

HSPICE[®] Command Reference

Version X-2005.09, September 2005

SYNOPTSYS[®]

Copyright Notice and Proprietary Information

Copyright © 2005 Synopsys, Inc. All rights reserved. This software and documentation contain confidential and proprietary information that is the property of Synopsys, Inc. The software and documentation are furnished under a license agreement and may be used or copied only in accordance with the terms of the license agreement. No part of the software and documentation may be reproduced, transmitted, or translated, in any form or by any means, electronic, mechanical, manual, optical, or otherwise, without prior written permission of Synopsys, Inc., or as expressly provided by the license agreement.

Right to Copy Documentation

The license agreement with Synopsys permits licensee to make copies of the documentation for its internal use only. Each copy shall include all copyrights, trademarks, service marks, and proprietary rights notices, if any. Licensee must assign sequential numbers to all copies. These copies shall contain the following legend on the cover page:

“This document is duplicated with the permission of Synopsys, Inc., for the exclusive use of _____ and its employees. This is copy number _____.”

Destination Control Statement

All technical data contained in this publication is subject to the export control laws of the United States of America. Disclosure to nationals of other countries contrary to United States law is prohibited. It is the reader's responsibility to determine the applicable regulations and to comply with them.

Disclaimer

SYNOPSYS, INC., AND ITS LICENSORS MAKE NO WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, WITH REGARD TO THIS MATERIAL, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

Registered Trademarks (®)

Synopsys, AMPS, Arcadia, C Level Design, C2HDL, C2V, C2VHDL, Cadabra, Calaveras Algorithm, CATS, CRITIC, CSim, Design Compiler, DesignPower, DesignWare, EPIC, Formality, HSIM, HSPICE, Hypermodel, iN-Phase, in-Sync, Leda, MAST, Meta, Meta-Software, ModelTools, NanoSim, OpenVera, PathMill, Photolynx, Physical Compiler, PowerMill, PrimeTime, RailMill, RapidScript, Saber, SiVL, SNUG, SolvNet, Superlog, System Compiler, Testify, TetraMAX, TimeMill, TMA, VCS, Vera, and Virtual Stepper are registered trademarks of Synopsys, Inc.

Trademarks (™)

Active Parasitics, AFGen, Apollo, Apollo II, Apollo-DPII, Apollo-GA, ApolloGAI, Astro, Astro-Rail, Astro-Xtalk, Aurora, AvanTestchip, AvanWaves, BCView, Behavioral Compiler, BOA, BRT, Cedar, ChipPlanner, Circuit Analysis, Columbia, Columbia-CE, Comet 3D, Cosmos, CosmosEnterprise, CosmosLE, CosmosScope, CosmosSE, Cyclelink, Davinci, DC Expert, DC Expert *Plus*, DC Professional, DC Ultra, DC Ultra Plus, Design Advisor, Design Analyzer, Design Vision, DesignerHDL, DesignTime, DFM-Workbench, Direct RTL, Direct Silicon Access, Discovery, DW8051, DWPCI, Dynamic-Macromodeling, Dynamic Model Switcher, ECL Compiler, ECO Compiler, EDAnavigator, Encore, Encore PQ, Evaccess, ExpressModel, Floorplan Manager, Formal Model Checker, FoundryModel, FPGA Compiler II, FPGA *Express*, Frame Compiler, Galaxy, Gatran, HANEX, HDL Advisor, HDL Compiler, Hercules, Hercules-Explorer, Hercules-II, Hierarchical Optimization Technology, High Performance Option, HotPlace, HSIM^{Plus}, HSPICE-Link, iN-Tandem, Integrator, Interactive Waveform Viewer, i-Virtual Stepper, Jupiter, Jupiter-DP, JupiterXT, JupiterXT-ASIC, JvXtreme, Liberty, Libra-Passport, Library Compiler, Libra-Visa, Magellan, Mars, Mars-Rail, Mars-Xtalk, Medici, Metacapture, Metacircuit, Metamanager, Metamixsim, Milkyway, ModelSource, Module Compiler, MS-3200, MS-3400, Nova Product Family, Nova-ExploreRTL, Nova-Trans, Nova-VeriLint, Nova-VHDLint, Optimum Silicon, Orion_ec, Parasitic View, Passport, Planet, Planet-PL, Planet-RTL, Polaris, Polaris-CBS, Polaris-MT, Power Compiler, PowerCODE, PowerGate, ProFPGA, ProGen, Prospector, Protocol Compiler, PSMGen, Raphael, Raphael-NES, RoadRunner, RTL Analyzer, Saturn, ScanBand, Schematic Compiler, Scirocco, Scirocco-i, Shadow Debugger, Silicon Blueprint, Silicon Early Access, SinglePass-SoC, Smart Extraction, SmartLicense, SmartModel Library, Softwire, Source-Level Design, Star, Star-DC, Star-MS, Star-MTB, Star-Power, Star-Rail, Star-RC, Star-RCXT, Star-Sim, Star-SimXT, Star-Time, Star-XP, SWIFT, Taurus, TimeSlice, TimeTracker, Timing Annotator, TopoPlace, TopoRoute, Trace-On-Demand, True-Hspice, TSUPREM-4, TymeWare, VCS Express, VCSi, Venus, Verification Portal, VFormal, VHDL Compiler, VHDL System Simulator, VirSim, and VMC are trademarks of Synopsys, Inc.

Service Marks (SM)

MAP-in, SVP Café, and TAP-in are service marks of Synopsys, Inc.

SystemC is a trademark of the Open SystemC Initiative and is used under license.
ARM and AMBA are registered trademarks of ARM Limited.
All other product or company names may be trademarks of their respective owners.

Printed in the U.S.A.

HSPICE[®] Command Reference, X-2005.09

Contents

Inside This Manual	xv
The HSPICE Documentation Set	xvi
Searching Across the HSPICE Documentation Set	xvii
Other Related Publications	xvii
Conventions	xviii
Customer Support	xix

1. Command Categories	1
Alter Blocks	1
Analysis	1
Conditional Block	2
Digital Vector	2
Encryption	2
Field Solver	3
Files	3
Input/Output Buffer Information Specification (IBIS)	3
Library Management	3
Model Definition	4
Node Naming	4
Output Porting	4
Setup	4
Simulation Runs	5
Subcircuits	5
Verilog-A	5

Contents

2. Commands in HSPICE Netlists	7
.AC	9
.ALIAS	14
.ALTER	16
.BIASCHK	18
.CONNECT	23
.DATA	25
.DC	32
.DCMATCH	38
.DCVOLT	40
.DEL LIB	42
.DISTO	46
.DOUT	49
.EBD	52
.ELSE	54
.ELSEIF	55
.END	56
.ENDDATA	57
.ENDIF	58
.ENDL	59
.ENDS	60
.EOM	61
.FFT	62
.FOUR	65
.FSOPTIONS	66
.GLOBAL	68
.GRAPH	69
.HDL	71
.IBIS	72
.IC	76

Contents

.ICM	78
.IF	79
.INCLUDE	81
.LAYERSTACK	82
.LIB	84
.LIN	88
.LOAD	91
.MACRO	93
.MALIAS	96
.MATERIAL	98
.MEASURE	100
.MEASURE (Rise, Fall, and Delay Measurements)	101
.MEASURE (Average, RMS, and Peak Measurements)	105
.MEASURE (FIND and WHEN)	107
.MEASURE (Equation Evaluation/ Arithmetic Expression)	111
.MEASURE (Average, RMS, MIN, MAX, INTEG, and PP)	112
.MEASURE (Integral Function)	115
.MEASURE (Derivative Function)	116
.MEASURE (Error Function)	119
.MEASURE (Pushout Bisection)	121
.MODEL	123
.NET	129
.NODESET	131
.NOISE	132
.OP	133
.OPTION	135
.PARAM	137
.PAT	141
.PKG	143
.PLOT	145
.PRINT	147

Contents

.PROBE	151
.PROTECT	153
.PZ	154
.SAMPLE	156
.SAVE	157
.SENS	159
.SHAPE	161
.SHAPE (Defining Rectangles)	162
.SHAPE (Defining Circles)	163
.SHAPE (Defining Polygons)	164
.SHAPE (Defining Strip Polygons)	166
.STIM	167
.SUBCKT	172
.TEMP	175
.TF	177
.TITLE	178
.TRAN	179
.UNPROTECT	184
.VEC	185
.WIDTH	186
<hr/>	
3. Options in HSPICE Netlists	187
General Control Options	188
CPU Options	188
Interface Options	188
Analysis Options	189
Error Options	189
Version Option	189
Model Analysis Options	189
General Model Analysis Options	189
MOSFET Model Analysis Options	189

Inductor Model Analysis Options	190
BJT and Diode Model Analysis Options	190
DC Operating Point, DC Sweep, and Pole/Zero Options	190
DC Accuracy Options	190
DC Matrix Options	190
DC Pole/Zero I/O Options	190
DC Convergence Options	191
DC Initialization Control Options	191
Transient and AC Small Signal Analysis Options	191
Transient/AC Accuracy Options	191
Transient/AC Speed Options	192
Transient/AC Timestep Options	192
Transient/AC Algorithm Options	192
.BIASCHK Options	192
Transient Control Options	193
Transient Control Method Options	193
Transient Control Tolerance Options	193
Transient Control Limit Options	193
Transient Control Matrix Options	194
Iteration Count Dynamic Timestep Options	194
Input/Output Options	194
AC Control Options	194
Common Model Interface Options	194
Verilog-A Options	194
.OPTION ABSH	195
.OPTION ABSI	196
.OPTION ABSMOS	197
.OPTION ABSTOL	198
.OPTION ABSV	199
.OPTION ABSVAR	200
.OPTION ABSVDC	201
.OPTION ACCT	202
.OPTION ACCURATE	203
.OPTION ACOUT	204

Contents

.OPTION ALT999 or ALT9999	205
OPTION ALTCC	206
.OPTION ALTCHK	207
.OPTION ARTIST	208
.OPTION ASPEC	209
.OPTION AUTOSTOP	210
.OPTION BADCHR	211
.OPTION BEEP	212
.OPTION BIASFILE	213
.OPTION BIAWARN	214
.OPTION BINPRINT	215
.OPTION BKPSIZ	216
.OPTION BRIEF	217
.OPTION BYPASS	218
.OPTION BYTOL	219
.OPTION CAPTAB	220
.OPTION CDS	221
.OPTION CHGTOL	222
.OPTION CMIFLAG	223
.OPTION CO	224
.OPTION CONVERGE	225
.OPTION CPTIME	226
.OPTION CSDF	227
.OPTION CSHDC	228
.OPTION CSHUNT	229
.OPTION CUSTCMI	230
.OPTION CVTOL	231
.OPTION D_IBIS	232
.OPTION DCAP	233
.OPTION DCCAP	234
.OPTION DCFOR	235

.OPTION DCHOLD	236
.OPTION DCIC	237
.OPTION DCON	238
.OPTION DCSTEP	239
.OPTION DCTRAN	240
.OPTION DEFAD	241
.OPTION DEFAS	242
.OPTION DEFL	243
.OPTION DEFNRD	244
.OPTION DEFNRS	245
.OPTION DEFPPD	246
.OPTION DEFPS	247
.OPTION DEFW	248
.OPTION DELMAX	249
.OPTION DI	250
.OPTION DIAGNOSTIC	251
.OPTION DLENCSDF	252
.OPTION DV	253
.OPTION DVDT	254
.OPTION DVTR	255
.OPTION EPSMIN	256
.OPTION EXPLI	257
.OPTION EXPMAX	258
.OPTION FAST	259
.OPTION FFTOUT	260
.OPTION FS	261
.OPTION FT	262
.OPTION GDCPATH	263
.OPTION GENK	264
.OPTION GMAX	265
.OPTION GMIN	266

Contents

.OPTION GMINDC	267
.OPTION GRAMP	268
.OPTION GSHDC	269
.OPTION GSHUNT	270
.OPTION H9007	271
.OPTION HIER_SCALE	272
.OPTION ICSWEEP	273
.OPTION IMAX	274
.OPTION IMIN	275
.OPTION INGOLD	276
.OPTION INTERP	277
.OPTION ITL1	278
.OPTION ITL2	279
.OPTION ITL3	280
.OPTION ITL4	281
.OPTION ITL5	282
.OPTION ITLPTRAN	283
.OPTION ITLPZ	284
.OPTION ITRPRT	285
.OPTION KCLTEST	286
.OPTION KLIM	287
.OPTION LENNAM	288
.OPTION LIMPTS	289
.OPTION LIMITIM	290
.OPTION LIST	291
.OPTION LVLTIM	292
.OPTION MAXAMP	293
.OPTION MAXORD	294
.OPTION MBYPASS	295
.OPTION MCBRIEF	296
.OPTION MEASDGT	297

Contents

.OPTION MEASFAIL	298
.OPTION MEASFILE	299
.OPTION MEASSORT	300
.OPTION MEASOUT	301
.OPTION MENTOR	302
.OPTION METHOD	303
.OPTION MODMONTE	304
.OPTION MODSRH	305
.OPTION MONTECON	306
.OPTION MU	307
.OPTION NEWTOL	308
.OPTION NODE	309
.OPTION NOELCK	310
.OPTION NOISEMINFREQ	311
.OPTION NOMOD	312
.OPTION NOPAGE	313
.OPTION NOPIV	314
.OPTION NOTOP	315
.OPTION NOWARN	316
.OPTION NUMDGT	317
.OPTION NXX	318
.OPTION OFF	319
.OPTION OPFILE	320
.OPTION OPTLST	321
.OPTION OPTS	322
.OPTION PARHIER	323
.OPTION PATHNUM	324
.OPTION PIVOT	325
.OPTION PIVREF	327
.OPTION PIVREL	328
.OPTION PIVTOL	329

Contents

.OPTION PLIM	330
.OPTION POST	331
.OPTION POSTLVL	332
.OPTION POST_VERSION	333
.OPTION POSTTOP	334
.OPTION PROBE	335
.OPTION PSF	336
.OPTION PURETP	337
.OPTION PUTMEAS	338
.OPTION RELH	339
.OPTION RELI	340
.OPTION RELMOS	341
.OPTION RELQ	342
.OPTION RELTOL	343
.OPTION RELV	344
.OPTION RELVAR	345
.OPTION RELVDC	346
.OPTION RESMIN	347
.OPTION RISETIME	348
.OPTION RMAX	349
.OPTION RMIN	350
.OPTION RUNLVL	351
.OPTION SCALE	353
.OPTION SCALM	354
.OPTION SDA	355
.OPTION SEARCH	356
.OPTION SEED	357
.OPTION SLOPETOL	358
.OPTION SPARSE	359
.OPTION SPICE	360
.OPTION SPMODEL	361

.OPTION STATFL	362
.OPTION SYMB	363
.OPTION TIMERES	364
.OPTION TNOM	365
.OPTION TRCON	366
.OPTION TRTOL	368
.OPTION UNWRAP	369
.OPTION VAMODEL	370
.OPTION VERIFY	371
.OPTION VFLOOR	372
.OPTION VNTOL	373
.OPTION WACC	374
.OPTION WNFLAG	375
.OPTION WARNLIMIT	376
.OPTION WL	377
.OPTION XDTEMP	378
.OPTION ZUKEN	379

4. Commands in Digital Vector Files	393
ENABLE	394
IDELAY	395
IO	397
ODELAY	398
OUT or OUTZ	400
PERIOD	401
RADIX	402
SLOPE	403
TDELAY	404
TFALL	406
TRISE	407
TRIZ	409

Contents

TSKIP.....	410
TUNIT	411
VIH.....	412
VIL	413
VNAME	414
VOH.....	416
VOL	418
VREF.....	420
VTH	421

Index	423
--------------------	------------

About This Manual

This manual describes the individual HSPICE commands you can use to simulate and analyze your circuit designs.

Inside This Manual

This manual contains the chapters described below. For descriptions of the other manuals in the HSPICE documentation set, see the next section, [The HSPICE Documentation Set](#).

Chapter	Description
Chapter 1, Command Categories	Lists all commands you can use in HSPICE, arranged by task.
Chapter 2, Commands in HSPICE Netlists	Contains an alphabetical listing of all commands you can use in an HSPICE netlist.
Chapter 3, Options in HSPICE Netlists	Describes the simulation options you can set using various forms of the <code>.OPTION</code> command.
Chapter 4, Commands in Digital Vector Files	Contains an alphabetical listing of the commands you can use in a digital vector file.

About This Manual

The HSPICE Documentation Set

The HSPICE Documentation Set

This manual is a part of the HSPICE documentation set, which includes the following manuals:

Manual	Description
HSPICE Simulation and Analysis User Guide	Describes how to use HSPICE to simulate and analyze your circuit designs. This is the main HSPICE user guide.
HSPICE Signal Integrity Guide	Describes how to use HSPICE to maintain signal integrity in your chip design.
HSPICE Applications Manual	Provides application examples and additional HSPICE user information.
HSPICE Command Reference	Provides reference information for HSPICE commands.
HPSPICE Elements and Device Models Manual	Describes standard models you can use when simulating your circuit designs in HSPICE, including passive devices, diodes, JFET and MESFET devices, and BJT devices.
HPSPICE MOSFET Models Manual	Describes standard MOSFET models you can use when simulating your circuit designs in HSPICE.
HSPICE RF Manual	Describes a special set of analysis and design capabilities added to HSPICE to support RF and high-speed circuit design.
AvanWaves User Guide	Describes the AvanWaves tool, which you can use to display waveforms generated during HSPICE circuit design simulation.

Manual	Description
HSPICE Quick Reference Guide	Provides key reference information for using HSPICE, including syntax and descriptions for commands, options, parameters, elements, and more.
HSPICE Device Models Quick Reference Guide	Provides key reference information for using HSPICE device models, including passive devices, diodes, JFET and MESFET devices, and BJT devices.

Searching Across the HSPICE Documentation Set

Synopsys includes an index with your HSPICE documentation that lets you search the entire HSPICE documentation set for a particular topic or keyword. In a single operation, you can instantly generate a list of hits that are hyperlinked to the occurrences of your search term. For information on how to perform searches across multiple PDF documents, see the HSPICE release notes (available on SolvNet at <http://solvnet.synopsys.com>) or the Adobe Reader online help.

Note: To use this feature, the HSPICE documentation files, the Index directory, and the index.pdx file must reside in the same directory. (This is the default installation for Synopsys documentation.) Also, Adobe Acrobat must be invoked as a standalone application rather than as a plug-in to your web browser.

Other Related Publications

For additional information about HSPICE, see:

- The HSPICE release notes, available on SolvNet (see [Accessing SolvNet on page xix](#))
- Documentation on the Web, which provides PDF documents and is available through SolvNet at <http://solvnet.synopsys.com>
- The Synopsys MediaDocs Shop, from which you can order printed copies of Synopsys documents, at <http://mediadocs.synopsys.com>

About This Manual

Conventions

You might also want to refer to the documentation for the following related Synopsys products:

- CosmosScope
- Aurora
- Raphael
- VCS

Conventions

The following conventions are used in Synopsys documentation:

Convention	Description
Courier	Indicates command syntax.
<i>Italic</i>	Indicates a user-defined value, such as <i>object_name</i> .
Bold	Indicates user input—text you type verbatim—in syntax and examples.
[]	Denotes optional parameters, such as <code>write_file [-f filename]</code>
...	Indicates that a parameter can be repeated as many times as necessary: <code>pin1 [pin2 ... pinN]</code>
	Indicates a choice among alternatives, such as <code>low medium high</code>
\	Indicates a continuation of a command line.
/	Indicates levels of directory structure.
Edit > Copy	Indicates a path to a menu command, such as opening the Edit menu and choosing Copy.
Control-c	Indicates a keyboard combination, such as holding down the Control key and pressing c.

Customer Support

Customer support is available through SolvNet online customer support and through contacting the Synopsys Technical Support Center.

Accessing SolvNet

SolvNet includes an electronic knowledge base of technical articles and answers to frequently asked questions about Synopsys tools. SolvNet also gives you access to a wide range of Synopsys online services, which include downloading software, viewing Documentation on the Web, and entering a call to the Support Center.

To access SolvNet:

1. Go to the SolvNet Web page at <http://solvnet.synopsys.com>.
2. If prompted, enter your user name and password. (If you do not have a Synopsys user name and password, follow the instructions to register with SolvNet.)

If you need help using SolvNet, click SolvNet Help in the Support Resources section.

Contacting the Synopsys Technical Support Center

If you have problems, questions, or suggestions, you can contact the Synopsys Technical Support Center in the following ways:

- Open a call to your local support center from the Web by going to <http://solvnet.synopsys.com> (Synopsys user name and password required), then clicking “Enter a Call to the Support Center.”
- Send an e-mail message to your local support center.
 - E-mail support_center@synopsys.com from within North America.
 - Find other local support center e-mail addresses at http://www.synopsys.com/support/support_ctr.
- Telephone your local support center.
 - Call (800) 245-8005 from within the continental United States.

About This Manual
Customer Support

- Call (650) 584-4200 from Canada.
- Find other local support center telephone numbers at http://www.synopsys.com/support/support_ctr.

Command Categories

Lists all commands you can use in HSPICE, arranged by task.

Alter Blocks

Use these commands in your HSPICE netlist to run alternative simulations of your netlist by using different data.

`.ALIAS`
`.TEMP`

`.ALTER`

`.DEL LIB`

Analysis

Use these commands in your HSPICE netlist to start different types of HSPICE analysis to save the simulation results into a file, and to load the results of a previous simulation into a new simulation.

`.AC`

`.LIN`

`.SAMPLE`

`.DC`

`.NET`

`.SENS`

`.DCMATCH`

`.NOISE`

`.TEMP`

1: Command Categories

Conditional Block

<code>.DISTO</code>	<code>.OP</code>	<code>.TF</code>
<code>.FFT</code>	<code>.PAT</code>	<code>.TRAN</code>
<code>.FOUR</code>	<code>.PZ</code>	

Conditional Block

Use these commands in your HSPICE netlist to setup a conditional block. HSPICE does not execute the commands in the conditional block, unless the specified conditions are true.

