “Current: Independent Voltage Sources” in the *HSPICE Simulation and Analysis User Guide*.

Current: Element Branches

General Form	Izn (Wwww)
n	Node position number in element statement.
Wwww	Element name. If the element is within a subcircuit, then to access its current output, append a dot and the subcircuit name to the element name.
z	Current output type. See Nodal Voltage in Chapter 8 of the *HSPICE User Guide* for specific output types.

See “Current: Element Branches” in the *HSPICE Simulation and Analysis User Guide*.

Group Time Delay t

General Form	VT (n1, <n2>) or IT(Vxxx) or ITn(Wwww)
n1, n2	Node names. If you omit n2, HSPICE assumes grough (node 0).
Vxxx	Independent voltage source element name.
n	Node position number in element statement.
Wwww	Element name

Since there is discontinuity in phase each 360 degrees, the same discontinuity occurs in the Td, even though Td is continuous.

See “Group Time Delay” in the *HSPICE User Guide*.

Network Output

General Form	Xij (z), ZIN(z), ZOUT(z), YIN(z), YOUT(z)
ij	i and j can be 1 or 2. They identify the matrix parameter to print.
X	Specifies Z for impedance, Y for admittance, H for hybrid, or S for scattering parameters.
YIN	Input admittance.
YOUT	Output admittance.
z	Output type. If you omit z, HSPICE prints the magnitude of the output variable.
ZIN	Input impedance. For a one-port network, `ZIN`, `Z11`, and `H11` are the same.
ZOUT	Output impedance.

See “Network” in the *HSPICE Simulation and Analysis User Guide*.

Noise and Distortion

General Form	ovar <(z)>

See “Nodal Voltage” on page 80 for specific output types.

ovar	Noise and distortion analysis parameter.
z	Output type (only for distortion).

See “Noise and Distortion” in the *HSPICE Simulation and Analysis User Guide*.

Element Template Output

Use for DC, AC, or Transient Analysis.

General Form	Elname:Property
Elname	Name of the element.
Property	Property name of an element, such as a user-input parameter, state variable, stored charge, capacitance current, capacitance, or derivative of a variable.

See “Element Template Output” in the *HSPICE Simulation and Analysis User Guide*.

Element Template Listings

Resistor

Name	Alias	Description
G	LV1	Conductance at analysis temperature
R	LV2	Resistance at reference temperature
TC1	LV3	First temperature coefficient
TC2	LV4	Second temperature coefficient

Capacitor

Name	Alias	Description
CEFF	LV1	Computed effective capacitance
IC	LV2	Initial condition
Q	LX0	Charge stored in capacitor
CURR	LX1	Current flowing through capacitor
VOLT	LX2	Voltage across capacitor
–	LX3	Capacitance (not used in HSPICE releases after 95.3)

Inductor

Name	Alias	Description
LEFF	LV1	Computed effective inductance
IC	LV2	Initial condition
FLUX	LX0	Flux in the inductor
VOLT	LX1	Voltage across inductor
CURR	LX2	Current flowing through inductor
–	LX4	Inductance (not used in HSPICE releases after 95.3)

Mutual Inductor

Name	Alias	Description
K	LV1	Mutual inductance

Voltage-Controlled Voltage Source (E Element)

Name	Alias	Description
VOLT	LX0	Source voltage
CURR	LX1	Current through source
CV	LX2	Controlling voltage
DV	LX3	Derivative of source voltage with respect to control current

Current-Controlled Current Source (F Element)

Name	Alias	Description
CURR	LX0	Current through source
CI	LX1	Controlling current
DI	LX2	Derivative of source current with respect to control current

Voltage-Controlled Current Source (G Element)

Name	Alias	Description
CURR	LX0	Current through the source, if VCCS
R	LX0	Resistance value, if VCR
C	LX0	Capacitance value, if VCCAP
CV	LX1	Controlling voltage
CQ	LX1	Capacitance charge, if VCCAP
DI	LX2	Derivative of source current with respect to control voltage
ICAP	LX2	Capacitance current, if VCCAP
VCAP	LX3	Voltage across capacitance, if VCCAP

Current-Controlled Voltage Source (H Element)

Name	Alias	Description
VOLT	LX0	Source voltage
CURR	LX1	Source current
CI	LX2	Controlling current

Name	Alias	Description
DV	LX3	Derivative of source voltage with respect to control current

Independent Voltage Source

Name	Alias	Description
VOLT	LV1	DC/transient voltage
VOLTM	LV2	AC voltage magnitude
VOLTP	LV3	AC voltage phase

Independent Current Source

Name	Alias	Description
CURR	LV1	DC/transient current
CURRM	LV2	AC current magnitude
CURRP	LV3	AC current phase