<code>.ELSE</code>	<code>.ELSEIF</code>	<code>.ENDIF</code>
<code>.IF</code>		

Digital Vector

Use these commands in your digital vector (VEC) file.

<code>ENABLE</code>	<code>SLOPE</code>	<code>VIH</code>
<code>IDELAY</code>	<code>TDELAY</code>	<code>VIL</code>
<code>IO</code>	<code>TFALL</code>	<code>VNAME</code>
<code>ODELAY</code>	<code>TRISE</code>	<code>VOH</code>
<code>OUT or OUTZ</code>	<code>TRIZ</code>	<code>VOL</code>
<code>PERIOD</code>	<code>TSKIP</code>	<code>VREF</code>
<code>RADIX</code>	<code>TUNIT</code>	<code>VTH</code>

Encryption

Use these commands in your HSPICE netlist to mark the start and end of an encrypted section of a netlist.

<code>.PROTECT</code>	<code>.UNPROTECT</code>
-----------------------	-------------------------

1: Command Categories

Field Solver

Field Solver

Use these commands in your HSPICE netlist to define a field solver.

`.FSOPTIONS`

`.LAYERSTACK`

`.MATERIAL`

`.SHAPE`

Files

Use this command in your HSPICE netlist to call other files that are not part of the netlist.

`.VEC`

Input/Output Buffer Information Specification (IBIS)

Use these commands in your HSPICE netlist for specifying input/output buffer information.

`.EBD`

`.IBIS`

`.ICM`

`.PKG`

Library Management

Use these commands in your HSPICE netlist to manage libraries of circuit designs, and to call other files when simulating your netlist.

`.DEL LIB`

`.INCLUDE`

`.PROTECT`

`.ENDL`

`.LIB`

`.UNPROTECT`

1: Command Categories

Model Definition

Model Definition

Use these commands in your HSPICE netlist to define models.

`.MALIAS`

`.MODEL`

Node Naming

Use these commands in your HSPICE netlist to name nodes in circuit designs.

`.CONNECT`

`.GLOBAL`

Output Porting

Use these commands in your HSPICE netlist to specify the output of a simulation to a printer, plotter, or graph. You can also define the parameters to measure, and to report in the simulation output.

`.BIASCHK`

`.MEASURE`

`.PROBE`

`.DOUT`

`.PLOT`

`.STIM`

`.GRAPH`

`.PRINT`

`.WIDTH`

Setup

Use these commands in your HSPICE netlist to setup your netlist for simulation.

`.DATA`

`.IC`

`.PARAM`

`.DCVOLT`

`.LOAD`

`.SAVE`

`.ENDDATA`

`.NODESET`

`.TITLE`

`.GLOBAL`

`.OPTION`

1: Command Categories

Simulation Runs

Simulation Runs

Use these commands in your HSPICE netlist to mark the start and end of individual simulation runs, and conditions that apply throughout an individual simulation run.

`.END`

`.TEMP`

`.TITLE`

Subcircuits

Use these commands in your HSPICE netlist to define subcircuits, and to add instances of subcircuits to your netlist.

`.ENDS`

`.INCLUDE`

`.MODEL`

`.EOM`

`.MACRO`

`.SUBCKT`

Verilog-A

Use the following command in your HSPICE netlist to declare the Verilog-A source name and path within the netlist.

`.HDL`

1: Command Categories

Verilog-A

2

Commands in HSPICE Netlists

Contains an alphabetical listing of all commands you can use in an HSPICE netlist.

Here are the commands described in this chapter. For a list of commands grouped according to tasks that use each command, see [Chapter 1, Command Categories](#).

.AC	.FSOPTIONS	.OPTION
.ALIAS	.GLOBAL	.PARAM
.ALTER	.GRAPH	.PAT
.BIASCHK	.HDL	.PKG
.CONNECT	.IBIS	.PLOT
.DATA	.IC	.PRINT
.DC	.ICM	.PROBE
.DCMATCH	.IF	.PROTECT
.DCVOLT	.INCLUDE	.PZ

2: Commands in HSPICE Netlists

.DEL LIB	.LAYERSTACK	.SAMPLE
.DISTO	.LIB	.SAVE
.DOUT	.LIN	.SENS
.EBD	.LOAD	.SHAPE
.ELSE	.MACRO	.STIM
.ELSEIF	.MALIAS	.SUBCKT
.END	.MATERIAL	.TEMP
.ENDDATA	.MEASURE	.TF
.ENDIF	.MODEL	.TITLE
.ENDL	.NET	.TRAN
.ENDS	.NODESET	.UNPROTECT
.EOM	.NOISE	.VEC
.FFT	.OP	.WIDTH
.FOUR		

.AC**Syntax****Single/Double Sweep**

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

Sweep Using Parameters

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

In HSPICE RF, you can run a parameter sweep around a single analysis, but the parameter sweep cannot change .OPTION values.

Optimization

```
.AC DATA = datanm OPTIMIZE = opt_par_fun  
+ RESULTS = measnames MODEL = optmod
```

HSPICE RF supports optimization for bisection only.

2: Commands in HSPICE Netlists

.AC

Random/Monte Carlo

```
.AC type np fstart fstop <SWEEP MONTE = val>  
+ <firstrun = num1>
```

-or-

```
.AC type np fstart fstop <SWEEP MONTE = list<(>  
+ <num1:num2> <num3> <num5:num6> <num7> <)> >
```

Example 1

```
.AC DEC 10 1K 100MEG
```

This example performs a frequency sweep, by 10 points per decade, from 1kHz to 100MHz.

Example 2

```
.AC LIN 100 1 100HZ
```

This example runs a 100-point frequency sweep from 1- to 100-Hz.

Example 3

```
.AC DEC 10 1 10K SWEEP cload LIN 20 1pf 10pf
```

This example performs an AC analysis for each value of `cload`. This results from a linear sweep of `cload` between 1- and 10-pF (20 points), sweeping the frequency by 10 points per decade, from 1- to 10-kHz.

Example 4

```
.AC DEC 10 1 10K SWEEP rx POI 2 5k 15k
```

This example performs an AC analysis for each value of `rx`, 5k and 15k, sweeping the frequency by 10 points per decade, from 1- to 10-kHz.

Example 5

```
.AC DEC 10 1 10K SWEEP DATA = datanm
```

This example uses the `.DATA` statement to perform a series of AC analyses, modifying more than one parameter. The `datanm` file contains the parameters.

Example 6

```
.AC DEC 10 1 10K SWEEP MONTE = 30
```

This example illustrates a frequency sweep, and a Monte Carlo analysis (not supported in HSPICE RF) with 30 trials.

Example 7

```
AC DEC 10 1 10K SWEEP MONTE = 10 firstrun=15
```

This example illustrates a frequency sweep and a Monte Carlo analysis from the 15th to the 24th trials.

Example 8

```
.AC DEC 10 1 10K SWEEP MONTE = list(10 20:30 35:40 50)
```

This example illustrates a frequency sweep and a Monte Carlo analysis at 10th trial, and then from the 20th to 30th trial, followed by the 35th to 40th trial, and finally at 50th trial.

Description

You can use the .AC statement in several different formats, depending on the application as shown in the examples. You can also use the .AC statement to perform data-driven analysis in HSPICE.

If the input file includes an .AC statement, HSPICE runs AC analysis for the circuit, over a selected frequency range for each parameter in the second sweep.

For AC analysis, the data file must include at least one independent AC source element statement (for example, VI INPUT GND AC 1V). HSPICE checks for this condition, and reports a fatal error if you did not specify such AC sources.

You also cannot use this statement in HSPICE RF.

Argument	Definition
DATA = <i>datanm</i>	Data name, referenced in the .AC statement (not supported in HSPICE RF).
<i>incr</i>	Increment value of the voltage, current, element, or model parameter. If you use <i>type</i> variation, specify the <i>np</i> (number of points) instead of <i>incr</i> .

2: Commands in HSPICE Netlists

.AC

Argument	Definition
<i>fstart</i>	Starting frequency. If you use POI (list of points) type variation, use a list of frequency values, not <i>fstart</i> <i>fstop</i> .
<i>fstop</i>	Final frequency.
<i>MONTE = val</i>	Produces a number (<i>val</i>) of randomly-generated values (HSPICE only; not supported in HSPICE RF). HSPICE uses these values to select parameters from a distribution, either <i>Gaussian</i> , <i>Uniform</i> , or <i>Random Limit</i> .
<i>np</i>	Number of points, or points per decade or octave, depending on which keyword precedes it.
<i>start</i>	Starting voltage or current, or any parameter value for an element or model.
<i>stop</i>	Final voltage or current, or any parameter value for an element or a model.
<i>SWEEP</i>	Indicates that the .AC statement specifies a second sweep.
<i>TEMP</i>	Indicates a temperature sweep
<i>type</i>	Can be any of the following keywords: <ul style="list-style-type: none">• DEC – decade variation.• OCT – octave variation.• LIN – linear variation.• POI – list of points.
<i>var</i>	Name of an independent voltage or current source, element or model parameter, or the TEMP (temperature sweep) keyword. HSPICE or HSPICE RF supports source value sweep, referring to the source name (SPICE style). If you select a parameter sweep, a .DATA statement, and a temperature sweep, then you must choose a parameter name for the source value. You must also later refer to it in the .AC statement. The parameter name cannot start with V or I.

Argument	Definition
firstrun	The <code>val</code> value specifies the number of Monte Carlo iterations to perform. The <code>firstrun</code> value specifies the desired number of iterations. HSPICE runs from <code>num1</code> to <code>num1+val-1</code> .
list	The iterations at which HSPICE performs a Monte Carlo analysis. You can write more than one number after <code>list</code> . The colon represents "from ... to ...". Specifying only one number makes HSPICE run at only the specified point.

See Also

[.DC](#)
[.TRAN](#)

.ALIAS

Syntax

```
.ALIAS <model_name1> <model_name2>
```

Example 1

You delete a library named *poweramp*, that contains a model named *pa1*. Another library contains an equivalent model named *par1*. You can then alias the *pa1* model name to the *par1* model name:

```
.ALIAS pa1 par1
```

During simulation when HSPICE encounters a model named *pa1* in your netlist, it initially cannot find this model because you used a `.ALTER` statement to delete the library that contained the model. However, the `.ALIAS` statement indicates to use the *par1* model in place of the old *pa1* model and HSPICE *does* find this new model in another library, so simulation continues.

You must specify an old model name and a new model name to use in its place. You cannot use `.ALIAS` without any model names:

```
.ALIAS
```

or with only *one* model name:

```
.ALIAS pa1
```

You also cannot alias a model name to *more than one* model name, because then the simulator would not know which of these new models to use in place of the deleted or renamed model:

```
.ALIAS pa1 par1 par2
```

For the same reason, you cannot alias a model name to a second model name, and then alias the second model name to a third model name:

```
.ALIAS pa1 par1  
.ALIAS par1 par2
```

If your netlist does not contain an `.ALTER` command, and if the `.ALIAS` does not report a usage error, then the `.ALIAS` does not affect the simulation results.

Example 2

Your netlist might contain the statement:

```
.ALIAS myfet nfet
```

Without a `.ALTER` statement, HSPICE does not use `nfet` to replace `myfet` during simulation.

If your netlist contains one or more `.ALTER` commands, the first simulation uses the original `myfet` model. After the first simulation, if the netlist references `myfet` from a deleted library, `.ALIAS` substitutes `nfet` in place of the missing model.

- If HSPICE finds model definitions for both `myfet` and `nfet`, it reports an error and aborts.
- If HSPICE finds a model definition for `myfet`, but not for `nfet`, it reports a warning, and simulation continues by using the original `myfet` model.
- If HSPICE finds a model definition for `nfet`, but not for `myfet`, it reports a *replacement successful* message.

Description

You can use `.ALTER` statements to rename a model to rename a library containing a model, or to delete an entire library of models in HSPICE. If your netlist references the old model name, then after you use one of these types of `.ALTER` statements, HSPICE no longer finds this model.

Note: HSPICE RF does not support the `.ALIAS` statement.

For example, if you use `.DEL LIB` in the `.ALTER` block to delete a library, the `.ALTER` command deletes all models in this library. If your netlist references one or more models in the deleted library, then HSPICE no longer finds the models.

To resolve this issue, HSPICE provides a `.ALIAS` command to let you alias the old model name to another model name that HSPICE can find in the existing model libraries.

See Also

[.ALTER](#)
[.MALIAS](#)

.ALTER

Syntax

```
.ALTER <title_string>
```

Example

```
.ALTER simulation_run2
```

Description

You can use the `.ALTER` statement to rerun an HSPICE simulation by using different parameters and data. HSPICE RF does not support the `.ALTER` statement.

Use parameter (variable) values for `.PRINT` and `.PLOT` statements, before you alter them. The `.ALTER` block cannot include `.PRINT`, `.PLOT`, `.GRAPH` or any other input/output statements. You can include analysis statements (`.DC`, `.AC`, `.TRAN`, `.FOUR`, `.DISTO`, `.PZ`, and so on) in a `.ALTER` block in an input netlist file.

However, if you change only the analysis type, and you do not change the circuit itself, then simulation runs faster if you specify all analysis types in one block, instead of using separate `.ALTER` blocks for each analysis type.

The `.ALTER` sequence or block can contain:

- Element statements (except source elements)
- `.ALIAS` statements
- `.DATA` statements
- `.DEL LIB` statements
- `.IC` (initial condition) and `.NODESET` statements
- `.INCLUDE` statements
- `.LIB` statements
- `.MODEL` statements
- `.OP` statements
- `.OPTION` statements
- `.PARAM` statements
- `.TEMP` statements

- [.TF](#) statements
- [.TRAN](#), [.DC](#), and [.AC](#) statements

Argument	Definition
<i>title_string</i>	Any string up to 72 characters. HSPICE prints the appropriate title string for each <code>.ALTER</code> run in each section heading of the output listing, and in the graph data (<code>.tr#</code>) files.

Note: The `.MALIAS` command is not officially supported in `.ALTER` blocks.

See Also

[.OPTION MEASFILE](#)

.BIASCHK

Syntax

As an expression monitor:

```
.BIASCHK 'expression' <limit = lim> <noise = ns>  
+ <max = max> <min = min>  
+ <simulation = op | dc | tr | all> <monitor = v | i | w | l >  
+ <tstart = time1> <tstop = time2> <autostop>
```

As an element and model monitor:

```
.BIASCHK type <region=cutoff | linear | saturation>  
+ terminal1=t1 <terminal2=t2> <limit=lim>  
+ <noise=ns> <max=max> <min=min>  
+ <simulation=op | dc | tr | all> <monitor=v | i | w | l>  
+ <name=name1, name2, ...>  
+ <mname=modname_1, modname_2, ...>  
+ <tstart=time1> <tstop=time2> <autostop>  
+ <except=name_1,name_2, ...>
```

Example 1

This example uses the .BIASCHK statement to monitor an expression:

```
.biaschk 'v(1)' min = 'v(2)*2' simulation= op
```

Example 2

This example uses the .BIASCHK statement to monitor an element and model type:

```
.biaschk nmos terminal1 = vg terminal2 = vs  
simulation = tr name = m1
```

In this example, terminal1 and terminal2 are the terminals between which you want to check.

Description

The .BIASCHK statement can monitor the voltage bias, current, device-size, expression and region during analysis, and reports:

- Element name
- Time
- Terminals

- Bias that exceeds the limit
- Number of times the bias exceeds the limit for an element

HSPICE saves the information as both a warning and a BIASCHK summary in the **.lis* file or a file you define in the `.OPTION BIASFILE` command option. You can use this command only for active elements, capacitors, and subcircuits.

If a model name, referenced in an active element statement, contains a period (`.`), then `.BIASCHK` reports an error. This occurs because it is unclear whether a reference such as `x.123` is a model name or a sub-circuit name (123 model in the `x` subcircuit).

More than one simulation type or all simulation types can be set in one `.BIASCHK` command, and more than one region can be set in one `.BIASCHK` command.

Instance (element) and model names can contain wildcards, either `"?"` (stands for one character) or `"*"` (stands for 0 or more characters).

If you do not set *name* and *mname*, HSPICE checks all elements of this type for bias voltage (you must include type in the biaschk card). However, if `type = subckt`, at least one *name* or *mname* must be specified in the `.BIASCHK` command; otherwise, a warning message is issued and this command ignored.

After a simulation that uses the `.BIASCHK` command runs, HSPICE outputs a results summary including the element name, time, terminals, model name, and the number of times the bias exceeded the limit for a specified element.

Interactions with Other Options

If you set `.OPTION BIAWARN` to 1, HSPICE immediately outputs a warning message that includes the element name, time, terminals and model name when the limit is exceeded during the analysis you define. If you set the `autostop` keyword, HSPICE automatically stops at that situation.

If you set `.OPTION BIASFILE`, HSPICE outputs the summary into a file you define in the biasfile. Otherwise, HSPICE outputs the summary to a **.lis* file.

2: Commands in HSPICE Netlists

.BIASCHK

Argument	Definition
<i>type</i>	Element type to check. MOS (C, BJT, ...) For a monitor, <i>type</i> can be DIODE, BIPOLAR, BJT, JFET, MOS, NMOS, PMOS, C, or SUBCKT. When used with REGION, <i>type</i> can be MOS only.
<i>terminal 1</i> , <i>terminal 2</i>	Terminals, between which HSPICE checks (that is, checks between <i>terminal1</i> and <i>terminal2</i>): <ul style="list-style-type: none">• For MOS level 57: <i>nd, ng, ns, ne, np, n6</i>• For MOS level 58: <i>nd, ngf, ns, ngb</i>• For MOS level 59: <i>nd, ng, ns, ne, np</i>• For other MOS level: <i>nd, ng, ns, nb</i>• For capacitor: <i>n1, n2</i>• For diode: <i>np, nn</i>• For bipolar: <i>nc, nb, ne, ns</i>• For JFET: <i>nd, ng, ns, nb</i> For <i>type</i> = <i>subckt</i> , the terminal names are those pins defined by the subcircuit definition of <i>mname</i> .
<i>limit</i>	Biaschk limit that you define. Reports an error if the bias voltage (between appointed terminals of appointed elements and models) is larger than the limit.
<i>noise</i>	Biaschk noise that you define. The default is 0.1v. Noise-filter some of the results (the local maximum bias voltage, that is larger than the limit). The next local max replaces the local max, if all of the following conditions are satisfied: <pre>local_max-local_min<noise>. next local_max-local_min<noise>.</pre> This local max is smaller than the next local max. For a parasitic diode, HSPICE ignores the smaller local max biased voltage, and does not output this voltage. To disable this feature, set the noise detection level to 0.
<i>max</i>	Maximum value.

Argument	Definition
<code>min</code>	Minimum value.
<code>name</code>	<p>Element name to check. If <code>name</code> and <code>mname</code> are not both set for the element type, the elements of this type are all checked. You can define more than one element name in keyword <code>name</code> with a comma (,) delimiter.</p> <p>If doing bias checking for subcircuits:</p> <ul style="list-style-type: none"> • When both <code>mname</code> and <code>name</code> are defined while multiple <code>name</code> definitions are allowed, if a name is also an instance of <code>mname</code>, then only those names are checked, others will be ignored. • This command is ignored if no <code>name</code> is an instance of <code>mname</code>. • For <code>name</code> definitions which are not of the type defined in <code>mname</code> will be ignored. • If a <code>mname</code> is not defined, the subcircuit type is determined by the first <code>name</code> definition.
<code>mname</code>	<p>Model name. HSPICE checks elements of the model for bias. If you define <code>mname</code>, then HSPICE checks all devices of this model. You can define more than one model name in keyword <code>mname</code> with the comma (,) delimiter.</p> <p>If <code>mname</code> and <code>name</code> are not both set for the element type, the elements of this type are all checked.</p> <p>If doing bias checking for subcircuits:</p> <ul style="list-style-type: none"> • Once there is one and only one <code>mname</code> defined, the terminal names for this <code>.command</code> are those pins defined by the <code>subckt</code> definition of <code>mname</code>. • Multiple <code>mname</code> definitions are not allowed. • Wildcarding is not supported for <code>mname</code>. • If only <code>mname</code> is specified in <code>subckt</code> bias check, then all subcircuits will be checked.
<code>region</code>	<p>Values can be <code>cutoff</code>, <code>linear</code>, or <code>saturation</code>. HSPICE monitors when the MOS device, defined in the <code>.BIASCHK</code> command, enters and leaves the specified region (such as <code>cutoff</code>).</p>
<code>simulation</code>	<p>The simulation type you want to monitor. You can specify <code>op</code>, <code>dc</code>, <code>tr</code> (transient), and <code>all</code> (<code>op</code>, <code>dc</code>, and <code>tr</code>). The <code>tr</code> option is the default simulation type.</p>

2: Commands in HSPICE Netlists

.BIASCHK

Argument	Definition
monitor	The kind of value you want to monitor. You can specify v (voltage), i (current), w, and l (device size) for the element type. This parameter is not used for an expression-type monitor.
tstart	The biaschk start time during transient analysis. The default is 0.
tstop	The biaschk end time during transient analysis. The analysis ends on its own by default if you do not set this parameter.
autostop	When set, HSPICE supports an autostop for a biaschk card so that it can report error messages and stop the simulation immediately.
except	Lets you specify the element or instance that you do not want to bias check.

See Also

[.OPTION BIASFILE](#)
[.OPTION BIAWARN](#)

.CONNECT

Syntax

```
.CONNECT node1 node2
```

Example

```
...  
.subckt eye_diagram node1 node2 ...  
.connect node1 node2  
...  
.ends
```

This is now the same as the following:

```
...  
.subckt eye_diagram node1 node1 ...  
...  
.ends  
...
```

HSPICE reports the following error message:

```
**error**: subcircuit definition duplicates node node1
```

To apply any HSPICE statement to *node2*, apply it to *node1* instead. Then, to change the netlist construction to recognize *node2*, use a `.ALTER` statement.

Example 2

```
*example for .connect  
vcc 0 cc 5v  
r1 0 1 5k  
r2 1 cc 5k  
.tran 1n 10n  
.print i(vcc) v(1)  
.alter  
.connect cc 1  
.end
```

The first `.TRAN` simulation includes two resistors. Later simulations have only one resistor, because `r2` is shorted by connecting `cc` with `1`. `v(1)` does not print out, but `v(cc)` prints out instead.

Use multiple `.CONNECT` statements to connect several nodes together.

Example 3

```
.CONNECT node1 node2  
.CONNECT node2 node3
```

2: Commands in HSPICE Netlists

.CONNECT

This example connects both *node2* and *node3* to *node1*. All connected nodes must be in the same subcircuit or all in the main circuit. The first HSPICE simulation evaluates only *node1*; *node2*, and *node3* are the same node as *node1*. Use `.ALTER` statements to simulate *node2* and *node3*.

If you set `.OPTION NODE`, then HSPICE prints out a node connection table.

Description

The `.CONNECT` statement connects two nodes in your HSPICE netlist so that simulation evaluates 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.

If you connect *node2* to *node1*, HSPICE does not recognize *node2* at all. You also cannot use this statement in HSPICE RF.

Argument	Definition
<i>node1</i>	Name of the first of two nodes to connect to each other.
<i>node2</i>	Name of the second of two nodes to connect to each other. The first node replaces this node in the simulation.

See Also

[.ALTER](#)
[.CONNECT](#)
[.OPTION NODE](#)

.DATA

Syntax

Inline statement:

```
.DATA datanm pnam1 <pnam2 pnam3 ... pnamxxx >  
+ pval1<pval2 pval3 ... pvalxxx>  
+ pval1' <pval2' pval3' ... pvalxxx'>  
.ENDDATA
```

External File statement for concatenated data files:

```
.DATA datanm MER  
+ FILE = 'filename1' pname1 = colnum <pname2 = colnum ...>  
+ <FILE = 'filename2' pname1 = colnum  
+ <pname2 = colnum ...>> ... <OUT = 'fileout'>  
.ENDDATA
```

Column Laminated statement:

```
.DATA datanm LAM  
+ FILE = 'filename1' pname1 = colnum  
+ <pname2 = colnum ...>  
+ <FILE = 'filename2' pname1 = colnum  
+ <pname2 = colnum ...>> ... <OUT = 'fileout'>  
.ENDDATA
```

Example

```
* Inline .DATA statement  
.TRAN 1n 100n SWEEP DATA = devinf  
.AC DEC 10 1hz 10khz SWEEP DATA = devinf  
.DC TEMP -55 125 10 SWEEP DATA = devinf  
.DATA devinf width length thresh cap  
+ 50u 30u 1.2v 1.2pf  
+ 25u 15u 1.0v 0.8pf  
+ 5u 2u 0.7v 0.6pf  
.ENDDATA
```

HSPICE or HSPICE RF performs the above analyses for each set of parameter values defined in the .DATA statement. For example, the program first uses the width = 50u, length = 30u, thresh = 1.2v, and cap = 1.2pf parameters to perform .TRAN, .AC, and .DC analyses.

HSPICE or HSPICE RF then repeats the analyses for width = 25u, length = 15u, thresh = 1.0v, and cap = 0.8pf, and again for the values on each subsequent line in the .DATA block.

2: Commands in HSPICE Netlists

.DATA

Example 2

```
* .DATA as the inner sweep
M1 1 2 3 0 N    W = 50u    L = LN
  VGS 2 0 0.0v
  VBS 3 0 VBS
  VDS 1 0 VDS
  .PARAM VDS = 0 VBS = 0 L = 1.0u
  .DC DATA = vdot
  .DATA vdot
    VBS    VDS    L
    0      0.1    1.5u
    0      0.1    1.0u
    0      0.1    0.8u
   -1     0.1    1.0u
   -2     0.1    1.0u
   -3     0.1    1.0u
    0     1.0    1.0u
    0     5.0    1.0u
  .ENDDATA
```

This example performs a DC sweep analysis for each set of VBS, VDS, and L parameters in the .DATA vdot block. That is, HSPICE or HSPICE RF runs eight DC analyses, one for each line of parameter values in the .DATA block.

Example 3

```
* .DATA as the outer sweep
  .PARAM W1 = 50u W2 = 50u L = 1u CAP = 0
  .TRAN 1n 100n SWEEP DATA = d1
  .DATA d1
    W1    W2    L    CAP
    50u   40u   1.0u  1.2pf
    25u   20u   0.8u  0.9pf
  .ENDDATA
```

In this example:

- The default start time for the .TRAN analysis is 0.
- The time increment is 1 ns.
- The stop time is 100 ns.

This results in transient analyses at every time value from 0 to 100 ns in steps of 1 ns by using the first set of parameter values in the .DATA d1 block. Then HSPICE or HSPICE RF reads the next set of parameter values, and performs another 100 transient analyses. It sweeps time from 0 to 100 ns in 1 ns steps. The outer sweep is time, and the inner sweep varies the parameter values. HSPICE or HSPICE RF performs two hundred analyses: 100 time increments, times 2 sets of parameter values.

Example 4

```
* External File .DATA for concatenated data files
.DATA datanm MER
  + FILE = filename1 pname1 = colnum
  + <pname2 = colnum ...>
  + <FILE = filename2 pname1 = colnum
  + <pname2 = colnum ...>>
  + ...
  + <OUT = fileout>
.ENDDATA
```

Example 5

If you concatenate the three files (*file1*, *file2*, and *file3*).

```
file1  file2  file3
a a a  b b b  c c c
a a a  b b b  c c c
a a a
```

The data appears as follows:

```
a a a
a a a
a a a
b b b
b b b
c c c
c c c
```

The number of lines (rows) of data in each file does not need to be the same. The simulator assumes that the associated parameter of each column of the A file is the same as each column of the other files.

The .DATA statement for this example is:

```
* External File .DATA statement
.DATA inputdata MER
  FILE = 'file1' p1 = 1 p2 = 3 p3 = 4
  FILE = 'file2' p1 = 1
  FILE = 'file3'
.ENDDATA
```

This listing concatenates *file1*, *file2*, and *file3* to form the *inputdata* dataset. The data in *file1* is at the top of the file, followed by the data in *file2*, and *file3*. The *inputdata* in the .DATA statement references the *dataname* specified in either the .DC, .AC, or .TRAN analysis statements. The parameter fields specify the column that contains the parameters (you must already have defined the parameter names in .PARAM statements). For

2: Commands in HSPICE Netlists

.DATA

example, the values for the *p1* parameter are in column 1 of *file1* and *file2*. The values for the *p2* parameter are in column 3 of *file1*.

For data files with fewer columns than others, HSPICE or HSPICE RF assigns values of zero to the missing parameters.

Example 6

Three files (*D*, *E*, and *F*) contain the following columns of data:

File D	File E	File F
d1 d2 d3	e4 e5	f6
d1 d2 d3	e4 e5	f6
d1 d2 d3	e4 e5	f6

The laminated data appears as follows:

d1 d2 d3	e4 e5	f6
d1 d2 d3	e4 e5	f6
d1 d2 d3	e4 e5	f6

The number of columns of data does not need to be the same in the three files.

The number of lines (rows) of data in each file does not need to be the same. HSPICE interprets missing data points as zero. HSPICE RF does not support column-laminated data files.

The `.DATA` statement for this example is:

```
* Column-Laminated .DATA statement
.DATA dataname LAM
  FILE = 'file1' p1 = 1 p2 = 2 p3 = 3
  FILE = 'file2' p4 = 1 p5 = 2
  OUT = 'fileout'
.ENDDATA
```

This listing laminates columns from *file1*, and *file2*, into the *fileout* output file. Columns one, two, and three of *file1*, and columns one and two of *file2*, are designated as the columns to place in the output file. You can specify up to 10 files per `.DATA` statement.

If you run HSPICE on a different machine than the one on which the input data files reside (such as when you work over a network), use full path names instead of aliases. Aliases might have different definitions on different machines.

Description

Data-driven analysis syntax requires a `.DATA` statement, and an analysis statement that contains a `DATA = dataname` keyword.

You can use the `.DATA` statement to concatenate or column-laminate data sets to optimize measured I-V, C-V, transient or S parameter data.

You can also use the `.DATA` statement for a first or second sweep variable when you characterize cells, and test worst-case corners. Simulation reads data measured in a lab, such as transistor I-V data, one transistor at a time in an outer analysis loop. Within the outer loop, the analysis reads data for each transistor (IDS curve, GDS curve, and so on), one curve at a time in an inner analysis loop.

The `.DATA` statement specifies parameters that change values, and the sets of values to assign during each simulation. The required simulations run as an internal loop. This bypasses reading-in the netlist and setting-up the simulation, which saves computing time. In internal loop simulation, you can also plot simulation results against each other, and print them in a single output.

You can enter any number of parameters in a `.DATA` block. The `.AC`, `.DC`, and `.TRAN` statements can use external and inline data provided in `.DATA` statements. The number of data values per line does not need to correspond to the number of parameters. For example, you do not need to enter 20 values on each line in the `.DATA` block, if each simulation pass requires 20 parameters: the program reads 20 values on each pass, no matter how you format the values.

Each `.DATA` statement can contain up to 50 parameters. If you need more than 50 parameters in a single `.DATA` statement, place 50 or fewer parameters in the `.DATA` statement, and use `.ALTER` statements for the remaining parameters.

HSPICE or HSPICE RF refers to `.DATA` statements by their datanames so each dataname must be unique. HSPICE or HSPICE RF support three `.DATA` statement formats:

- Inline data, which is parameter data, listed in a `.DATA` statement block. The `datanm` parameter in a `.DC`, `.AC`, or `.TRAN` analysis statement, calls this statement. The number of parameters that HSPICE or HSPICE RF reads, determines the number of columns of data. The physical number of data numbers per line does not need to correspond to the number of parameters. For example, if the simulation needs 20 parameters, you do not need 20 numbers per line.
- Data that is concatenated from external files. Concatenated data files are files with the same number of columns, placed one after another.

2: Commands in HSPICE Netlists

.DATA

- Data that is column-laminated from external files. Column lamination means that the columns of files with the same number of rows, are arranged side-by-side.

To use external files with the `.DATA` format:

- Use the `MER` and `LAM` keywords to tell HSPICE or HSPICE RF to expect external file data, rather than inline data.
- Use the `FILE` keyword to specify the external filename.
- You can use simple file names, such as `out.dat`, without the single or double quotes (`'` or `"`), but use the quotes when file names start with numbers, such as `"1234.dat"`.
- File names are case sensitive on Unix systems.

For data-driven analysis, specify the start time (time 0) in the analysis statement so analysis correctly calculates the stop time.

The following shows how different types of analysis use `.DATA` statements.

Operating point:

```
.DC DATA = dataname
```

DC sweep:

```
.DC vin 1 5 .25 SWEEP DATA = dataname
```

AC sweep:

```
.AC dec 10 100 10meg SWEEP DATA = dataname
```

TRAN sweep:

```
.TRAN 1n 10n SWEEP DATA = dataname
```

Argument	Definition
<i>colnum</i>	Column number in the data file for the parameter value. The column does not need to be the same between files.
<i>datanm</i>	Data name, referenced in the <code>.TRAN</code> , <code>.DC</code> , or <code>.AC</code> statement.
<i>filenamei</i>	Data file to read. HSPICE or HSPICE RF concatenates files in the order they appear in the <code>.DATA</code> statement. You can specify up to 10 files.

Argument	Definition
<i>fileouti</i>	Data file name, where simulation writes concatenated data. This file contains the full syntax for an inline <code>.DATA</code> statement, and can replace the <code>.DATA</code> statement that created it in the netlist. You can output the file, and use it to generate one data file from many.
<i>LAM</i>	Column-laminated (parallel merging) data files to use.
<i>MER</i>	Concatenated (series merging) data files to use.
<i>pnamei</i>	Parameter names, used for source value, element value, device size, model parameter value, and so on. You must declare these names in a <code>.PARAM</code> statement.
<i>pvali</i>	Parameter value.

See Also

[.AC](#)
[.DC](#)
[.ENDDATA](#)
[.PARAM](#)
[.TRAN](#)

.DC

Syntax

Sweep or Parameterized Sweep:

```
.DC var1 START = start1 STOP = stop1 STEP = incr1
```

```
.DC var1 START = <param_expr1>  
+ STOP = <param_expr2> STEP = <param_expr3>
```

```
.DC var1 start1 stop1 incr1  
+ <SWEEP var2 type np start2 stop2>
```

```
.DC var1 start1 stop1 incr1 <var2 start2 stop2 incr2>
```

HSPICE supports the start and stop syntax; HSPICE RF does not.

Data-Driven Sweep:

```
.DC var1 type np start1 stop1 <SWEEP DATA = datanm>
```

```
.DC DATA = datanm<SWEEP var2 start2 stop2 incr2>
```

```
.DC DATA = datanm
```

HSPICE supports the start and stop syntax; HSPICE RF does not.

Monte Carlo:

```
.DC var1 type np start1 stop1 <SWEEP MONTE = val>
```

```
.DC MONTE = val
```

HSPICE supports Monte Carlo analysis; HSPICE RF does not.

Optimization:

```
.DC DATA = datanm OPTIMIZE = opt_par_fun  
+ RESULTS = measnames MODEL = optmod
```

```
.DC var1 start1 stop1 SWEEP OPTIMIZE = OPTxxx  
+ RESULTS = measname MODEL = optmod
```

Example 1

```
.DC VIN 0.25 5.0 0.25
```

This example sweeps the value of the `VIN` voltage source, from 0.25 volts to 5.0 volts in increments of 0.25 volts.

Example 2

```
.DC VDS 0 10 0.5 VGS 0 5 1
```

This example sweeps the drain-to-source voltage, from 0 to 10 V in 0.5 V increments, at VGS values of 0, 1, 2, 3, 4, and 5 V.

Example 3

```
.DC TEMP -55 125 10
```

This example starts a DC analysis of the circuit, from -55°C to 125°C in 10°C increments.

Example 4

```
.DC TEMP POI 5 0 30 50 100 125
```

This script runs a DC analysis, at five temperatures: 0, 30, 50, 100, and 125°C.

Example 5

```
.DC xval 1k 10k .5k SWEEP TEMP LIN 5 25 125
```

This example runs a DC analysis on the circuit, at each temperature value. The temperatures result from a linear temperature sweep, from 25°C to 125°C (five points), which sweeps a resistor value named `xval`, from 1 k to 10 k in 0.5 k increments.

Example 6

```
.DC DATA = datanm SWEEP par1 DEC 10 1k 100k
```

This example specifies a sweep of the `par1` value, from 1 k to 100 k in increments of 10 points per decade.

Example 7

```
.DC par1 DEC 10 1k 100k SWEEP DATA = datanm
```

This example also requests a DC analysis, at specified parameters in the `.DATA datanm` statement. It also sweeps the `par1` parameter, from 1k to 100k in increments of 10 points per decade.

2: Commands in HSPICE Netlists

.DC

Example 8

```
.DC par1 DEC 10 1k 100k SWEEP MONTE = 30
```

This example invokes a DC sweep of the *par1* parameter from 1k to 100k by 10 points per decade by using 30 randomly generated (Monte Carlo) values. HSPICE supports Monte Carlo analysis; HSPICE RF does not.

Example 9

```
* Schmitt Trigger Example
*file: bjtschmt.sp    bipolar schmitt trigger
.OPTION post = 2
vcc 6 0 dc 12
vin 1 0 dc 0 pwl(0,0 2.5u,12 5u,0)
cbl 2 4 .1pf
rc1 6 2 1k
rc2 6 5 1k
rb1 2 4 5.6k
rb2 4 0 4.7k
re 3 0 .47k
diode 0 1 dmod
q1 2 1 3 bmod 1 ic = 0,8
q2 5 4 3 bmod 1 ic = .5,0.2
.dc vin 0,12,.1
.model dmod d is = 1e-15 rs = 10
.model bmod npn is = 1e-15 bf = 80 tf = 1n
+ cjc = 2pf cje = 1pf rc = 50 rb = 100 vaf = 200
.plot v(1) v(5)
.graph dc model = schmittplot input = v(1)
+ output = v(5) 4.0 5.0
.model schmittplot plot xscal = 1 yscal = 1
+ xmin = .5u xmax = 1.2u
.end
```

Example 10

```
.DC par1 DEC 10 1k 100k SWEEP MONTE = 10 firstrun = 11
```

This example invokes a DC sweep of the *par1* parameter from 1k to 100k by 10 points per decade and uses 10 randomly generated (Monte Carlo) values from 11th to 20th trials.

Example 11

```
.DC par1 DEC 10 1k 100k SWEEP MONTE = list(10 20:30 35:40 50)
```

This example invokes a DC sweep of the *par1* parameter from 1k to 100k by 10 points per decade and a Monte Carlo analysis at the 10th trial, then from the 20th to the 30th, followed by the 35th to 40th trials, and finally at the 50th trial.

Description

You can use the `.DC` statement in DC analysis, to:

- Sweep any parameter value (HSPICE and HSPICE RF).
- Sweep any source value (HSPICE and HSPICE RF).
- Sweep temperature range (HSPICE and HSPICE RF).
- Perform a DC Monte Carlo (random sweep) analysis (HSPICE only; not supported in HSPICE RF).
- Perform a data-driven sweep (HSPICE and HSPICE RF).
- Perform a DC circuit optimization for a data-driven sweep (HSPICE and HSPICE RF).
- Perform a DC circuit optimization by using start and stop (HSPICE only; not supported in HSPICE RF).
- Perform a DC model characterization (HSPICE only; not supported in HSPICE RF).

The format for the `.DC` statement depends on the application that uses it.

Argument	Definition
<code>DATA = <i>datanm</i></code>	<i>Datanm</i> is the reference name of a <code>.DATA</code> statement.
<code>incr1 ...</code>	Voltage, current, element, or model parameters; or temperature increments.
<code>MODEL</code>	Specifies the optimization reference name. The <code>.MODEL OPT</code> statement uses this name in an optimization analysis
<code>MONTE = <i>val</i></code>	<i>val</i> is the number of randomly-generated values, which you can use to select parameters from a distribution. The distribution can be <i>Gaussian</i> , <i>Uniform</i> , or <i>Random Limit</i> .
<code>np</code>	Number of points per decade or per octave or just number of points, based on which keyword precedes it.
<code>OPTIMIZE</code>	Specifies the parameter reference name, used for optimization in the <code>.PARAM</code> statement
<code>RESULTS</code>	Measure name used for optimization in the <code>.MEASURE</code> statement

2: Commands in HSPICE Netlists

.DC

Argument	Definition
<i>start1 ...</i>	Starting voltage, current, element, or model parameters; or temperature values. If you use the POI (list of points) variation type, specify a list of parameter values, instead of <i>start stop</i> . HSPICE supports the <i>start</i> and <i>stop</i> syntax; HSPICE RF does not.
<i>stop1 ...</i>	Final voltage, current, any element, model parameter, or temperature values.
SWEEP	Indicates that a second sweep has a different type of variation (DEC, OCT, LIN, POI, or DATA statement; or MONTE = <i>val</i>)
TEMP	Indicates a temperature sweep.
type	Can be any of the following keywords: <ul style="list-style-type: none">• DEC — decade variation• OCT — octave variation• LIN — linear variation• POI — list of points
<i>var1 ...</i>	<ul style="list-style-type: none">• Name of an independent voltage or current source, or• Name of any element or model parameter, or• TEMP keyword (indicating a temperature sweep). HSPICE supports a source value sweep, which refers to the source name (SPICE style). However, if you select a parameter sweep, a .DATA statement, and a temperature sweep, then you must select a parameter name for the source value. A later .DC statement must refer to this name. The parameter name must not start with V, I, or TEMP. In HSPICE RF, you can run a parameter sweep around a single analysis, but the parameter sweep cannot change any .OPTION value.
firstrun	The <i>val</i> value specifies the number of Monte Carlo iterations to perform. The <i>firstrun</i> value specifies the desired number of iterations. HSPICE runs from num1 to num1+ <i>val</i> -1.
list	The iterations at which HSPICE performs a Monte Carlo analysis. You can write more than one number after <i>list</i> . The colon represents "from ... to ...". Specifying only one number makes HSPICE run at only the specified point.

See Also

[.MODEL](#)
[.OPTION DCIC](#)
[.PARAM](#)

.DCMATCH

Syntax

```
.DCMATCH OUTVAR <THRESHOLD=T> <FILE=string>
+ <PERTURBATION=P> <INTERVAL=Int>
```

Example 1

```
.DCmatch V(9) V(4,2) I(VCC)
```

HSPICE reports DCmatch variations on the voltage of node 9, the voltage difference between nodes 4 and 2, and on the current through the source VCC.

Example 2

```
.DC XVal Start=1K Stop=9K Step=1K
.DCMATCH V(vcc) interval=3
```

The variable `XVal` is being sweep in the `.DC` command. It takes nine values in sequence from 1k to 9k in increments of 1k. Tabular output for the `.DCMATCH` command is only generated for the set `XVal={1k, 4k, 7k, 9k}`.

Description

You use this command to calculate the effects of local variations in device characteristics on the DC solution of a circuit.

You can perform only one DCMATCH analysis per simulation. Only the last `.DCMATCH` statement is used in case more than one is present. The others are discarded with warnings.

.