Diode

Name	Alias	Description
AREA	LV1	Diode area factor
AREAX	LV23	Area after scaling
IC	LV2	Initial voltage across diode
VD	LX0	Voltage across diode (VD), excluding RS (series resistance)
IDC	LX1	DC current through diode (ID), excluding RS. Total diode current is the sum of IDC and ICAP
GD	LX2	Equivalent conductance (GD)
QD	LX3	Charge of diode capacitor (QD)
ICAP	LX4	Current through diode capacitor. Total diode current is the sum of IDC and ICAP.
C	LX5	Total diode capacitance
PID	LX7	Photo current in diode

BJT

Name	Alias	Description
AREA	LV1	Area factor
ICVBE	LV2	Initial condition for base-emitter voltage (VBE)
ICVCE	LV3	Initial condition for collector-emitter voltage (VCE)
MULT	LV4	Number of multiple BJTs
FT	LV5	FT (Unity gain bandwidth)
ISUB	LV6	Substrate current
GSUB	LV7	Substrate conductance
LOGIC	LV8	LOG 10 (IC)
LOGIB	LV9	LOG 10 (IB)
BETA	LV10	BETA
LOGBETAI	LV11	LOG 10 (BETA) current
ICTOL	LV12	Collector current tolerance
IBTOL	LV13	Base current tolerance
RB	LV14	Base resistance
GRE	LV15	Emitter conductance, 1/RE
GRC	LV16	Collector conductance, 1/RC
PIBC	LV18	Photo current, base-collector
PIBE	LV19	Photo current, base-emitter
VBE	LX0	VBE
VBC	LX1	Base-collector voltage (VBC)
CCO	LX2	Collector current (CCO)
CBO	LX3	Base current (CBO)
GPI	LX4	$g_{\pi} = {}^{1}ib / {}^{1}vbe$, constant vbc
GU	LX5	$g\mu = {}^{1}ib / {}^{1}vbc$, constant vbe
GM	LX6	$g_m = {}^{1}ic / {}^{1}vbe + {}^{1}ic / {}^{1}vbe$, constant vce
G0	LX7	$g_0 = {}^{1}ic / {}^{1}vce$, constant vbe
QBE	LX8	Base-emitter charge (QBE)
CQBE	LX9	Base-emitter charge current (CQBE)
QBC	LX10	Base-collector charge (QBC)
CQBC	LX11	Base-collector charge current (CQBC)
QCS	LX12	Current-substrate charge (QCS)

Name	Alias	Description
CQCS	LX13	Current-substrate charge current (CQCS)
QBX	LX14	Base-internal base charge (QBX)
CQBX	LX15	Base-internal base charge current (CQBX)
GXO	LX16	1/Rbeff Internal conductance (GXO)
CEXBC	LX17	Base-collector equivalent current (CEXBC)
–	LX18	Base-collector conductance (GEQCBO) (not used in HSPICE releases after 95.3)
CAP_BE	LX19	cbe capacitance ($C\Pi$)
CAP_IBC	LX20	cbc internal base-collector capacitance ($C\mu$)
CAP_SCB	LX21	csc substrate-collector capacitance for vertical transistors csb substrate-base capacitance for lateral transistors
CAP_XBC	LX22	cbcx external base-collector capacitance
CMCMO	LX23	^{1}(TF*IBE) /^{1}vbc
VSUB	LX24	Substrate voltage

JFET

Name	Alias	Description
AREA	LV1	JFET area factor
VDS	LV2	Initial drain-source voltage
VGS	LV3	Initial gate-source voltage
PIGD	LV16	Photo current, gate-drain in JFET
PIGS	LV17	Photo current, gate-source in JFET
VGS	LX0	VGS
VGD	LX1	Gate-drain voltage (VGD)
CGSO	LX2	Gate-to-source (CGSO)
CDO	LX3	Drain current (CDO)
CGDO	LX4	Gate-to-drain current (CGDO)
GMO	LX5	Transconductance (GMO)
GDSO	LX6	Drain-source transconductance (GDSO)

Name	Alias	Description
GGSO	LX7	Gate-source transconductance (GGSO)
GGDO	LX8	Gate-drain transconductance (GGDO)
QGS	LX9	Gate-source charge (QGS)
CQGS	LX10	Gate-source charge current (CQGS)
QGD	LX11	Gate-drain charge (QGD)
CQGD	LX12	Gate-drain charge current (CQGD)
CAP_GS	LX13	Gate-source capacitance
CAP_GD	LX14	Gate-drain capacitance
–	LX15	Body-source voltage (not used in HSPICE releases after 95.3)
QDS	LX16	Drain-source charge (QDS)
CQDS	LX17	Drain-source charge current (CQDS)
GMBS	LX18	Drain-body (backgate) transconductance (GMBS)