Argument	Definition
OUTVAR	Valid node voltages, the difference between node pairs or branch currents.
THRESHOLD	Report devices with a relative contribution above Threshold in the summary table. <ul style="list-style-type: none"> T=0: reports results for all devices T<0: suppresses table output; however, individual results are still available through <code>.PROBE</code> or <code>.MEASURE</code> statements. The upper limit for <code>T</code> is 1, but at least 10 devices are reported, or all if there are less than 10. Default value is 0.01.

Argument	Definition
FILE	Valid file name for the output tables. Default is <code>basename.dm#</code> where “#” is the usual sequence number for HSPICE output files.
PERTURBATION	Indicates that perturbations of P standard deviation will be used in calculating the finite difference approximations to device derivatives. The valid range for P is 0.01 to 6, with a default value of 2.
INTERVAL	Applies only if a DC sweep is specified. Int is a positive integer. A summary is printed at the first sweep point, then for each subsequent increment of Int , and then, if not already printed, at the final sweep point. Only single sweeps are supported.

See Also

- [.DC](#)
- [.MEASURE \(Average, RMS, MIN, MAX, INTEG, and PP\)](#)
- [.MEASURE \(Equation Evaluation/ Arithmetic Expression\)](#)
- [.MEASURE \(FIND and WHEN\)](#)
- [.PROBE](#)

.DCVOLT

Syntax

```
.DCVOLT V(node1) = val1 V(node2) = val2 ...
```

```
.DCVOLT V node1 val1 <node2 val2 ...>
```

Example

```
.DCVOLT 11 5 4 -5 2 2.2
```

Description

Use the `.IC` statement or the `.DCVOLT` statement to set transient initial conditions in HSPICE, but not in HSPICE RF. How it initializes depends on whether the `.TRAN` analysis statement includes the `UIC` parameter.

Note: In HSPICE RF, `.IC` is always set to OFF.

If you specify the `UIC` parameter in the `.TRAN` statement, HSPICE does not calculate the initial DC operating point, but directly enters transient analysis. Transient analysis uses the `.IC` initialization values as part of the solution for timepoint zero (calculating the zero timepoint applies a fixed equivalent voltage source). The `.IC` statement is equivalent to specifying the `IC` parameter on each element statement, but is more convenient. You can still specify the `IC` parameter, but it does not have precedence over values set in the `.IC` statement.

If you do *not* specify the `UIC` parameter in the `.TRAN` statement, HSPICE computes the DC operating point solution, before the transient analysis. The node voltages that you specify in the `.IC` statement are fixed to determine the DC operating point. HSPICE RF does not output node voltage from operating point (`.OP`), if time (`t`) < 0. Transient analysis releases the initialized nodes to calculate the second and later time points.

Argument	Definition
val1 ...	Specifies voltages. The significance of these voltages depends on whether you specify the <code>UIC</code> parameter in the <code>.TRAN</code> statement.
node1 ...	Node numbers or names can include full paths or circuit numbers.

See Also

[.IC](#)
[.TRAN](#)

2: Commands in HSPICE Netlists

.DEL LIB

.DEL LIB

Syntax

```
.DEL LIB '<filepath>filename' entryname
```

```
.DEL LIB libnumber entryname
```

Example 1

This example uses an .ALTER block so it applies to HSPICE but not to HSPICE RF.

```
FILE1: ALTER1 TEST CMOS INVERTER
.OPTION ACCT LIST
.TEMP 125
.PARAM WVAL = 15U VDD = 5
*
.OP
.DC VIN 0 5 0.1
.PLOT DC V(3) V(2)
*
VDD 1 0 VDD
VIN 2 0
*
M1 3 2 1 1 P 6U 15U
M2 3 2 0 0 N 6U W = WVAL
*
.LIB 'MOS.LIB' NORMAL
.ALTER
.DEL LIB 'MOS.LIB' NORMAL          $removes LIB from memory
$PROTECTION
.PROT                               $protect statements
    $below .PROT
.LIB 'MOS.LIB' FAST                $get fast model library
.UNPROT
.ALTER
.OPTION NOMOD OPTS                $suppress printing model
    $parameters and print the
    $option summary
.TEMP -50 0 50                    $run with different
    $temperatures
.PARAM WVAL = 100U VDD = 5.5      $change the parameters
VDD 1 0 5.5                        $using VDD 1 0 5.5 to
    $change the power supply
    $VDD value doesn't
    $work
VIN 2 0 PWL 0NS 0 2NS 5 4NS 0 5NS 5
    $change the input
    $source
```

```
.OP VOL      $node voltage table of
             $operating points
.TRAN 1NS 5NS      $run with transient
             $also
M2 3 2 0 0 N 6U WVAL      $change channel width
.MEAS SW2 TRIG V(3) VAL = 2.5 RISE = 1 TARG V(3)
+ VAL = VDD CROSS = 2      $measure output
*
.END
```

Example 1 calculates a DC transfer function for a CMOS inverter.

1. First, HSPICE simulates the device by using the NORMAL inverter model from the MOS.LIB library.
2. Using the .ALTER block and the .LIB command, HSPICE substitutes a faster CMOS inverter, FAST for NORMAL.
3. HSPICE then resimulates the circuit.
4. Using the second .ALTER block, HSPICE executes DC transfer analysis simulations at three different temperatures, and with an n-channel width of 100 mm, instead of 15 mm.
5. HSPICE also runs a transient analysis in the second .ALTER block. Use the .MEASURE statement to measure the rise time of the inverter.

Example 2

This example uses an .ALTER block so it applies to HSPICE but not to HSPICE RF.

```
FILE2: ALTER2.SP CMOS INVERTER USING SUBCIRCUIT
.OPTION LIST ACCT
.MACRO INV 1 2 3
M1 3 2 1 1 P 6U 15U
M2 3 2 0 0 N 6U 8U
.LIB 'MOS.LIB' NORMAL
.EOM INV
XINV 1 2 3 INV
VDD 1 0 5
VIN 2 0
.DC VIN 0 5 0. 1
.PLOT V(3) V(2)
.ALTER
.DEL LIB 'MOS.LIB' NORMAL
.TF V(3) VIN      $DC small-signal transfer
                 $function
*
.MACRO INV 1 2 3      $change data within
```

2: Commands in HSPICE Netlists

.DEL LIB

```
        $subcircuit def
M1 4 2 1 1 P 100U 100U          $change channel
        $length,width,also
        $topology
M2 4 2 0 0 N 6U 8U            $change topology
R4 4 3 100                    $add the new element
C3 3 0 10P                    $add the new element
.LIB 'MOS.LIB' SLOW           $set slow model library
$.INC 'MOS2.DAT'              $not allowed to be used
        $inside subcircuit, allowed
        $outside subcircuit
.EOM INV
.END
```

In this example, the `.ALTER` block adds a resistor and capacitor network to the circuit. The network connects to the output of the inverter, and HSPICE simulates a DC small-signal transfer function.

Description

Use the `.DEL LIB` statement to remove library data from memory. The next time you run a simulation, the `.DEL LIB` statement removes the `.LIB` call statement with the same library number and entry name, from memory. You can then use a `.LIB` statement to replace the deleted library.

You can use the `.DEL LIB` statement with the `.ALTER` statement. HSPICE RF does not support the `.ALTER` statement.

Argument	Definition
<i>entryname</i>	Entry name, used in the library call statement to delete.
<i>filename</i>	Name of a file to delete from the data file. The file path, plus the file name, can be up to 256 characters long. You can use any file name that is valid for the operating system that you use. Enclose the file path and file name in single or double quote marks.
<i>filepath</i>	Path name of a file, if the operating system supports tree-structured directories.
<i>libnumber</i>	Library number, used in the library call statement to delete.

See Also

[.ALTER](#)
[.LIB](#)

.DISTO

Syntax

```
.DISTO Rload <inter <skw2 <refpwr <spwf>>>>
```

Example

```
.DISTO RL 2 0.95 1.0E-3 0.75
```

Description

The `.DISTO` statement computes the distortion characteristics of the circuit in an AC small-signal, sinusoidal, steady-state analysis. You can use the `.DISTO` statement in HSPICE, but not in HSPICE RF.

The program computes and reports five distortion measures at the specified load resistor. The analysis assumes that the input uses one or two signal frequencies.

- HSPICE uses the first frequency (F1, the nominal analysis frequency) to calculate harmonic distortion. The `.AC` statement frequency-sweep sets it.
- HSPICE uses the optional second input frequency (F2) to calculate intermodulation distortion. To set it implicitly, specify the `skw2` parameter, which is the F2/F1 ratio

HSPICE performs only one distortion analysis per simulation. HSPICE RF does not support the `.DISTO` statement. If your design contains more than one `.DISTO` statement, HSPICE runs only the last statement. The `.DISTO` statement calculates distortions for diodes, BJTs (levels 1, 2, 3, and 4), and MOSFETs (Level49 and Level53, Version 3.22).

You can use the `.DISTO` command only with the `.AC` command.

.

Argument	Definition
<i>Rload</i>	The resistor element name of the output load resistor, into which the output power feeds.
<i>refpwr</i>	Reference power level, used to compute the distortion products. If you omit <i>refpwr</i> , the default value is 1mW, measured in decibels magnitude (dbM). The value must be $\geq 1e-10$.

Argument	Definition
<i>skw2</i>	Ratio of the second frequency (F2) to the nominal analysis frequency (F1) in the range $1e-3 < skw2 < 0.999$. If you omit <i>skw2</i> , the default value is 0.9.
<i>spwf</i>	Amplitude of the second frequency (F2). The value must be $\geq 1e-3$. Default = 1.0.
<i>inter</i>	<p>Interval at which HSPICE prints a distortion-measure summary. Specifies a number of frequency points in the AC sweep (see the <i>np</i> parameter in the .AC command).</p> <ul style="list-style-type: none"> If you omit <i>inter</i> or set it to zero, HSPICE does not print a summary. To print or plot the distortion measures, use the <code>.PRINT</code> or <code>.PLOT</code> statement. If you set <i>inter</i> to 1 or higher, HSPICE prints a summary of the first frequency, and of each subsequent inter-frequency increment. To obtain a summary printout for only the first and last frequencies, set <i>inter</i> equal to the total number of increments needed to reach <i>fstop</i> in the <code>.AC</code> statement. For a summary printout of only the first frequency, set <i>inter</i> to greater than the total number of increments required to reach <i>fstop</i>. <p>HSPICE prints an extensive summary from the distortion analysis for each frequency listed. Use the <i>inter</i> parameter in the <code>.DISTO</code> statement to limit the amount of output generated.</p>

.DISTO Value	Description
<i>DIM2</i>	Intermodulation distortion, first difference. Relative magnitude and phase of the frequency component (F1 - F2).
<i>DIM3</i>	Intermodulation distortion, second difference. The relative magnitude and phase of the frequency component (2 · F1 - F2).
<i>HD2</i>	Second-order harmonic distortion. Relative magnitude and phase of the frequency component 2 · F1 (ignores F2).

2: Commands in HSPICE Netlists

.DISTO

.DISTO Value	Description
<i>HD3</i>	Third-order harmonic distortion. Relative magnitude and phase of the frequency component $3 \cdot F1$ (ignores $F2$).
<i>SIM2</i>	Intermodulation distortion, sum. Relative magnitude and phase of the frequency component $(F1 + F2)$.

See Also

[.AC](#)

.DOUT

Syntax

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

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

The first syntax specifies a single threshold voltage, *VTH*. A voltage level above *VTH* is high; any level below *VTH* is low.

The second syntax defines a threshold for both a logic high (*VHI*) and low (*VLO*).

Note: If you specify *VTH*, *VLO*, and *VHI* in the same statement, then only *VTH* is processed, and *VLO* and *VHI* are ignored.

Example

```
.PARAM VTH = 3.0  
.DOUT node1 VTH(0.0n 0 1.0n 1  
+ 2.0n X 3.0n U 4.0n Z 5.0n 0)
```

The `.PARAM` statement in this example sets the *VTH* variable value to 3. The `.DOUT` statement, operating on the *node1* node, uses *VTH* as its threshold voltage.

When *node1* is above 3V, it is a logic 1; otherwise, it is a logic 0.

- At 0ns, the expected state of *node1* is logic-low.
- At 1ns, the expected state is logic-high.
- At 2ns, 3ns, and 4ns, the expected state is “do not care”.
- At 5ns, the expected state is again logic low.

Description

The digital output (`.DOUT`) statement specifies the expected final state of an output signal.

During simulation, HSPICE or HSPICE RF compares simulation results with the expected output. If the states are different, an error report results.

2: Commands in HSPICE Netlists

.DOUT

Argument	Definition
<i>nd</i>	Node name.
<i>time</i>	Absolute timepoint.
<i>state</i>	Expected condition of the <i>nd</i> node at the specified <i>time</i> : <ul style="list-style-type: none">• 0 expect ZERO,LOW.• 1 expect ONE,HIGH.• else Don't care.
VTH	Single voltage threshold.
VLO	Voltage of the logic-low state.
VHI	Voltage of the logic-high state.

For both syntax cases, the *time*, *state* pair describes the expected output. During simulation, the simulated results are compared against the expected output vector. If the states are different, HSPICE RF reports an error message. Legal values for *state* are:

.DOUT State Value	Description
0	expect ZERO
1	expect ONE
X, x	do not care
U, u	do not care
Z, z	expect HIGH IMPEDANCE (don't care). HSPICE RF cannot detect a high impedance state so it treats Z, z as "don't care" state.

See Also

.GRAPH
.MEASURE
.PARAM
.PLOT
.PRINT
.PROBE
.STIM

.EBD

Syntax

```
.EBD ebdname
+ file = 'filename'
+ model = 'modelname'
+ component = 'compname:reference_designator'
+ {component = 'compname:reference_designator' ...}
```

Example

```
.ebd ebd
  + file = 'test.ebd'
  + model = '16Meg X 8 SIMM Module'
  + component = 'cmpnt:u21'
.ibis cmpnt
  + file = 'ebd.ibs'
  + component = 'SIMM'
  + hsp_ver = 2003.09 nowarn
```

This example corresponds to the following *.ebd* file:

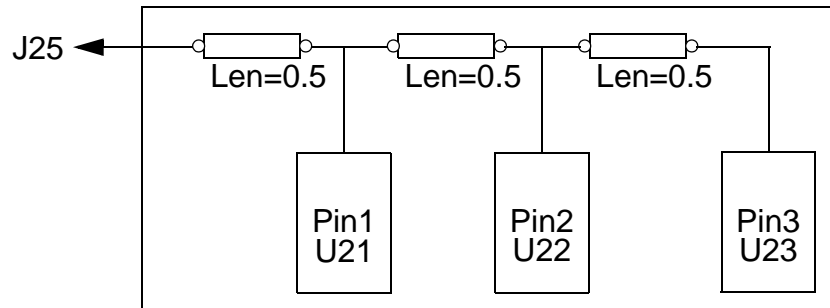
```
.....
[Begin Board Description] 16Meg X 8 SIMM Module
.....
[Pin List] signal_name
J25          POWER5
[Path Description] CAS_2
Pin J25
Len = 0.5 L=8.35n C=3.34p R=0.01 /
Node u21.1
Len = 0.5 L=8.35n C=3.34p R=0.01 /
Node u22.2
Len = 0.5 L=8.35n C=3.34p R=0.01 /
Node u23.3
```

Description

The `.EBD` command provides the IBIS(V 3.2) EBD feature. HSPICE uses the *.ebd* file when simulating the line connected with the `u21` reference_designator. The format of node names is *ebdname_SignalName*. For example, the format of a node name called J25 is `ebd_POWER5` (see Figure 1).

Argument	Definition
compname	Name of a <i>.ibs</i> file that describes a component.
reference_designator	Reference designator that maps the component.

Figure 1 Circuit Connection for EBD Example



See Also

- [.IBIS](#)
- [.PKG](#)

.ELSE

Syntax

```
.ELSE
```

Description

.ELSE precedes one or more commands in a conditional block, after the last .ELSEIF statement, but before the .ENDIF statement. HSPICE executes these commands by default, if the conditions in the preceding .IF statement, and in all of the preceding .ELSEIF statements in the same conditional block, are all false.

For the syntax and a description of how to use the .ELSE statement within the context of a conditional block, see the [.IF](#) statement.

See Also

[.ELSEIF](#)

[.ENDIF](#)

[.IF](#)

.ELSEIF

Syntax

```
.ELSEIF
```

Description

HSPICE executes the commands that follow the first `.ELSEIF` statement, only if *condition1* in the preceding `.IF` statement is false, and *condition2* in the first `.ELSEIF` statement is true.

If *condition1* in the `.IF` statement and *condition2* in the first `.ELSEIF` statement are both false, then HSPICE moves on to the next `.ELSEIF` statement, if there is one. If this second `.ELSEIF` condition is true, HSPICE executes the commands that follow the second `.ELSEIF` statement, instead of the commands after the first `.ELSEIF` statement.

HSPICE ignores the commands in all false `.IF` and `.ELSEIF` statements, until it reaches the first `.ELSEIF` condition that is true. If no `.IF` or `.ELSEIF` condition is true, HSPICE continues to the `.ELSE` statement

For the syntax and a description of how to use the `.ELSEIF` statement within the context of a conditional block, see the `.IF` statement.

See Also

- [.ELSE](#)
- [.ENDIF](#)
- [.IF](#)

2: Commands in HSPICE Netlists

.END

.END

Syntax

.END <comment>

Example

```
MOS OUTPUT
.OPTION NODE NOPAGE
VDS 3 0
VGS 2 0
M1 1 2 0 0 MOD1 L = 4U W = 6U AD = 10P AS = 10P
.MODEL MOD1 NMOS VTO = -2 NSUB = 1.0E15 TOX = 1000
+ UO = 550
VIDS 3 1
.DC VDS 0 10 0.5 VGS 0 5 1
.PRINT DC I(M1) V(2)
.END MOS OUTPUT
MOS CAPS
.OPTION SCALE = 1U SCALM = 1U WL ACCT
.OP
.TRAN .1 6
V1 1 0 PWL 0 -1.5V 6 4.5V
V2 2 0 1.5VOLTS
MODN1 2 1 0 0 M 10 3
.MODEL M NMOS VTO = 1 NSUB = 1E15 TOX = 1000
+ UO = 800 LEVEL = 1 CAPOP = 2
.PLOT TRAN V(1) (0,5) LX18(M1) LX19(M1) LX20(M1)
+ (0,6E-13)
.END MOS CAPS
```

Description

An .END statement must be the last statement in the input netlist file. The period preceding END is a required part of the statement.

Any text that follows the .END statement is a comment, and has no effect on that simulation.

An input file that contains more than one simulation run must include an .END statement for each simulation run. You can concatenate several simulations into a single file.

Argument	Definition
<comment>	Can be any comment. Typically, the comment is the name of the netlist file or of the simulation run, that this command terminates.

.ENDDATA

Syntax

`.ENDDATA`

Description

Use the `.ENDDATA` statement to end a `.DATA` block in an HSPICE input netlist.

See Also

[.DATA](#)

.ENDIF

Syntax

```
.ENDIF
```

Description

The `.ENDIF` statement ends a conditional block of commands that begins with an `.IF` statement.

For the syntax and a description of how to use the `.ENDIF` statement within the context of a conditional block, see the `.IF` statement.

See Also

- [.ELSE](#)
- [.ELSEIF](#)
- [.IF](#)

.ENDL

Syntax

.ENDL

Description

Use the .ENDL statement to end a .LIB statement in an HSPICE input netlist.

See Also

[.LIB](#)

.ENDS

Syntax

```
.ENDS <SUBNAME>
```

Example 1

```
.ENDS mos_circuit
```

This example terminates a subcircuit named *mos_circuit*.

Example 2

```
.ENDS
```

If you omit the subcircuit name as in this second example, this statement terminates all subcircuit definitions that begin with a `.SUBCKT` statement.

Description

Use the `.ENDS` statement to terminate a `.SUBCKT` statement.

This statement must be the last for any subcircuit definition that starts with a `.SUBCKT` command.

You can nest subcircuit references (calls) within subcircuits in HSPICE or HSPICE RF. However, in HSPICE RF, you cannot replicate output commands within subcircuit (`subckt`) definitions.

Argument	Definition
<SUBNAME>	Name of the subcircuit description to terminate, that begins with a <code>.SUBCKT</code> command.

See Also

[.SUBCKT](#)

.EOM

Syntax

```
.EOM <SUBNAME>
```

Example 1

```
.EOM diode_circuit
```

This example terminates a subcircuit named *diode_circuit*.

Example 2

```
.EOM
```

If you omit the subcircuit name as in this second example, this statement terminates all subcircuit definitions that begin with a `.MACRO` statement.

Description

Use the `.EOM` statement to terminate a `.MACRO` statement.

This statement must be the last for any subcircuit definition that starts with a `.MACRO` command.

You can nest subcircuit references (calls) within subcircuits in HSPICE or HSPICE RF. However, in HSPICE RF, you cannot replicate output commands within subcircuit (`subckt`) definitions.

Argument	Definition
<SUBNAME>	Name of the subcircuit description to terminate, that begins with a <code>.SUBCKT</code> command.

See Also

[.MACRO](#)

2: Commands in HSPICE Netlists

.FFT

.FFT

Syntax

```
.FFT <output_var> <START=value> <STOP=value>  
+ <NP=value> <FORMAT=keyword>  
+ <WINDOW=keyword> <ALFA=value>  
+ <FREQ=value> <FMIN=value> <FMAX=value>
```

Example 1

```
.FFT v(1)  
.FFT v(1,2) np=1024 start=0.3m stop=0.5m freq=5.0k  
+ window=kaiser alfa=2.5  
.FFT I(rload) start=0m to=2.0m fmin=100k fmax=120k  
+ format=unorm  
.FFT par('v(1) + v(2)') from=0.2u stop=1.2u  
+ window=harris
```

Example 2

```
.FFT v(1) np=1024  
.FFT v(2) np=1024
```

This example generates an *.ft0* file for the FFT of *v(1)*, and an *.ft1* file for the FFT of *v(2)*.

Description

The `.FFT` statement uses internal time point values to calculate the Discrete Fourier Transform (DFT) value, which HSPICE uses for spectrum analysis. A DFT uses sequences of time values to determine the frequency content of analog signals in circuit simulation.

You can specify only one output variable in an `.FFT` command. The following is an incorrect use of the command, because it contains two variables in one `.FFT` command:

```
.FFT v(1) v(2) np=1024
```

Argument	Definition
output_var	Can be any valid output variable, such as voltage, current, or power.
START	Start of the output variable waveform to analyze. Defaults to the START value in the <code>.TRAN</code> statement, which defaults to 0.

Argument	Definition
FROM	An alias for START in .FFT statements.
STOP	End of the output variable waveform to analyze. Defaults to the TSTOP value in the .TRAN statement.
TO	An alias for STOP, in .FFT statements.
NP	Number of points to use in the FFT analysis. NP must be a power of 2. If NP is not a power of 2, HSPICE automatically adjusts it to the closest higher number that is a power of 2. Default=1024.
FORMAT	Specifies the output format: <ul style="list-style-type: none"> • NORM= normalized magnitude (default) • UNORM=unnormalized magnitude
WINDOW	Specifies the window type to use: <ul style="list-style-type: none"> • RECT=simple rectangular truncation window (default). • BART=Bartlett (triangular) window. • HANN=Hanning window. • HAMM=Hamming window. • BLACK=Blackman window. • HARRIS=Blackman-Harris window. • GAUSS=Gaussian window. • KAISER=Kaiser-Bessel window.
ALFA	Parameter to use in GAUSS and KAISER windows to control the highest side-lobe level, bandwidth, and so on. $1.0 \leq \text{ALFA} \leq 20.0$ Default=3.0
FREQ	Frequency to analyze. If FREQ is non-zero, the output lists only the harmonics of this frequency, based on FMIN and FMAX. HSPICE also prints the THD for these harmonics. Default=0.0 (Hz).

2: Commands in HSPICE Netlists

.FFT

Argument	Definition
FMIN	Minimum frequency for which HSPICE prints FFT output into the listing file. THD calculations also use this frequency. $T = (\text{STOP} - \text{START})$ Default=1.0/T (Hz).
FMAX	Maximum frequency for which HSPICE prints FFT output into the listing file. THD calculations also use this frequency. Default=0.5*NP*FM IN (Hz).

See Also

[.TRAN](#)

.FOUR

Syntax

```
.FOUR freq ov1 <ov2 ov3 ...>
```

Example

```
.FOUR 100K V(5)
```

Description

This statement performs a Fourier analysis as part of the transient analysis. You can use the `.FOUR` statement in or HSPICE RF to HSPICE perform the Fourier analysis over the interval (`tstop-fperiod`, `tstop`), where:

- `tstop` is the final time, specified for the transient analysis.
- `fperiod` is a fundamental frequency period (`freq` parameter).

HSPICE or HSPICE RF performs Fourier analysis on 501 points of transient analysis data on the last $1/f$ time period, where f is the fundamental Fourier frequency. HSPICE or HSPICE RF interpolates transient data to fit on 501 points, running from (`tstop-1/f`) to `tstop`.

To calculate the phase, the normalized component, and the Fourier component, HSPICE or HSPICE RF uses 10 frequency bins. The Fourier analysis determines the DC component, and the first nine AC components. For improved accuracy, the `.FOUR` statement can use non-linear, instead of linear interpolation.

You can only use a `.FOUR` statement in conjunction with a `.TRAN` statement.

Argument	Definition
<code>freq</code>	Fundamental frequency
<code>ov1 ...</code>	Output variables to analyze.

See Also

[.TRAN](#)

.FSOPTIONS

Syntax

```
.FSOPTIONS name <ACCURACY=LOW|MEDIUM|HIGH> +
<GRIDFACTOR=val> <PRINTDATA=YES|NO>
+ <COMPUTE0=YES|NO> <COMPUTE0D=YES|NO>
+ <COMPUTERO=YES|NO> <COMPUTERS=YES|NO>
```

Description

Use the `.FSOPTIONS` statement to set various options for the field solver. The following rules apply to the Field Solver when specifying options with the `.FSOPTIONS` statement:

- The field solver always computes the L and C matrices.
- If `COMPUTERS=YES`, then the field solver starts, and calculates `Lo`, `Ro`, and `Rs`.
- For each accuracy mode, the field solver uses either the pre-defined number of segments or the number of segments that you specified. It then multiplies this number times the `GRIDFACTOR` to obtain the final number of segments.

Because a wide range of applications are available, the pre-defined accuracy level might not be accurate enough for some applications. If you need a higher accuracy than the value that the `HIGH` option sets, then increase either the `GRIDFACTOR` value or the `N`, `NH`, or `NW` values to increase the mesh density.

Argument	Definition
name	Option name.
ACCURACY	Sets the solver accuracy to one of the following: <ul style="list-style-type: none"> • LOW • MEDIUM • HIGH
GRIDFACTOR	Multiplication factor (integer) to determine the final number of segments used to define the shape. If you set <code>COMPUTERS=yes</code> , the field solver does not use this parameter to compute <code>Ro</code> and <code>Rs</code> values.
PRINTDATA	The solver prints output matrices.

Argument	Definition
COMPUTEGO	The solver computes the static conductance matrix.
COMPUTEGB	The solver computes the dielectric loss matrix.
COMPUTERO	The solver computes the DC resistance matrix.
COMPUTERS	The solver computes the skin-effect resistance matrix. This parameter uses the filament method solver to compute R_o and R_s .

See Also

[.LAYERSTACK](#)
[.MATERIAL](#)
[.SHAPE](#)

.GLOBAL

Syntax

```
.GLOBAL node1 node2 node3 ...
```

Example

This example shows global definitions for VDD and `input_sig` nodes.

```
.GLOBAL VDD input_sig
```

Description

The `.GLOBAL` statement globally assigns a node name in HSPICE or HSPICE RF. This means that all references to a global node name, used at any level of the hierarchy in the circuit, connect to the same node.

The most common use of a `.GLOBAL` statement is if your netlist file includes subcircuits. This statement assigns a common node name to subcircuit nodes. Another common use of `.GLOBAL` statements is to assign power supply connections of all subcircuits. For example, `.GLOBAL VCC` connects all subcircuits with the internal node name VCC.

Ordinarily, in a subcircuit, the node name consists of the circuit number, concatenated to the node name. When you use a `.GLOBAL` statement, HSPICE or HSPICE RF does not concatenate the node name with the circuit number, and assigns only the global name. You can then exclude the power node name in the subcircuit or macro call.

Argument	Definition
node1 node2	Name of a global nodes, such as supply and clock names; overrides local subcircuit definitions.

.GRAPH

Note: This is an obsolete command. You can gain the same functionality by using the [.PROBE](#) command.

Syntax

```
.GRAPH antype <MODEL = mname> <unam1 = > ov1,
+ <unam2 = >ov2 ... <unamn = >ovn (plo,phi)
```

Example

```
.GRAPH DC cgb = lx18(m1) cgd = lx19(m1)
+ cgs = lx20(m1)
.GRAPH DC MODEL = plotbjt
+ model_ib = i2(q1)      meas_ib = par(ib)
+ model_ic = il(q1)      meas_ic = par(ic)
+ model_beta = par('il(q1)/i2(q1)')
+ meas_beta = par('par(ic)/par(ib)')(1e-10,1e-1)
.MODEL plotbjt PLOT MONO = 1 YSCAL = 2 XSCAL = 2
+ XMIN = 1e-8 XMAX = 1e-1
```

Description

Use the `.GRAPH` statement when you need high-resolution plots of HSPICE simulation results. You cannot use `.GRAPH` statements in the PC version of HSPICE or in any versions of HSPICE RF.

Each `.GRAPH` statement creates a new `.gr#` file, where `#` ranges first from 0 to 9, and then from a to z. You can create up to 10000 graph files.

You can include wildcards in `.GRAPH` statements (HSPICE only).

Argument	Definition
<i>antype</i>	Type of analysis for the specified plots (outputs). Analysis types are: DC, AC, TRAN, NOISE, or DISTO (you cannot run DISTO analysis in HSPICE RF).
<i>mname</i>	Plot model name, referenced in the <code>.GRAPH</code> statement. Use <code>.GRAPH</code> and its plot name to create high-resolution plots directly from HSPICE.
<i>unam1...</i>	You can define output names, which correspond to the <code>ov1 ov2 ...</code> output variables (<i>unam1 unam2 ...</i>), and use them as labels, instead of output variables for a high resolution graphic output.

2: Commands in HSPICE Netlists

.GRAPH

Argument	Definition
<i>ov1 ...</i>	Output variables to print. Can be voltage, current, or element template variables (HSPICE only; HSPICE RF does not support element template output, or .GRAPH statements), from a different type of analysis. You can also use algebraic expressions as output variables, but you must define them inside the PAR() statement.
<i>plo, phi</i>	Lower and upper plot limits. Set the plot limits only at the end of the .GRAPH statement.

See Also

[.DOUT](#)
[.MEASURE](#)
[.PLOT](#)
[.PRINT](#)
[.PROBE](#)
[.STIM](#)

.HDL

Syntax

```
.HDL filename
```

Example 1

```
.hdl "/myhome/Verilog_A_lib/res.va"
```

This example loads the `res.va` Verilog-A model file from the `/myhome/Verilog_A_lib` directory.

Example 2

```
.hdl "va_models"
```

This example loads the `va_models.va` Verilog-A model file (not `va_model` file) from the current working directory.

Description

This `.HDL` command specifies the Verilog-A source name and path within the netlist. The Verilog-A file is assumed to have a `*.va` extension only when a prefix is provided.

In `.MODEL` statements, you must add the Verilog-A type of model cards. Every Verilog-A module can have one or more associated model cards. The type of model card(s) should be the same as the Verilog-A module name. Verilog-A module names cannot conflict with HSPICE built-in device keywords. If a conflict occurs, HSPICE issues a warning message and the Verilog-A module definition is ignored.

2: Commands in HSPICE Netlists

.IBIS

.IBIS

Syntax

```
.IBIS cname keyword_1 = value_1 ...  
+ [keyword_M= value_M]
```

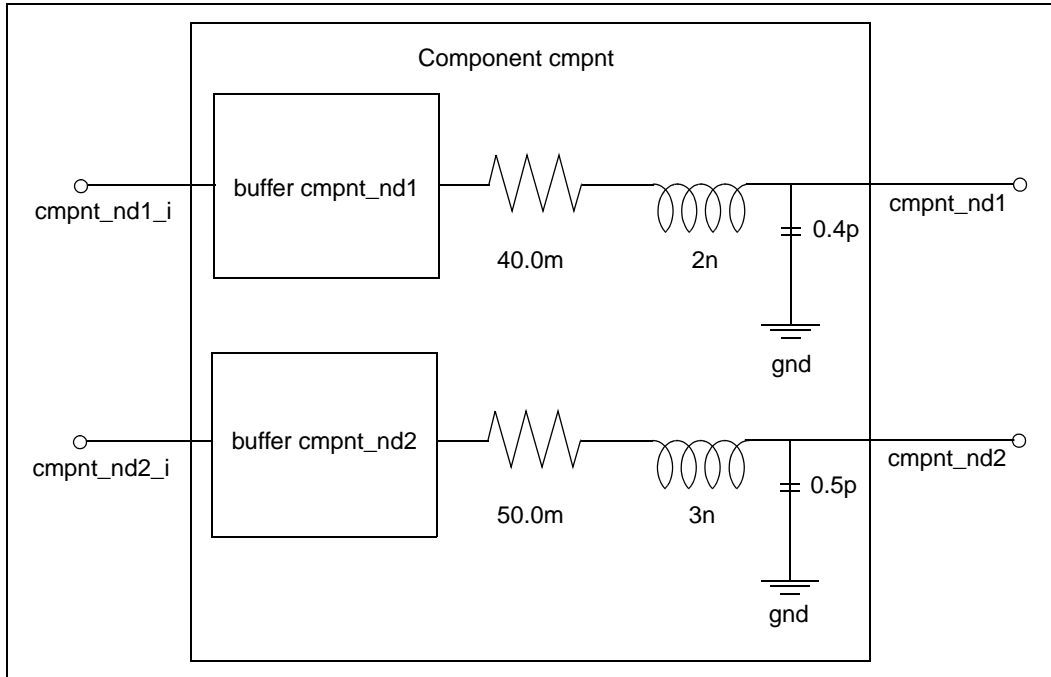
Example

```
.ibis cmpnt  
+ file = 'ebd.ibs'  
+ component = 'SIMM'  
+ hsp_ver = 2002.4 nowarn package = 2
```

This example corresponds to the following *ebd.ibs* file:

```
[Component]    SIMM  
[Manufacturer] TEST  
[Package]  
R_pkg      200m      NA      NA  
L_pkg      7.0nH      NA      NA  
C_pkg      1.5pF      NA      NA  
|  
[Pin]       signal_name  model_name  R_pin    L_pin    C_pin  
|  
1          ND1         ECL        40.0m    2n      0.4p  
2          ND2         NMOS       50.0m    3n      0.5p  
.....
```

Figure 2 Equivalent Circuit for EBD Example



Description

This is the general syntax for the .IBIS command when used with a component. The optional keywords are in square brackets.

Argument	Definition
<i>cname</i>	Instance name of this ibis command

2: Commands in HSPICE Netlists

.IBIS

Argument	Definition
<i>keyword_i</i> <i>value_i</i>	<p>Assigns the <i>value_i</i> value to the <i>keyword_i</i> keyword. Required keywords are:</p> <ul style="list-style-type: none">• <i>file</i>='file_name' identifies the IBIS file. The <i>file_name</i> parameter must be lower case and must specify either the absolute path for the file or the path relative to the directory from which you run the simulation. For example: file = '.ibis/at16245.ibs' file = '/home/oneuser/ibis/models/abc.ibs'• <i>component</i>='component_name' identifies the component for an <code>.ibis</code> command from the IBIS file, specified using the <i>file</i>='...' keyword. The <i>component_name</i> keyword is case-sensitive, and it must match one of the components from the IBIS file. For example: component = 'procfast' component = 'Virtex_SSTL_3-I_BG432' component = '10_pdref' \$ SPICE-formatted [Pin]

Argument	Definition
<i>keyword_m=</i> <i>value_m</i>	<p>Optional keywords:</p> <ul style="list-style-type: none"> • package = [0 1 2 3] (default is 3) • 0, does not add the rlc package into the component. • 1, adds [Package] (in .ibs file). • 2, adds [Pin] (in .ibs file). • 3, If [Package Model] is defined, set package with package model. If it is not defined, set package with [Pin]. If the package information is not set in [Pin], set package with [Package] as a default. Package Model can be defined in either the IBIS file or the PKG file. • The following optional keywords are the same as for the B Element (I/O buffer). For more information, see “Specifying Common Keywords” in the Modeling Input/Output Buffers Using IBIS chapter of the <i>HSPICE Signal Integrity Guide</i>. • typ • interpo • ramp_rwf • ramp_fwf • rwf_tune • fwf_tune • pd_scal • pu_scal • pc_scal • gc_scal • rwf_scal • fwf_scal • nowarn • hsp_ver • c_com_pu • c_com_pd • c_com_pc • c_com_gc

See Also

[.EBD](#)
[.PKG](#)

.IC**Syntax**

```
.IC V(node1) = val1 V(node2) = val2 ...
```

Example

```
.IC V(11) = 5 V(4) = -5 V(2) = 2.2
```

Description

Use the `.IC` statement or the `.DCVOLT` statement to set transient initial conditions in HSPICE, but not in HSPICE RF. How it initializes depends on whether the `.TRAN` analysis statement includes the `UIC` parameter.

Note: In HSPICE RF, `.IC` is always set to `OFF`.

If you specify the `UIC` parameter in the `.TRAN` statement, HSPICE does not calculate the initial DC operating point, but directly enters transient analysis. Transient analysis uses the `.IC` initialization values as part of the solution for timepoint zero (calculating the zero timepoint applies a fixed equivalent voltage source). The `.IC` statement is equivalent to specifying the `IC` parameter on each element statement, but is more convenient. You can still specify the `IC` parameter, but it does not have precedence over values set in the `.IC` statement.

If you do *not* specify the `UIC` parameter in the `.TRAN` statement, HSPICE computes the DC operating point solution before the transient analysis. The node voltages that you specify in the `.IC` statement are fixed to determine the DC operating point. HSPICE RF does not output node voltage from operating point (`.OP`), if time (`t`) < 0. Transient analysis releases the initialized nodes to calculate the second and later time points.

Argument	Definition
val1 ...	Specifies voltages. The significance of these voltages depends on whether you specify the <code>UIC</code> parameter in the <code>.TRAN</code> statement.
node1 ...	Node numbers or names can include full paths or circuit numbers.

See Also

[.DCVOLT](#)
[.TRAN](#)
[.OPTION DCIC](#)

.ICM

Syntax

```
.ICM icmname  
+ file = 'icmfilename'  
+ model = 'icmmodelname'
```

Example 1

```
.ICM icm1  
+ file = 'test1.icm'  
+ model = 'FourLineModel1'
```

Example 2

The following example shows how to reference a pin of ICM model in HSPICE netlist.

```
icm1_NodeMap1_pin1, icm1_NodeMap2_pin1,  
icm1_NodeMap2_pin2, ...
```

Description

The `.ICM` command automatically creates port names that reference the pin name of an ICM model and generate a series of element (W/S/RLGCK) nodes on the pin when one of the following conditions occur:

- If the model is described using [Nodal Path Description] `'icmname'_nodemapname'_pinname'`
- If the model is described using [Tree Path Description] `'icmname'_pinmapname'_pinname'`

Argument	Definition
<code>icmname</code>	<code>.ICM</code> command card name.
<code>icmfilename</code>	Name of an <code>.icm</code> file that contains an ICM model.
<code>icmmodelname</code>	Working model in an <code>.icm</code> file.
<code>nodemapname</code>	Name of the [ICM node map] keyword in an <code>.icm</code> file.
<code>pinmapname</code>	Name of the [ICM pin map] keyword in an <code>.icm</code> file.
<code>pinname</code>	Name of the first column of entries of the [ICM node map] or [ICM pin map].

.IF**Syntax**

```
.IF (condition1)
...
<.ELSEIF (condition2)
... >
<.ELSE
... >
.ENDIF
```

Example

```
.IF (a==b)
.INCLUDE /myhome/subcircuits/diode_circuit1
...
.ELSEIF (a==c)
.INCLUDE /myhome/subcircuits/diode_circuit2
...
.ELSE
.INCLUDE /myhome/subcircuits/diode_circuit3
...
.ENDIF
```

Description

HSPICE executes the commands that follow the first `.ELSEIF` statement, only if *condition1* in the preceding `.IF` statement is false, and *condition2* in the first `.ELSEIF` statement is true.

If *condition1* in the `.IF` statement and *condition2* in the first `.ELSEIF` statement are both false, then HSPICE moves on to the next `.ELSEIF` statement, if there is one. If this second `.ELSEIF` condition is true, HSPICE executes the commands that follow the second `.ELSEIF` statement, instead of the commands after the first `.ELSEIF` statement.

HSPICE ignores the commands in all false `.IF` and `.ELSEIF` statements, until it reaches the first `.ELSEIF` condition that is true. If no `.IF` or `.ELSEIF` condition is true, HSPICE continues to the `.ELSE` statement

`.ELSE` precedes one or more commands in a conditional block, after the last `.ELSEIF` statement, but before the `.ENDIF` statement. HSPICE executes these commands by default, if the conditions in the preceding `.IF` statement, and in all of the preceding `.ELSEIF` statements in the same conditional block, are all false.

2: Commands in HSPICE Netlists

.IF

The `.ENDIF` statement ends a conditional block of commands that begins with an `.IF` statement.

Argument	Definition
<i>condcition1</i>	Condition that must be true, before HSPICE executes the commands that follow the <code>.IF</code> statement.
<i>condition2</i>	Condition that must be true, before HSPICE executes the commands that follow the <code>.ELSEIF</code> statement. HSPICE executes the commands that follow <i>condition2</i> , only if <i>condition1</i> is false and <i>condition2</i> is true.

See Also

[.ELSE](#)
[.ELSEIF](#)
[.ENDIF](#)

.INCLUDE

Syntax

```
.INCLUDE '<filepath> filename'
```

Example

```
.INCLUDE /myhome/subcircuits/diode_circuit
```

Description

You can include a netlist as a subcircuit in one or more other netlists. To include another netlist in the current netlist, use the `.INCLUDE` statement.

Argument	Definition
<i>filepath</i>	<p>Path name of a file for computer operating systems that support tree-structured directories.</p> <p>A .INC file can contain nested .INC calls to itself or to another .INC file. If you use a relative path in a nested .INC call, the path starts from the directory of the parent .INC file, not from the work directory. If the path starts from the work directory, HSPICE can also find the .INC file, but prints a warning.</p>
<i>filename</i>	<p>Name of a file to include in the data file. The file path, plus the file name, can be up to 1024 characters long. You can use any valid file name for the computer's operating system. You <i>must</i> enclose the file path and name in single or double quotation marks.</p>

.LAYERSTACK

Syntax

```
.LAYERSTACK sname <BACKGROUND=mname>
+ <LAYER=(mname,thickness) ...>
```

Description

A layer stack defines a stack of dielectric or metal layers.

You must associate each transmission line system with *one*, and *only one*, layer stack. However, you can associate a single-layer stack with *many* transmission line systems.