MOSFET

Name	Alias	Description
L	LV1	Channel length (L)
W	LV2	Channel width (W)
AD	LV3	Area of the drain diode (AD)
AS	LV4	Area of the source diode (AS)
ICVDS	LV5	Initial condition for drain-source voltage (VDS)
ICVGS	LV6	Initial condition for gate-source voltage (VGS)
ICVBS	LV7	Initial condition for bulk-source voltage (VBS)
–	LV8	Device polarity: 1 = forward, -1 = reverse (not used in HSPICE releases after 95.3)
VTH	LV9	Threshold voltage (bias dependent)
VDSAT	LV10	Saturation voltage (VDSAT)
PD	LV11	Drain diode periphery (PD)
PS	LV12	Source diode periphery (PS)
RDS	LV13	Drain resistance (squares) (RDS)
RSS	LV14	Source resistance (squares) (RSS)

Name	Alias	Description
XQC	LV15	Charge sharing coefficient (XQC)
GDEFF	LV16	Effective drain conductance (1/ RDeff)
GSEFF	LV17	Effective source conductance (1/ RSeff)
IDBS	LV18	Drain-bulk saturation current at -1 volt bias
ISBS	LV19	Source-bulk saturation current at -1 volt bias
VDBEFF	LV20	Effective drain bulk voltage
BETAEFF	LV21	BETA effective
GAMMAEFF	LV22	GAMMA effective
DELTAL	LV23	ΔL (MOS6 amount of channel length modulation) (only valid for LEVELs 1, 2, 3 and 6)
UBEFF	LV24	UB effective (only valid for LEVELs 1, 2, 3 and 6)
VG	LV25	VG drive (only valid for LEVELs 1, 2, 3 and 6)
VFBEFF	LV26	VFB effective
–	LV31	Drain current tolerance (not used in HSPICE releases after 95.3)
IDSTOL	LV32	Source diode current tolerance
IDDTOL	LV33	Drain diode current tolerance
COVLGS	LV36	Gate-source overlap capacitance
COVLGD	LV37	Gate-drain overlap capacitance
COVLGB	LV38	Gate-bulk overlap capacitance
VBS	LX1	Bulk-source voltage (VBS)
VGS	LX2	Gate-source voltage (VGS)
VDS	LX3	Drain-source voltage (VDS)
CDO	LX4	DC drain current (CDO)
CBSO	LX5	DC source-bulk diode current (CBSO)
CBDO	LX6	DC drain-bulk diode current (CBDO)
GMO	LX7	DC gate transconductance (GMO)
GDSO	LX8	DC drain-source conductance (GDSO)

Name	Alias	Description
GMBSO	LX9	DC substrate transconductance (GMBSO)
GBDO	LX10	Conductance of the drain diode (GBDO)
GBSO	LX11	Conductance of the source diode (GBSO)
Meyer and Charge Conservation Model Parameters		
QB	LX12	Bulk charge (QB)
CQB	LX13	Bulk charge current (CQB)
QG	LX14	Gate charge (QG)
CQG	LX15	Gate charge current (CQG)
QD	LX16	Channel charge (QD)
CQD	LX17	Channel charge current (CQD)
CGGBO	LX18	$GGBO = \partial Qg/\partial Vgb$ = CGS + CGD + CGB
CGDBO	LX19	$CGDBO = \partial Qg/\partial Vdb$, (for Meyer CGD = -CGDBO)
CGSBO	LX20	$CGSBO = \partial Qg/\partial Vsb$, (for Meyer CGS = -CGSBO)
CBGBO	LX21	$BGBO = \partial Qb/\partial Vgb$, (for Meyer CGB = -CBGBO)
CBDBO	LX22	CBDBO = -dQb/dVd intrinsic floating body-to-drain capacitance
CBSBO	LX23	CBSBO = -dQb/dVs intrinsic floating body-to-source capacitance
QBD	LX24	Drain-bulk charge (QBD)
–	LX25	Drain-bulk charge current (CQBD) (not used in HSPICE releases after 95.3)
QBS	LX26	Source-bulk charge (QBS)
–	LX27	Source-bulk charge current (CQBS) (not used in HSPICE releases after 95.3)
CAP_BS	LX28	Bulk-source capacitance
CAP_BD	LX29	Bulk-drain capacitance
CQS	LX31	Channel charge current (CQS)
CDGBO	LX32	$CDGBO = \partial Qd/\partial Vgb$

Name	Alias	Description
CDDBO	LX33	$DDBO = \partial Qd/\partial Vdb$
CDSBO	LX34	$DSBO = \partial Qd/\partial Vsb$

Saturable Core Element

Name	Alias	Description
MU	LX0	Dynamic permeability (mu) Weber/(amp-turn-meter)
H	LX1	Magnetizing force (H) Ampere-turns/meter
B	LX2	Magnetic flux density (B) Webers/meter^2

Saturable Core Winding

Name	Alias	Description
LEFF	LV1	Effective winding inductance (Henry)
IC	LV2	Initial condition
FLUX	LX0	Flux through winding (Weber-turn)
VOLT	LX1	Voltage across winding (Volt)