In the layer stack:

- Layers are listed from bottom to top.
- Metal layers (ground planes) are located only at the bottom, only at the top, or both at the top and bottom.
- Layers are stacked in the y-direction, and the bottom of a layer stack is at $y=0$.
- All conductors must be located above $y=0$.
- Background material must be dielectric.

The following limiting cases apply to the `.LAYERSTACK` command:

- Free space without ground:
`.LAYERSTACK mystack`
- Free space with a (bottom) ground plane:
`.LAYERSTACK halfSpace PEC 0.1mm`

Argument	Definition
sname	Layer stack name.
mname	Material name.
BACKGROUND	Background dielectric material name. By default, the Field Solver assumes AIR for the background.
thickness	Layer thickness.

See Also

[.FSOPTIONS](#)
[.MATERIAL](#)
[.SHAPE](#)

.LIB

Syntax

Use the following syntax for library calls:

```
.LIB '<filepath> filename' entryname
```

Use the following syntax to define library files:

```
.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
```

Example 1

```
* Library call
.LIB 'MODELS' cmos1
```

Example 2

```
.LIB MOS7
$ Any valid set of HSPICE commands
.
.
.
.ENDL MOS7
```

Example 3

The following are an illegal example and a legal example of nested .LIB statements for the *file3* library.

Illegal:

```
.LIB MOS7
...
.LIB 'file3' MOS7 $ This call is illegal in MOS7 library
...
...
.ENDL
```


Legal:

```
.LIB MOS7
...
.LIB 'file1' MOS8
.LIB 'file2' MOS9
.LIB CTT $ file2 is already open for the CTT
    $ entry point
.ENDL
```

Example 4

```
.LIB TT
$TYPICAL P-CHANNEL AND N-CHANNEL CMOS LIBRARY
$ PROCESS: 1.0U CMOS, FAB7
$ following distributions are 3 sigma ABSOLUTE GAUSSIAN
.PARAM TOX = AGAUSS(200,20,3)  $ 200 angstrom +/- 20a
+ XL = AGAUSS(0.1u,0.13u,3)  $ polysilicon CD
+ DELVTON = AGAUSS(0.0,.2V,3)  $ n-ch threshold change
+ DELVTOP = AGAUSS(0.0,.15V,3)
    $ p-ch threshold change
.INC '/usr/meta/lib/cmos1_mod.dat'
    $ model include file
.ENDL TT
.LIB FF
$HIGH GAIN P-CH AND N-CH CMOS LIBRARY 3SIGMA VALUES
.PARAM TOX = 220 XL = -0.03 DELVTON = -.2V
+ DELVTOP = -0.15V
.INC '/usr/meta/lib/cmos1_mod.dat'
    $ model include file
.ENDL FF
```

This example is a .LIB call statement of model skew parameters, and features both worst-case and statistical distribution data. The statistical distribution median value is the default for all non-Monte Carlo analysis. The model is in the /usr/meta/lib/cmos1_mod.dat include file.

```
.MODEL NCH NMOS LEVEL = 2 XL = XL TOX = TOX
+ DELVTO = DELVTON .....
.MODEL PCH PMOS LEVEL = 2 XL = XL TOX = TOX
+ DELVTO = DELVTOP .....
```

The .model keyword (left side) equates to the skew parameter (right side). A .model keyword can be the same as a skew parameter.

2: Commands in HSPICE Netlists

.LIB

Description

To create and read from libraries of commonly-used commands, device models, subcircuit analysis, and statements (library calls) in library files, use the `.LIB` call statement. As HSPICE or HSPICE RF encounters each `.LIB` call name in the main data file, it reads the corresponding entry from the designated library file, until it finds an `.ENDL` statement.

You can also place a `.LIB` call statement in an `.ALTER` block.

To build libraries (library file definition), use the `.LIB` statement in a library file. For each macro in a library, use a library definition statement (`.LIB` *entryname*) and an `.ENDL` statement.

The `.LIB` statement begins the library macro, and the `.ENDL` statement ends the library macro. The text after a library file entry name must consist of HSPICE or HSPICE RF statements.

Library calls can call other libraries (nested library calls), if they are different files. You can nest library calls to any depth. Use nesting with the `.ALTER` statement to create a sequence of model runs. Each run can consist of similar components by using different model parameters, without duplicating the entire input file.

The simulator uses the `.LIB` statement and the `.INCLUDE` statement to access the models and skew parameters. The library contains parameters that modify `.MODEL` statements.

Argument	Definition
<i>filepath</i>	Path to a file. Used where a computer supports tree-structured directories. When the LIB file (or alias) is in the same directory where you run HSPICE or HSPICE RF, you do not need to specify a directory path; the netlist runs on any machine. Use the <code>“./”</code> syntax in the filepath to designate the parent directory of the current directory.
<i>entryname</i>	Entry name for the section of the library file to include. The first character of an <i>entryname</i> cannot be an integer.
<i>filename</i>	Name of a file to include in the data file. The combination of <i>filepath</i> plus <i>filename</i> can be up to 256 characters long, structured as any filename that is valid for the computer's operating system. Enclose the file path and file name in single or double quotation marks. Use the <code>“./”</code> syntax in the <i>filename</i> to designate the parent directory of the current directory.

See Also

[.ALTER](#)

[.ENDL](#)

[.INCLUDE](#)

.LIN

Syntax

```
.LIN <sparcalc = [1|0] <modelname = ...>>  
+ <filename = ...> <format=[selem|citi|touchstone]>  
+ <noisecalc = [1|0] <gdcalc = [1|0]>  
+ <mixedmode2port = [dd|dc|ds|cd|cc|cs|sd|sc|ss]>  
+ <dataformat = [ri|ma|db]>
```

Example

```
.LIN sparcalc=1 modelname=my_custom_model  
+ filename=mydesign format=touchstone noisecalc=1  
+ gdcalc=1 dataformat=ri
```

This example extracts linear transfer parameters for a general multi-port network, performs a 2-port noise analysis, and performs a group-delay analysis for a model named `my_custom_model`. The output is in the `mydesign` output file, which is in the TOUCHSTONE format. The data format in the TOUCHSTONE file is real-imaginary.

Description

The `.LIN` command extracts noise and linear transfer parameters for a general multi-port network.

When used with P (port) element(s) and `.AC` commands, `.LIN` makes available a broad set of linear port-wise measurements:

- standard and mixed-mode multi-port S (scattering) parameters
- standard and mixed-mode multi-port Y/Z parameters
- standard mode multi-port H parameter
- standard mode two-port noise parameters
- standard and mixed-mode group delays
- standard mode stability factors
- standard mode gain factors
- standard mode matching coefficients

The `.LIN` command computes the S (scattering), Y (admittance), Z (impedance) parameters directly, and H (hybrid) parameters directly based on the location of the port (P) elements in your circuit, and the specified values for their reference impedances.

The .LIN command also supports mixed-mode transfer parameters calculation and group delay analysis when used together with mixed-mode P elements.

By default, the .LIN command creates a .sc# file with the same base name as your netlist. This file contains S parameter, noise parameter, and group delay data as a function of the frequency. You can use this file as model data for the S element.

Argument	Definition
sparcalc	If 1, extract S parameters (default).
modelname	Model name listed in the .MODEL statement in the .sc# model output file.
filename	Output file name (default=netlist name).
format	Output file format: <ul style="list-style-type: none"> • selem is for S element .sc# format, which you can include in the netlist. • citi is CITIfile format. • touchstone is TOUCHSTONE file format.
noisecalc	If 1, extract noise parameters (perform 2-port noise analysis). Default=0.
gdcalc	If 1, extract group delay (perform group delay analysis). Default=0.

2: Commands in HSPICE Netlists

.LIN

Argument	Definition
mixedmode2port	<p>The mixedmode2port keyword describes the mixed-mode data map of output mixed mode S parameter matrix. The availability and default value for this keyword depends on the first two port (P element) configuration as follows:</p> <ul style="list-style-type: none">• case 1: p1=p2=single (standard mode P element) available: ss default: ss• case 2: p1=p2=balanced (mixed mode P element) available: dd, cd, dc, cc default: dd• case 3: p1=balanced p2=single available: ds, cs default: ds• case 4: p1=single p2=balanced available: sd, sc default: sd
dataformat	<p>The dataformat keyword describe the data format output to the .sc#/touchstone/citi file.</p> <ul style="list-style-type: none">• dataformat=RI, real-imaginary. This is the default for .sc#/citi file.• dataformat=MA, magnitude-phase. This is the default format for touchstone file.• dataformat=DB, DB(magnitude)-phase. <p>HSPICE uses six digits for both frequency and S Parameters in HSPICE generated data files (.sc#/touchstone/citifile). The number of digits for noise parameters are five in .sc# and touchstone files and six digits in citifiles.</p>

.LOAD

Syntax

```
.LOAD <FILE = load_file> <RUN = PREVIOUS | CURRENT>
```

Example 1

```
.TITLE
.SAVE FILE=design.ic
.LOAD FILE=design.ic0
        $load--design.ic0 save--design.ic0
.alter
...        $load--none    save--design.ic1
.alter
...        $load--none    save--design.ic2
.end
```

This example loads a file name *design.ic0*, which you previously saved using a `.SAVE` command.

Example 2

```
.TITLE
.SAVE FILE=design.ic
.LOAD FILE=design.ic RUN=PREVIOUS
        $load--none    save--design.ic0
.alter
...        $load--design.ic0 save--design.ic1
.alter
...        $load--design.ic1 save--design.ic2
.end
```

Example 3

```
.TITLE
.SAVE FILE=design.ic
.LOAD FILE=design.ic RUN=CURRENT
        $load--design.ic0 save--design.ic0
.alter
...        $load--design.ic1 save--design.ic1
.alter
...        $load--design.ic2 save--design.ic2
.end
```

Description

Use the `.LOAD` statement to input the contents of a file, that you stored using the `.SAVE` statement in HSPICE.

2: Commands in HSPICE Netlists

.LOAD

Note: HSPICE RF does not support the `.SAVE` and `.LOAD` (save and restart) statements.

Files stored with the `.SAVE` statement contain operating point information for the point in the analysis at which you executed `.SAVE`.

Do not use the `.LOAD` command for concatenated netlist files.

Argument	Definition
<i>load_file</i>	Name of the file in which <code>.SAVE</code> saved an operating point for the circuit under simulation. The format of the file name is <code><design>.ic#</code> . Default is <code><design>.ic0</code> , where <i>design</i> is the root name of the design.
RUN=	Used only outside of <code>.ALTER</code> statements in a netlist that contains <code>.ALTER</code> statements. The format of file name is <code><design>.ic</code> .
PREVIOUS	Each <code>.ALTER</code> run uses the saved operating point from the previous <code>.ALTER</code> run in the same simulation.
CURRENT	Each <code>.ALTER</code> run uses the saved operating point from the current <code>.ALTER</code> run in the last simulation.

See Also

[.ALTER](#)
[.SAVE](#)

.MACRO

In HSPICE RF, you cannot replicate output commands within subcircuit (subckt) definitions.

Syntax

```
.MACRO subnam n1 <n2 n3 ...> <parnam = val>
.EOM
```

Example 1

```
*FILE SUB2.SP TEST OF SUBCIRCUITS
.OPTION LIST ACCT
  V1 1 0 1
.PARAM P5 = 5 P2 = 10
.SUBCKT SUB1 1 2 P4 = 4
  R1 1 0 P4
  R2 2 0 P5
  X1 1 2 SUB2 P6 = 7
  X2 1 2 SUB2
.ENDS
*
.MACRO SUB2 1 2 P6 = 11
  R1 1 2 P6
  R2 2 0 P2
.EOM
  X1 1 2 SUB1 P4 = 6
  X2 3 4 SUB1 P6 = 15
  X3 3 4 SUB2
*
.MODEL DA D CJA = CAJA CJP = CAJP VRB = -20 IS = 7.62E-18
+ PHI = .5 EXA = .5 EXP = .33
.PARAM CAJA = 2.535E-16 CAJP = 2.53E-16
.END
```

The preceding example defines two subcircuits: SUB1 and SUB2. These are resistor divider networks, whose resistance values are parameters (variables). The X1, X2, and X3 statements call these subcircuits. Because the resistor values are different in each call, these three calls produce different subcircuits.

Example 2

```
.SUBCKT Inv a y Strength = 3
  Mp1 <MosPinList> pMosMod L = 1.2u W = 'Strength * 2u'
  Mn1 <MosPinList> nMosMod L = 1.2u W = 'Strength * 1u'
.ENDS
...
```

2: Commands in HSPICE Netlists

.MACRO

```
xInv0 a y0 Inv          $ Default devices: p device = 6u,  
          $ n device = 3u  
xInv1 a y1 Inv Strength = 5          $ p device = 10u, n  
device = 5u  
xInv2 a y2 Inv Strength = 1          $ p device = 2u, n  
device = 1u  
...
```

This example implements an inverter that uses a *Strength* parameter. By default, the inverter can drive three devices. Enter a new value for the *Strength* parameter in the element line to select larger or smaller inverters for the application.

Description

You can create a subcircuit description for a commonly-used circuit, and include one or more references to the subcircuit in your netlist.

To define a subcircuit in your netlist, use the `.MACRO` statement. Use the `.EOM` statement to terminate a `.MACRO` statement.

Argument	Definition
<i>subnam</i>	Specifies a reference name for the subcircuit model call.
<i>n1 ...</i>	Node numbers for external reference; cannot be the ground node (zero). Any element nodes that are in the subcircuit, but are not in this list, are strictly local with three exceptions: <ul style="list-style-type: none">• Ground node (zero).• Nodes assigned using BULK = node in MOSFET or BJT models.• Nodes assigned using the <code>.GLOBAL</code> statement.
<i>parnam</i>	A parameter name set to a value. Use only in the subcircuit. To override this value, assign it in the subcircuit call, or set a value in a <code>.PARAM</code> statement.
<i>SubDefaultsList</i>	<SubParam1>=<Expression> [<SubParam2>=<Expression>...]

See Also

[.ENDS](#)
[.EOM](#)
[.MACRO](#)
[.SUBCKT](#)

.MALIAS

Syntax

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

- *model_name* is the model name defined in the .model card.
- *alias_name1...* is the alias that an instance (element) of the model references.

Example

```
*file: test malias statement
.OPTION acct tnom=50 list gmin=1e-14 post
.temp 0.0 25
.tran .1 2
vdd 2 0 pwl 0 -1 1 1
d1 2 1 zend dtemp=25
d2 1 0 zen dtemp=25
* malias statements
.malias zendef = zen zend
* model definition
.model zendef d (vj=.8 is=1e-16 ibv=1e-9 bv=6.0 rs=10
+ tt=0.11n n=1.0 eg=1.11 m=.5 cjo=1pf tref=50)
.end
```

- *zendef* is a diode model
- *zen* and *zend* are its aliases.
- The *zendef* model points to both the *zen* and *zend* aliases.

Description

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. You cannot use the .MALIAS statement in HSPICE RF.

.MALIAS differs from .ALIAS in two ways:

- A model can define the alias in an .ALIAS statement, but not the alias in a .MALIAS statement. The .MALIAS statement applies to an element (an instance of the model), not to the model itself.
- The .ALIAS command works only if you include .ALTER in the netlist. You can use .MALIAS without .ALTER.

You can use `.MALIAS` to alias to a model name that you defined in a `.MODEL` statement or to alias to a subcircuit name that you defined in a `.SUBCKT` statement. The syntax for `.MALIAS` is the same in either usage.

Note: Using `.MALIAS` in `.ALTER` blocks is not recommended or supported.

See Also

[.ALIAS](#)

.MATERIAL

Note: You cannot use the `.MATERIAL` statement in HSPICE RF.

Syntax

```
.MATERIAL mname METAL|DIELECTRIC <ER=val>
+ <UR=val> <CONDUCTIVITY=val> <LOSSTANGENT=val>
```

Description

The field solver assigns the following default values for metal:

- `CONDUCTIVITY` = -1 (perfect conductor)
- `ER` = 1
- `UR` = 1

`PEC` is a pre-defined metal name. You cannot redefine its default values.

The field solver assigns the following default values for dielectrics:

- `CONDUCTIVITY` = 0 (lossless dielectric)
- `LOSSTANGENT` = 0 (lossless dielectric)
- `ER` = 1
- `UR` = 1

`AIR` is a pre-defined dielectric name. You cannot redefine its default values.

Because the field solver does not currently support magnetic materials, it ignores `UR` values.

Argument	Definition
<code>mname</code>	Material name.
<code>METAL DIELECTRIC</code>	Material type: METAL or DIELECTRIC.
<code>ER</code>	Dielectric constant (relative permittivity).
<code>UR</code>	Relative permeability.
<code>CONDUCTIVITY</code>	Static field conductivity of conductor or lossy dielectric (S/m).
<code>LOSSTANGENT</code>	Alternating field loss tangent of dielectric ($\tan \delta$).

See Also

[.LAYERSTACK](#)

.MEASURE

Description

Use the `.MEASURE` statement to modify information and to define the results of successive HSPICE or HSPICE RF simulations. The `.MEASURE` statement prints user-defined electrical specifications of a circuit. Optimization (HSPICE only) uses `.MEASURE` statements extensively. The specifications include:

- propagation
- delay
- rise time
- fall time
- peak-to-peak voltage
- minimum and maximum voltage over a specified period
- other user-defined variables

You can also use `.MEASURE` with either the error function (`ERRfun`) or `GOAL` parameter to optimize circuit component values (HSPICE only), and to curve-fit measured data to model parameters.

The `.MEASURE` statement can use several different formats, depending on the application. You can use it for either DC sweep, AC, or transient analyses.

See Also

- [.AC](#)
- [.DC](#)
- [.DCMATCH](#)
- [.DOUT](#)
- [.GRAPH](#)
- [.OPTION MEASDGT](#)
- [.OPTION MEASFAIL](#)
- [.OPTION MEASFILE](#)
- [.OPTION MEASSORT](#)
- [.OPTION MEASOUT](#)
- [.PLOT](#)
- [.PRINT](#)
- [.PROBE](#)
- [.STIM](#)
- [.TRAN](#)

.MEASURE (Rise, Fall, and Delay Measurements)

Syntax

```
.MEASURE <DC | AC | TRAN> result TRIG ... TARG ...  
+ <GOAL = val> <MINVAL = val> <WEIGHT = val>
```

The input syntax for delay, rise time, and fall time in HSPICE RF is:

```
.MEASURE <TRAN > varname TRIG_SPEC TARG_SPEC
```

In this syntax, *varname* is the user-defined variable name for the measurement (the time difference between TRIG and TARG events). The input syntax of *TRIG_SPEC* and *TARG_SPEC* is:

```
TRIG var VAL = val < TD = td > < CROSS = c | LAST >  
+ < RISE = r | LAST > < FALL = f | LAST >  
+ <TRIG AT = time>
```

```
TARG var VAL = val < TD = td > < CROSS = c | LAST >  
+ < RISE = r | LAST > < FALL = f | LAST >  
+ <TRIG AT = time>
```

Example 1

```
* Example of rise/fall/delay measurement  
.MEASURE TRAN tdlay TRIG V(1) VAL = 2.5 TD = 10n  
+ RISE = 2 TARG V(2) VAL = 2.5 FALL = 2
```

This example measures the propagation delay between nodes 1 and 2 for a transient analysis. HSPICE measures the delay from the second rising edge of the voltage at node 1 to the second falling edge of node 2. The measurement begins when the second rising voltage at node 1 is 2.5 V, and ends when the second falling voltage at node 2 is 2.5 V. The $TD = 10n$ parameter counts the crossings, after 10 ns has elapsed. HSPICE prints results as *tdlay = <value>*.

Example 2

```
.MEASURE TRAN riset TRIG I(Q1) VAL = 0.5m RISE = 3  
+ TARG I(Q1) VAL = 4.5m RISE = 3  
* Rise/fall/delay measure with TRIG and TARG specs  
.MEASURE pwidth TRIG AT = 10n TARG V(IN) VAL = 2.5  
+ CROSS = 3
```

2: Commands in HSPICE Netlists

.MEASURE (Rise, Fall, and Delay Measurements)

In the last example, `TRIG. AT = 10n` starts measuring time at $t = 10$ ns in the transient analysis. The `TARG` parameters end time measurement when $V(IN) = 2.5$ V, on the third crossing. `pwidth` is the printed output variable.

If you use the `.TRAN` analysis statement with a `.MEASURE` statement, do not use a non-zero start time in `.TRAN` statement or the `.MEASURE` results might be incorrect.

Example 3

```
.MEAS TRAN TDEL12 TRIG V(signal1) VAL='VDD/2'  
+ RISE=10 TARG V(signal2) VAL='VDD/2' RISE=1 TD=TRIG
```

This example shows a target that is delayed until the trigger time before the target counts the edges.

Description

Use the Rise, Fall, and Delay form of the `.MEASURE` statement to measure independent-variable (time, frequency, or any parameter or temperature) differential measurements such as rise time, fall time, slew rate, or any measurement that requires determining independent variable values. This format specifies `TRIG` and `TARG` substatements. These two statements specify the beginning and end of a voltage or current amplitude measurement.

Argument	Definition
<i>MEASURE</i>	Specifies measurements. You can abbreviate to <i>MEAS</i> .
<i>result</i>	Name associated with the measured value in the HSPICE or HSPICE RF output. This example measures the independent variable, beginning at the trigger, and ending at the target: <ul style="list-style-type: none">• Transient analysis measures time.• AC analysis measures frequency.• DC analysis measures the DC sweep variable. If simulation reaches the target before the trigger activates, the resulting value is negative. Do not use DC, TRAN, or AC as the <i>result</i> name.
<i>TRIG...</i>	Identifies the beginning of trigger specifications.

2: Commands in HSPICE Netlists
 .MEASURE (Rise, Fall, and Delay Measurements)

Argument	Definition
<i>TARG ...</i>	Identifies the beginning of target specifications. The input syntax for delay, rise time, and fall time in HSPICE RF is: <code>.MEASURE < TRAN > varname TRIG_SPEC TARG_SPEC</code> <i>varname</i> is the user-defined variable name for the measurement, the time difference between TRIG and TARG events.
<DC AC TRAN>	Specifies the analysis type of the measurement. If you omit this parameter, HSPICE or HSPICE RF uses the last analysis mode that you requested.
<i>GOAL</i>	Specifies the desired measure value in ERR calculation for optimization. To calculate the error, the simulation uses the equation: $ERRfun = (GOAL - result) / GOAL .$
<i>MINVAL</i>	If the absolute value of GOAL is less than MINVAL, the MINVAL replaces the GOAL value in the denominator of the ERRfun expression. Used only in ERR calculation for optimization. Default = 1.0e-12.
<i>WEIGHT</i>	Multiplies the calculated error by the weight value. Used only in ERR calculation for optimization. Default = 1.0.

TRIG/TARG Parameter	Definition
<i>TRIG</i>	Indicates the beginning of the trigger specification.
<i>trig_val</i>	Value of <i>trig_var</i> , which increments the counter for crossings, rises, or falls, by one.
<i>trig_var</i>	Specifies the name of the output variable, that determines the logical beginning of a measurement. If HSPICE or HSPICE RF reaches the target before the trigger activates, .MEASURE reports a negative value.
<i>TARG</i>	Indicates the beginning of the target signal specification.

2: Commands in HSPICE Netlists

.MEASURE (Rise, Fall, and Delay Measurements)

TRIG/TARG Parameter	Definition
<i>targ_val</i>	Specifies the value of the <i>targ_var</i> , which increments the counter for crossings, rises, or falls, by one.
<i>targ_var</i>	Name of the output variable, at which HSPICE or HSPICE RF determines the propagation delay with respect to the <i>trig_var</i> .
<i>time_delay</i>	Amount of simulation time that must elapse, before HSPICE or HSPICE RF enables the measurement. Simulation counts the number of crossings, rises, or falls, only after the <i>time_delay</i> value. Default trigger delay is zero.

.MEASURE (Average, RMS, and Peak Measurements)

Syntax

```
.MEASURE <TRAN > out_var func var  
+ FROM = start TO = end
```

Example 1

```
.MEAS TRAN RMSVAL RMS V(OUT) FROM = 0NS TO = 10NS
```

In this example, the `.MEASURE` statement calculates the RMS voltage of the OUT node, from 0ns to 10ns. It then labels the result RMSVAL.

Example 2

```
.MEAS MAXCUR MAX I(VDD) FROM = 10NS TO = 200NS
```

In this example, the `.MEASURE` statement finds the maximum current of the VDD voltage supply, between 10ns and 200ns in the simulation. The result is called MAXCUR.

Example 3

```
.MEAS P2P PP PAR('V(OUT)/V(IN)')  
+ FROM = 0NS TO = 200NS
```

In this example, the `.MEASURE` statement uses the ratio of V(OUT) and V(IN) to find the peak-to-peak value in the interval of 0ns to 200ns.

Description

This `.MEASURE` statement reports the average, RMS, or peak value of the specified output variable.

2: Commands in HSPICE Netlists

.MEASURE (Average, RMS, and Peak Measurements)

Argument	Definition
<i>varname</i>	User-defined variable name for the measurement.
<i>func</i>	One of the following keywords: <ul style="list-style-type: none">• AVG: Average area under <i>var</i>, divided by the period of interest.• MAX: Maximum value of <i>var</i> over the specified interval.• MIN: Minimum value of <i>var</i> over the specified interval.• PP: Peak-to-peak: reports the maximum value, minus the minimum of <i>var</i> over the specified interval.• RMS: Root mean squared: calculates the square root of the area under the var^2 curve, divided by the period of interest.• INTEG: Integral of <i>var</i> over the specified period.
<i>out_var</i> <i>var</i>	Name of the output variable, which can be either the node voltage or the branch current of the circuit. You can also use an expression, consisting of the node voltages or the branch current.
<i>start</i>	Starting time of the measurement period.
<i>end</i>	Ending time of the measurement period.

.MEASURE (FIND and WHEN)

Syntax

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

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

.MEASURE <DC | AC | TRAN> 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>

.MEASURE <DC | AC | TRAN> 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>

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

.MEASURE DC result FIND <DCMATCH_TOTAL |
+ DCMATCH(InstanceName)> AT = val
```

Example

```
* MEASURE statement using FIND/WHEN
.MEAS TRAN TRT FIND PAR('V(3)-V(4)')
+ WHEN V(1)=PAR('V(2)/2') RISE = LAST
.MEAS STIME WHEN V(4) = 2.5 CROSS = 3
```

In this example, the first measurement, TRT, calculates the difference between V(3) and V(4) when V(1) is half the voltage of V(2) at the last rise event.

2: Commands in HSPICE Netlists

.MEASURE (FIND and WHEN)

The second measurement, STIME, finds the time when V(4) is 2.5V at the third rise-fall event. A CROSS event is a rising or falling edge.

Description

The FIND and WHEN functions of the .MEASURE statement specify to measure:

- Any independent variables (time, frequency, parameter).
- Any dependent variables (voltage or current, for example).
- Derivative of a dependent variable, if a specific event occurs.

Argument	Definition
<i>CROSS = c</i> <i>RISE = r</i> <i>FALL = f</i>	<p>Numbers indicate which CROSS, FALL, or RISE event to measure. For example:</p> <pre>.meas tran tdlay trig v(1) val=1.5 td=10n + rise=2 targ v(2) val=1.5 fall=2</pre> <p>In the above example, rise=2 specifies to measure the v(1) voltage, only on the first two rising edges of the waveform. The value of these first two rising edges is 1. However, trig v(1) val=1.5 indicates to trigger when the voltage on the rising edge voltage is 1.5, which never occurs on these first two rising edges. So the v(1) voltage measurement never finds a trigger.</p> <ul style="list-style-type: none">• RISE = r, the WHEN condition is met, and measurement occurs after the designated signal has risen r rise times.• FALL = f, measurement occurs when the designated signal has fallen f fall times. <p>A crossing is either a rise or a fall so for CROSS = c, measurement occurs when the designated signal has achieved a total of c crossing times as a result of either rising or falling.</p> <p>For TARG, the LAST keyword specifies the last event.</p>

Argument	Definition
<i>LAST</i>	<p>HSPICE or HSPICE RF measures when the last CROSS, FALL, or RISE event occurs.</p> <ul style="list-style-type: none"> CROSS = LAST, measurement occurs the last time the WHEN condition is true for a rising or falling signal. FALL = LAST, measurement occurs the last time the WHEN condition is true for a falling signal. RISE = LAST, measurement occurs the last time the WHEN condition is true for a rising signal. <p>LAST is a reserved word; you cannot use it as a parameter name in the above .MEASURE statements.</p>
<i>AT = val</i>	<p>Special case for trigger specification. <i>val</i> is:</p> <ul style="list-style-type: none"> Time for TRAN analysis. Frequency for AC analysis. Parameter for DC analysis. <i>SweepValue</i> from .DC mismatch analysis. <p>The trigger determines where measurement takes place.</p>
<i><DC AC TRAN></i>	<p>Analysis type for the measurement. If you omit this parameter, HSPICE or HSPICE RF assumes the last analysis type that you requested.</p>
<i>FIND</i>	<p>Selects the FIND function.</p>
<i>GOAL</i>	<p>Desired .MEASURE value. Optimization uses this value in ERR calculation. The following equation calculates the error:</p> $ERR_{fun} = (GOAL - result) / GOAL .$ <p>In HSPICE RF output, you cannot apply .MEASURE to waveforms generated from another .MEASURE statement in a parameter sweep.</p>
<i>LAST</i>	<p>Starts measurement at the last CROSS, FALL, or RISE event.</p> <ul style="list-style-type: none"> For CROSS = LAST, measurement starts the last time the WHEN condition is true for either a rising or falling signal. For FALL = LAST, measurement starts the last time the WHEN condition is true for a falling signal. For RISE = LAST, measurement starts the last time the WHEN condition is true for a rising signal. <p>LAST is a reserved word. Do not use it as a parameter name in these .MEASURE statements.</p>

2: Commands in HSPICE Netlists

.MEASURE (FIND and WHEN)

Argument	Definition
<i>MINVAL</i>	If the absolute value of GOAL is less than MINVAL, then MINVAL replaces the GOAL value in the denominator of the ERRfun expression. Used only in ERR calculation for optimization. Default = 1.0e-12.
<i>out_var(1,2,3)</i>	These variables establish conditions that start a measurement.
<i>result</i>	Name of a measured value in the HSPICE or HSPICE RF output.
<i>TD</i>	Time at which measurement starts.
<i>WEIGHT</i>	Multiplies the calculated error by the weight value. Used only in ERR calculation for optimization. Default = 1.0.
<i>WHEN</i>	Selects the WHEN function.
DCMATCH (<i>InstanceName</i>)	.DCMATCH contribution from <i>InstanceName</i> .
DCMATCH_TOTAL	.DCMATCH total output variation.

.MEASURE (Equation Evaluation/ Arithmetic Expression)

Syntax

```
.MEASURE <DC | TRAN | AC> result PARAM = 'equation'  
+ <GOAL = val> <MINVAL = val>  
  
.MEASURE TRAN varname PARAM = "expression"
```

Example

```
.MEAS TRAN V3MAX MAX V(3) FROM 0NS TO 100NS  
.MEAS TRAN V2MIN MIN V(2) FROM 0NS TO 100NS  
.MEAS VARG PARAM = '(V2MIN + V3MAX)/2'
```

The first two measurements, V3MAX and V2MIN, set up the variables for the third .MEASURE statement.

- V3MAX is the maximum voltage of V(3) between 0ns and 100ns of the simulation.
- V2MIN is the minimum voltage of V(2) during that same interval.
- VARG is the mathematical average of the V3MAX and V2MIN measurements.

Description

Use the Equation Evaluation form of the .MEASURE statement to evaluate an equation, that is a function of the results of previous .MEASURE statements. The equation must not be a function of node voltages or branch currents.

The *expression* option is an arithmetic expression, that uses results from other prior .MEASURE statements.

Expressions used in arithmetic expression must not be a function of node voltages or branch currents. Expressions used in all other .MEASURE statements can contain either node voltages or branch currents, but must not use results from other .MEASURE statements.

2: Commands in HSPICE Netlists

.MEASURE (Average, RMS, MIN, MAX, INTEG, and PP)

.MEASURE (Average, RMS, MIN, MAX, INTEG, and PP)

Syntax

```
.MEASURE <DC | AC | TRAN> result func out_var  
+ <FROM = val> <TO = val> <GOAL = val>  
+ <MINVAL = val> <WEIGHT = val>  
  
.MEASURE DC results <MAX>  
+ <DCMATCH_TOTAL | DCMATCH(InstanceName)>
```

Example 1

```
.MEAS TRAN avgval AVG V(10) FROM = 10ns TO = 55ns
```

This example calculates the average nodal voltage value for node 10, during the transient sweep, from the time 10 ns to 55 ns. It prints out the result as *avgval*.

Example 2

```
.MEAS TRAN MAXVAL MAX V(1,2) FROM = 15ns TO = 100ns
```

This example finds the maximum voltage difference between nodes 1 and 2 for the time period from 15 ns to 100 ns.

Example 3

```
.MEAS TRAN MINVAL MIN V(1,2) FROM = 15ns TO = 100ns  
.MEAS TRAN P2PVAL PP I(M1) FROM = 10ns TO = 100ns
```

Description

Average (AVG), RMS, MIN, MAX, and peak-to-peak (PP) measurement modes report statistical functions of the output variable, rather than analysis values.

- AVG calculates the area under an output variable, divided by the periods of interest.
- RMS divides the square root of the area under the output variable square, by the period of interest.
- MIN reports the minimum value of the output function, over the specified interval.
- MAX reports the maximum value of the output function, over the specified interval.
- PP (peak-to-peak) reports the maximum value, minus the minimum value, over the specified interval.

AVG, RMS, and INTEG have no meaning in a DC data sweep so if you use them, HSPICE or HSPICE RF issues a warning message.

Argument	Definition
<i><DC AC TRAN></i>	Specifies the analysis type for the measurement. If you omit this parameter, HSPICE or HSPICE RF assumes the last analysis mode that you requested.
<i>FROM</i>	Specifies the initial value for the <i>func</i> calculation. For transient analysis, this value is in units of time.
<i>TO</i>	Specifies the end of the <i>func</i> calculation.
<i>GOAL</i>	Specifies the .MEASURE value. Optimization uses this value for ERR calculation. This equation calculates the error: $ERRfun = (GOAL - result) / GOAL$ In HSPICE RF simulation output, you cannot apply .MEASURE to waveforms generated from another .MEASURE statement in a parameter sweep.
<i>func</i>	Indicates one of the measure statement types: <ul style="list-style-type: none"> • AVG (average): Calculates the area under the out_var, divided by the periods of interest. • MAX (maximum): Reports the maximum value of the out_var, over the specified interval. • MIN (minimum): Reports the minimum value of the out_var, over the specified interval. • PP (peak-to-peak): Reports the maximum value, minus the minimum value of the out_var, over the specified interval. • RMS (root mean squared): Calculates the square root of the area under the out_var2 curve, divided by the period of interest.
<i>result</i>	Name of the measured value in the output. The value is a function of the variable (<i>out_var</i>) and <i>func</i> .
<i>out_var</i>	Name of any output variable whose function (<i>func</i>) the simulation measures.
<i>WEIGHT</i>	Multiplies the calculated error, by the weight value. Used only in ERR calculation for optimization. Default = 1.0.

2: Commands in HSPICE Netlists

.MEASURE (Average, RMS, MIN, MAX, INTEG, and PP)

Argument	Definition
DCMATCH (<i>InstanceName</i>)	.DCMATCH contribution from <i>InstanceName</i> .
DCMATCH_TOT AL	.DCMATCH total output variation.

.MEASURE (Integral Function)

Syntax

```
.MEASURE <DC | AC | TRAN> result INTEGRAL out_var  
+ <FROM = val> <TO = val> <GOAL = val>  
+ <MINVAL = val> <WEIGHT = val>
```

Example

```
.MEAS TRAN charge INTEG I(cload) FROM = 10ns  
+ TO = 100ns
```

This example calculates the integral of $I(cload)$, from 10 ns to 100 ns.

Description

The INTEGRAL function reports the integral of an output variable, over a specified period.

The INTEGRAL function (with func), uses the same syntax as the average (AVG), RMS, MIN, MAX, and peak-to-peak (PP) measurement mode to defined the INTEGRAL (INTEG).

2: Commands in HSPICE Netlists

.MEASURE (Derivative Function)

.MEASURE (Derivative Function)

Syntax

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

```
.MEASURE <DC | AC | TRAN> result DERIVATIVE out_var  
+ WHEN var2 = val <RISE = r | LAST>  
+ <FALL = f | LAST> <CROSS = c | LAST> <TD = tdval>  
+ <GOAL = goalval> <MINVAL = minval>  
+ <WEIGHT = weightval>
```

```
.MEASURE <DC | AC | TRAN> result DERIVATIVE out_var  
+ WHEN var2 = var3 <RISE = r | LAST>  
+ <FALL = f | LAST> <CROSS = c | LAST> <TD = tdval>  
+ <GOAL = goalval> <MINVAL = minval>  
+ <WEIGHT = weightval>
```

Example 1

```
.MEAS TRAN slew rate DERIV V(out) AT = 25ns
```

This example calculates the derivative of V(out), at 25 ns.

Example 2

```
.MEAS TRAN slew DERIV v(1) WHEN v(1) = '0.90*vdd'
```

This example calculates the derivative of v(1) when v(1) is equal to 0.9*vdd.

Example 3

```
.MEAS AC delay DERIV 'VP(output)/360.0' AT = 10khz
```

This example calculates the derivative of VP(output)/360.0 when the frequency is 10 kHz.

Description

The DERIVATIVE function provides the derivative of:

- An output variable, at a specified time or frequency.
- Any sweep variable, depending on the type of analysis.
- A specified output variable when some specific event occurs.

Argument	Definition
<i>AT = val</i>	Value of <i>out_var</i> , at which the derivative is found.
<i>CROSS = c</i> <i>RISE = r</i> <i>FALL = f</i>	<p>The numbers indicate which occurrence of a CROSS, FALL, or RISE event starts a measurement.</p> <ul style="list-style-type: none"> For RISE = r when the designated signal has risen r rise times, the WHEN condition is met, and measurement starts. For FALL = f, measurement starts when the designated signal has fallen f fall times. <p>A crossing is either a rise or a fall so for CROSS = c, measurement starts when the designated signal has achieved a total of c crossing times as a result of either rising or falling.</p>
<DC AC TRAN>	Specifies the analysis type to measure. If you omit this parameter, HSPICE or HSPICE RF assumes the last analysis mode that you requested.
<i>DERIVATIVE</i>	Selects the derivative function. You can abbreviate to DERIV.
<i>GOAL</i>	<p>Specifies the desired .MEASURE value. Optimization uses this value for ERR calculation. This equation calculates the error:</p> $ERRfun = (GOAL - result)/GOAL$ <p>In HSPICE RF output, you cannot apply .MEASURE to waveforms generated from another .MEASURE statement in a parameter sweep.</p>
<i>LAST</i>	<p>Measures when the last CROSS, FALL, or RISE event occurs.</p> <ul style="list-style-type: none"> CROSS = LAST, measures the last time the WHEN condition is true for a rising or falling signal. FALL = LAST, measures the last time WHEN is true for a falling signal. RISE = LAST, measures the last time WHEN is true for a rising signal. <p>LAST is a reserved word; do not use it as a parameter name in the above .MEASURE statements.</p>
<i>MINVAL</i>	<p>If the absolute value of GOAL is less than MINVAL, MINVAL replaces the GOAL value in the denominator of the ERRfun expression. Used only in ERR calculation for optimization. Default = 1.0e-12.</p>

2: Commands in HSPICE Netlists

.MEASURE (Derivative Function)

Argument	Definition
<i>out_var</i>	Variable for which HSPICE or HSPICE RF finds the derivative.
<i>result</i>	Name of the measured value in the output.
<i>TD</i>	Identifies the time when measurement starts.
<i>var(2,3)</i>	These variables establish conditions that start a measurement.
<i>WEIGHT</i>	Multiplies the calculated error, between result and GOAL, by the weight value. Used only in ERR calculation for optimization. Default = 1.0.
<i>WHEN</i>	Selects the WHEN function.

.MEASURE (Error Function)

Syntax

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

Description

The relative error function reports the relative difference between two output variables. You can use this format in optimization and curve-fitting of measured data. The relative error format specifies the variable to measure and calculate, from the .PARAM variable. To calculate the relative error between the two, HSPICE or HSPICE RF uses the ERR, ERR1, ERR2, or ERR3 functions. With this format, you can specify a group of parameters to vary to match the calculated value and the measured data.

Argument	Definition
<DC AC TRAN>	Specifies the analysis type for the measurement. If you omit this parameter, HSPICE or HSPICE RF assumes the last analysis mode that you requested.
<i>result</i>	Name of the measured result in the output.
<i>ERRfun</i>	ERRfun indicates which error function to use: ERR, ERR1, ERR2, or ERR3.
<i>meas_var</i>	Name of any output variable or parameter in the data statement. <i>M</i> denotes the <i>meas_var</i> in the error equation.
<i>calc_var</i>	Name of the simulated output variable or parameter in the .MEASURE statement to compare with <i>meas_var</i> . <i>C</i> is the <i>calc_var</i> in the error equation.
<i>IGNOR YMIN</i>	If the absolute value of <i>meas_var</i> is less than the IGNOR value, then the ERRfun calculation does not consider this point. Default = 1.0e-15.

2: Commands in HSPICE Netlists

.MEASURE (Error Function)

Argument	Definition
<i>FROM</i>	Specifies the beginning of the ERRfun calculation. For transient analysis, the <i>from</i> value is in units of time. Defaults to the first value of the sweep variable.
<i>WEIGHT</i>	Multiplies the calculated error, by the weight value. Used only in ERR calculation for optimization. Default = 1.0.
<i>YMAX</i>	If the absolute value of meas_var is greater than the YMAX value, then the ERRfun calculation does not consider this point. Default = 1.0e+15.
<i>TO</i>	End of the ERRfun calculation. Default is last value of the sweep variable.
<i>MINVAL</i>	If the absolute value of meas_var is less than MINVAL, MINVAL replaces the meas_var value in the denominator of the ERRfun expression. Used only in ERR calculation for optimization. Default = 1.0e-12.

.MEASURE (Pushout Bisection)

Syntax

```
.MEASURE TRAN MeasureName MeasureClause  
pushout=time <lower/upper>
```

-or-

```
.MEASURE TRAN MeasureName MeasureClause  
pushout_per=percentage <lower/upper>
```

Example 1

```
.Param DelayTime = Opt1 ( 0.0n, 0.0n , 5.0n )  
.Tran 1n 8n Sweep Optimize=Opt1 Result=setup_prop + Model=OptMod  
.Measure Tran setup_prop Trig v(data)  
+ Val = 'v(Vdd) 2' fall = 1 Targ v(D_Output)  
+ Val = 'v(Vdd)' rise = 1 pushout=1.5n lower
```

In this example, the parameter to be optimized is Delaytime and the evaluation goal is setup_prop. The Pushout=1.5 lower means that the setup_prop of the final solution is not 1.5n far from the setup_prop of the lower bound of the parameter (0.0n).

Example 2

```
.Measure Tran setup_prop Trig v(data)  
+ Val = 'v(Vdd)/2' fall = 1 Targ v(D_Output)  
+ Val = 'v(Vdd)' rise = 1 pushout_per=0.1 lower
```

In this example, the differences between the setup_prop of the final solution and that of the lower bound of the parameter (0.0n) is not more than 10%.

Description

Pushout is only employed in bisection analysis. In Pushout Bisection, instead of finding the last point just before failure, you specify a maximum allowed pushout time to control the distance from failure.

2: Commands in HSPICE Netlists

.MEASURE (Pushout Bisection)

Argument	Definition
pushout=time	Specifies the time. An appropriate time must be specified to obtain the pushout result (an absolute time).
pushout_per= percentage	Defines a relative error. If you specify a 0.1 relative error, the T_lower or T_upper and T_pushout have more than a 10% difference in value. This occurrence causes the iteration to stop and output the optimized parameter.
lower/upper	Specifies the parameter boundary values for pushout comparison. These arguments are optional. If the parameter is defined as .PARAM <ParamName>=OPTxxx(<Initial>, <min>. <max>), the "lower" means the lower bound "min", and the "upper" means the upper bound "max". Default=lower.

.MODEL

Syntax

```
.MODEL mname type <VERSION = version_number>  
+ <pname1 = val1 pname2 = val2 ...>  
  
.MODEL mname OPT <parameter=val ...>
```

The following is the .MODEL syntax for use with .GRAPH:

```
.MODEL mname PLOT (pname1 = val1 pname2 = val2...)
```

The following syntax is used for a Monte Carlo analysis:

```
.MODEL mname ModelType (<LEVEL=val>  
+ <keyname1=val1><keyname2=val2>  
+ <keyname3=val3><LOT</n></distribution>><value>  
+ <DEV</n></distribution>><value>...)  
+ <VERSION=version_number>
```

Example 1

```
.MODEL MOD1 NPN BF=50 IS=1E-13 VBF=50 AREA=2 PJ=3,  
+ N=1.05
```

Example 2

This example shows a .MODEL statement used for a Monte Carlo analysis:

```
.model m1 nmos level=6 bulk=2 vt=0.7 dev/2 0.1  
+ tox=520 lot/gauss 0.3 a1=.5 a2=1.5 cdb=10e-16  
+ csb=10e-16 tcv=.0024
```

Description

Use the .MODEL command to include an instance (element) of a pre-defined HSPICE model in your input netlist.

For each optimization within a data file, specify a .MODEL statement. HSPICE can then execute more than one optimization per simulation run. The .MODEL optimization statement defines:

- Convergence criteria.
- Number of iterations.
- Derivative methods.

2: Commands in HSPICE Netlists

.MODEL

Argument	Definition
<i>mname</i>	<p>Model name reference. Elements must use this name to refer to the model.</p> <p>If model names contain periods (.), the automatic model selector might fail.</p> <p>When used with .GRAPH, this is the plot model name, referenced in .GRAPH statements.</p>
<i>type</i>	<p>Selects a model type. Must be one of the following.</p> <ul style="list-style-type: none">AMP operational amplifier modelC capacitor modelCORE magnetic core modelD diode modelL inductor model or magnetic core mutual inductor modelNJF n-channel JFET modelNMOS n-channel MOSFET modelNPN npn BJT modelOPT optimization modelPJF p-channel JFET modelPLOT plot model for the .GRAPH statementPMOS p-channel MOSFET modelPNP pnp BJT modelR resistor modelU lossy transmission line model (lumped)W lossy transmission line modelSP S parameter

Argument	Definition
CENDIF	<p>Selects different derivative methods. Default=1.0e-9.</p> <p>The following calculates the gradient of the RESULTS functions: $\text{Transpose}(\text{Jacobi}(F(X))) * F(X)$, where $F(X)$ is the RESULT function</p> <p>If the resulting gradient is less than CENDIF, HSPICE uses more accurate but more time-consuming derivative methods. By default, HSPICE uses faster but less-accurate derivative methods. To use the more-accurate methods, set CENDIF to a larger value than GRAD.</p> <p>If the gradient of the RESULTS function is less than GRAD, optimization finishes before CENDIF takes effect.</p> <ul style="list-style-type: none"> • If the value is too large, the optimizer requires more CPU time. • If the value is too small, the optimizer might not find as accurate an answer.
CLOSE	<p>Initial estimate of how close parameter initial value estimates are to the solution. CLOSE multiplies changes in new parameter estimates. If you use a large CLOSE value, the optimizer takes large steps toward the solution. For a small value, the optimizer takes smaller steps toward the solution. You can use a smaller value for close parameter estimates, and a larger value for rough initial guesses. Default=1.0.</p> <ul style="list-style-type: none"> • If CLOSE is greater than 100, the steepest descent in the Levenburg-Marquardt algorithm dominates. • If CLOSE is less than 1, the Gauss-Newton method dominates. <p>For more details, see L. Spruiell, "Optimization Error Surfaces," <i>Meta-Software Journal</i>, Volume 1, Number 4, December 1994.</p>
CUT	<p>Modifies CLOSE, depending on how successful iterations are, toward the solution.</p> <p>If the last iteration succeeds, descent toward the CLOSE solution decreases by the CUT value. That is, $\text{CLOSE} = \text{CLOSE} / \text{CUT}$</p> <p>If the last iteration was not a successful descent to the solution, CLOSE increases by CUT squared. That is, $\text{CLOSE} = \text{CLOSE} * \text{CUT} * \text{CUT}$</p> <p>CUT drives CLOSE up or down, depending on the relative success in finding the solution. The CUT value must be > 1. Default = 2.0.</p>
DEV	<p>(Monte Carlo) DEV tolerance, which is independent (each device varies independently).</p>

2: Commands in HSPICE Netlists

.MODEL

Argument	Definition
DIFSIZ	Increment change in a parameter value for gradient calculations ($\Delta x = DIFSIZ \cdot \max(x, 0.1)$). If you specify delta in a .PARAM statement, then $\Delta x = \text{delta}$. Default = 1e-3.
<i>distribution</i>	(Monte Carlo) The distribution function name, which must be specified as GAUSS, AGAUSS, LIMIT, UNIF, or AUNIF. If you do not set the distribution function, the default distribution function is used. The default distribution function is uniform distribution.
GRAD	Represents possible convergence, if the gradient of the RESULTS function is less than GRAD. Most applications use values of 1e-6 to 1e-5. Too large a value can stop the optimizer before finding the best solution. Too small a value requires more iterations. Default=1.0e-6.
ITROPT	Maximum number of iterations. Typically, you need no more than 20-40 iterations to find a solution. Too many iterations can imply that the RELIN, GRAD, or RELOUT values are too small. Default=20.
LEVEL	Selects an optimizing algorithm. <ul style="list-style-type: none">• LEVEL=1 specifies the Modified Levenberg-Marquardt method. You would use this setting with multiple optimization parameters and goals.• LEVEL=2 specifies the BISECTION method in HSPICE RF. You would use this setting with one optimization parameter.• LEVEL=3 specifies the PASSFAIL method. You would use this setting with two optimization parameter. This argument is ignored when METHOD has been specified.
LOT	(Monte Carlo) The LOT tolerance, which requires all devices that refer to the same model use the same adjustments to the model parameter.
LOT/n DEV/n	(Monte Carlo) Specifies which of ten random number generators numbered 0 through 9 are used to calculate parameter value deviations. This correlates deviations between parameters in the same model as well as between models. The generators for DEV and LOT tolerances are distinct: Ten generators exist for both DEV tracking and LOT tracking. N must be an integer 0 to 9.
<i>keyword</i>	(Monte Carlo) Model parameter keyword.
MAX	Sets the upper limit on CLOSE. Use values > 100. Default=6.0e+5.

Argument	Definition
METHOD	<p>Specifies an optimization method.</p> <ul style="list-style-type: none"> METHOD = LM specifies the Modified Levenberg-Marquardt method. METHOD = BISECTION specifies the Bisection method. METHOD = PASSFAIL specifies the Passfail method. <p>This argument supersedes LEVEL when present.</p>
PARMIN	<p>Allows better control of incremental parameter changes, during error calculations. Default=0.1. This produces more control over the trade-off between simulation time and optimization result accuracy. To calculate parameter increments, HSPICE uses the relationship:</p> $Dpar_val = DIFSIZ \cdot \text{MAX}(par_val, \text{PARMIN})$
PLOT	<p>A .GRAPH statement model.</p>
<i>pname1 ...</i>	<p>Parameter name. Assign a model parameter name (<i>pname1</i>) from the parameter names for the appropriate model type. Each model section provides default values. For legibility, enclose the parameter assignment list in parentheses, and use either blanks or commas to separate each assignment. Use a plus sign (+) to start a continuation line.</p> <p>When used with .GRAPH, each .GRAPH statement includes several model parameters. If you do not specify model parameters, HSPICE uses the default values of the model parameters, described in the following table. <i>Pnamn</i> is one of the model parameters of a .GRAPH statement, and <i>valn</i> is the value of pnamn. Valn can be more than one parameter.</p>
RELIN	<p>Sets the relative input parameter ($\text{delta_par_val} / \text{MAX}(par_val, 1e-6)$) for convergence. If all optimizing input parameters vary by no more than RELIN between iterations, the solution converges. RELIN is a relative variance test so a value of 0.001 implies that optimizing parameters vary by less than 0.1%, from one iteration to the next. Default=0.001.</p>
RELOUT	<p>Sets the relative tolerance to finish optimization. For RELOUT=0.001, if the relative difference in the RESULTS functions, from one iteration to the next, is less than 0.001, then optimization is finished. Default=0.001.</p>

2: Commands in HSPICE Netlists

.MODEL

Argument	Definition
<i>VERSION</i>	<p>HSPICE or HSPICE RF version number. Allows portability of the BSIM (LEVEL=13) and BSIM2 (LEVEL = 39) models, between HSPICE releases. HSPICE release numbers, and the corresponding version numbers, are:</p> <p>HSPICE <i>release</i> <i>Version number</i></p> <p>9007B 9007.02 9007D 9007.04 92A 92.01 92B 92.02 93A 93.01 93A.02 93.02 95.3 95.3 96.1 96.1</p> <p>The <i>VERSION</i> parameter is valid only for LEVEL 13 and LEVEL 39 models. Use it with HSPICE Release H93A.02 and higher. If you use the parameter with any other model, or with a release before H93A.02, HSPICE issues a warning, but the simulation continues. You can also use <i>VERSION</i> to denote the BSIM3v3 version number only in model LEVELs 49 and 53. For LEVELs 49 and 53, the <i>HSPVER</i> parameter denotes the HSPICE or HSPICE RF release number.</p>

.NET

Syntax

One-Port Network

```
.NET input <RIN = val>
```

```
.NET input <val>
```

Two-Port Network

```
.NET output input <ROUT = val> <RIN = val>
```

Example

One-Port Network

```
.NET VINAC RIN = 50  
.NET IIN RIN = 50
```

Two-Port Network

```
.NET V(10,30) VINAC ROUT = 75 RIN = 50  
.NET I(RX) VINAC ROUT = 75 RIN = 50
```

Description

You can use the `.NET` statement or HSPICE RF to compute parameters for:

- Z impedance matrix.
- Y admittance matrix.
- H hybrid matrix
- S scattering matrix.

You can use the `.NET` statement only in conjunction with the `.AC` statement.

HSPICE or HSPICE RF also computes:

- Input impedance.
- Output impedance.
- Admittance.

This analysis is part of AC small-signal analysis. To run network analysis, specify the frequency sweep for the `.AC` statement.

2: Commands in HSPICE Netlists

.NET

Argument	Definition
<i>input</i>	Name of the voltage or current source for AC input.
<i>output</i>	Output port. It can be: <ul style="list-style-type: none">• An output voltage, $V(n1<,n2>)$.• An output current, $I(\text{source})$, or $I(\text{element})$.
<i>RIN</i>	Input or source resistance. RIN calculates output impedance, output admittance, and scattering parameters. The default RIN value is 1 ohm.
<i>ROUT</i>	Output or load resistance. ROUT calculates input impedance, admittance, and scattering parameters. Default=1 ohm.

See Also

[.AC](#)

.NODESET

Syntax

```
.NODESET V(node1) = val1 <V(node2) = val2 ...>
```

or

```
.NODESET node1 val1 <node2 val2>
```

Example

```
.NODESET V(5:SETX) = 3.5V V(X1.X2.VINT) = 1V  
.NODESET V(12) = 4.5 V(4) = 2.23  
.NODESET 12 4.5 4 2.23 1 1
```

Description

The `.NODESET` statement initializes all specified nodal voltages for DC operating point analysis. Use the `.NODESET` statement to correct convergence problems in DC analysis. If you set the node values in the circuit close to the actual DC operating point solution, you enhance convergence of the simulation. The HSPICE or HSPICE RF simulator uses the `NODESET` voltages only in the first iteration to set an initial guess for DC operating point analysis.

Argument	Definition
node1 ...	Node numbers or names can include full paths or circuit numbers.
val1	Specifies voltages.

See Also

[.DC](#)

.NOISE

Syntax

```
.NOISE ovv srcnam inter
```

Example

```
.NOISE V(5) VIN 10
```

Description

Use the `.NOISE` and `.AC` statements to control the noise analysis of the circuit. You can use the `.NOISE` statement only in conjunction with the `.AC` statement.

Argument	Definition
<i>ovv</i>	Nodal voltage output variable. Defines the node at which HSPICE or HSPICE RF sums the noise.
<i>srcnam</i>	Name of the independent voltage or current source to use as the noise input reference
<i>inter</i>	Interval at which HSPICE or HSPICE RF prints a noise analysis summary. <i>inter</i> specifies how many frequency points to summarize in the AC sweep. If you omit <i>inter</i> , or set it to zero, HSPICE or HSPICE RF does not print a summary. If <i>inter</i> is equal to or greater than one, HSPICE or HSPICE RF prints summary for the first frequency, and once for each subsequent increment of the <i>inter</i> frequency. The noise report is sorted according to the contribution of each node to the overall noise level.

See Also

[.AC](#)

.OP

Syntax

```
.OP <format> <time> <format> <time>... <interpolation>
```

Example 1

```
.OP .5NS CUR 10NS VOL 17.5NS 20NS 25NS
```

This example calculates:

- Operating point at .05ns.
- Currents at 10 ns for the transient analysis.
- Voltages at 17.5 ns, 20 ns and 25 ns for the transient analysis.

Example 2

```
.OP
```

This example calculates a complete DC operating point solution.

Description

When you include an `.OP` statement in an input file, HSPICE or HSPICE RF calculates the DC operating point of the circuit. You can also use the `.OP` statement to produce an operating point during a transient analysis. You can include only one `.OP` statement in a simulation.

If an analysis requires calculating an operating point, you do not need to specify the `.OP` statement; HSPICE or HSPICE RF calculates an operating point. If you use a `.OP` statement, and if you include the `UIC` parameter in a `.TRAN` analysis statement, then simulation omits the `time = 0` operating point analysis, and issues a warning in the output listing.

2: Commands in HSPICE Netlists

.OP

Argument	Definition
format	<p>Any of the following keywords. Only the first letter is required. Default = ALL</p> <ul style="list-style-type: none">• ALL: Full operating point, including voltage, currents, conductances, and capacitances. This parameter outputs voltage/current for the specified time.• BRIEF: Produces a one-line summary of each element's voltage, current, and power. Current is stated in milliamperes, and power is in milliwatts.• CURRENT: Voltage table with a brief summary of element currents and power.• DEBUG: Usually invoked only if a simulation does not converge. Debug prints the non-convergent nodes, with the new voltage, old voltage, and the tolerance (degree of non-convergence). It also prints the non-convergent elements with their tolerance values.• NONE: Inhibits node and element printouts, but performs additional analysis that you specify.• VOLTAGE: Voltage table only. <p>The preceding keywords are mutually-exclusive; use only one at a time.</p>
time	<p>Place this parameter directly after ALL, VOLTAGE, CURRENT, or DEBUG. It specifies the time at which HSPICE or HSPICE RF prints the report. HSPICE RF returns node voltages only if time (t) is 0.</p>
interpolation	<p>Selects the interpolation method for .OP time points during transient analysis, or no interpolation. Only the first character is required; that is, typing <i>i</i> has the same effect as typing <i>interpolation</i>. Default is not active.</p> <p>If you specify <i>interpolation</i>, all of the time points in the .OP statement (except time=0) use the interpolation method to calculate the OP value during the transient analysis. If you use this keyword, it must be at the end of the .OP statement. HSPICE ignores any word after this keyword.</p>

See Also

[.TRAN](#)

.OPTION

Syntax

```
.OPTION opt1 <opt2 opt3 ...>
```

Argument	Definition
opt1 ...	Specifies input control options. Many options are in the form <code><opt> = x</code> , where <code><opt></code> is the option name and <code>x</code> is the value assigned to that option. Options are described in detail in Chapter 3, Options in HSPICE Netlists .

Example

```
.OPTION BRIEF $ Sets BRIEF to 1 (turns it on)
* Netlist, models,
...
.OPTION BRIEF = 0 $ Turns BRIEF off
```

This example sets the `BRIEF` option to 1 to suppress a printout. It then resets `BRIEF` to 0 later in the input file to resume the printout.

Description

You use the `.OPTION` command to modify various aspects of a Synopsys HSPICE or HSPICE RF simulation, including:

- output types
- accuracy
- speed
- convergence

You can set any number of options in one `.OPTION` statement, and you can include any number of `.OPTION` statements in an input netlist file. Most options default to 0 (OFF) when you do not assign a value by using either `.OPTION <opt> = <val>` or the option with no assignment: `.OPTION <opt>`.

To reset options, set them to 0 (`.OPTION <opt> = 0`). To redefine an option, enter a new `.OPTION` statement; HSPICE or HSPICE RF uses the last definition.

You can use the following types of options with this command. For detailed information on individual options, see [Chapter 3, Options in HSPICE Netlists](#).

2: Commands in HSPICE Netlists

.OPTION

- [General Control Options](#)
- [CPU Options](#)
- [Interface Options](#)
- [Analysis Options](#)
- [Error Options](#)
- [Version Option](#)
- [Model Analysis Options](#)
- [DC Operating Point, DC Sweep, and Pole/Zero Options](#)
- [Transient and AC Small Signal Analysis Options](#)
- [Transient Control Options](#)
- [Input/Output Options](#)
- [AC Control Options](#)
- [Common Model Interface Options](#)
- [Verilog-A Options](#)

For instructions on how to use options that are relevant to a specific simulation type, see the appropriate DC, transient, and AC analysis chapters in the *HSPICE Simulation and Analysis User Guide*.

.PARAM

Syntax

Simple parameter assignment:

```
.PARAM <ParamName>=<RealNumber>
```

Algebraic parameter assignments:

```
.PARAM <ParamName>='<AlgebraicExpression>'
```

```
.PARAM <ParamName1>=<ParamName2>
```

User-defined functions:

```
.PARAM <ParamName>(<pv1>[<pv2>])='<Expression>'
```

Pre-defined analysis functions:

```
.PARAM <FunctionName> = <Value>
```

Optimized parameter assignment:

```
.PARAM parameter=OPTxxx (initial_guess, low_limit,  
+ upper_limit)
```

```
.PARAM parameter=OPTxxx (initial_guess, low_limit,  
+ upper_limit, delta)
```

```
.PARAM <paramname>=str('string')
```

Example 1

```
* Simple parameter assignment  
.PARAM power_cycles=256
```

Example 2

```
* Numerical parameter assignment  
.PARAM TermValue = 1g  
  rTerm Bit0 0 TermValue  
  rTerm Bit1 0 TermValue  
...
```

Example 3

```
* Parameter assignment using expressions  
.PARAM Pi          = '355/113'  
.PARAM Pi2         = '2*Pi'  
.PARAM npRatio     = 2.1
```

2: Commands in HSPICE Netlists

.PARAM

```
.PARAM nWidth          = 3u
.PARAM pWidth          = 'nWidth * npRatio'
Mpl   ... <pModelName> W = pWidth
Mnl   ... <nModelName> W = nWidth
...
```

Example 4

```
* Algebraic parameter
.param x=cos(2)+sin(2)
```

Example 5

```
* Algebraic expression as an output variable
.PRINT DC v(3) gain=PAR('v(3)/v(2)')
+ PAR('V(4)/V(2)')
```

Example 6

```
* My own user-defined functions
.PARAM <MyFunc( x, y )> = 'Sqrt((x*x)+(y*y))'
.PARAM CentToFar (c)           = '(((c*9)/5)+32)'
.PARAM F(p1,p2)                = 'Log(Cos(p1)*Sin(p2))'
.PARAM SqrDProd (a,b)          = '(a*a)*(b*b)'
```

Example 7

```
* Pre-defined analysis function
.PARAM mcVar = Agauss(1.0,0.1)
```

Example 8

```
.PARAM vtx=OPT1(.7,.3,1.0) uox=OPT1(650,400,900)
```

In this example, `uox` and `vtx` are the variable model parameters, which optimize a model for a selected set of electrical specifications.

The estimated initial value for the `vtx` parameter is 0.7 volts. You can vary this value within the limits of 0.3 and 1.0 volts for the optimization procedure. The optimization parameter reference name (OPT1) references the associated optimization analysis statement (not shown).

Example 9

```
.PARAM fltmod = str('bpfmodel')
s1 n1 n2 n3 n_ref fqmodel=fltmod zo=50 fbase=25e6 fmax=1e9
```

This example shows how you can define and use string parameters.

Description

The `.PARAM` statement defines parameters. Parameters in HSPICE or HSPICE RF are names that have associated numeric values.

A parameter definition in HSPICE or HSPICE RF always uses the last value found in the input netlist (subject to local versus global parameter rules).

Use any of the following methods to define parameters:

- A simple parameter assignment is a constant real number. The parameter keeps this value, unless a later definition changes its value, or an algebraic expression assigns a new value during simulation. HSPICE or HSPICE RF does not warn you if it reassigns a parameter.
- An algebraic parameter (equation) is an algebraic expression of real values, a predefined or user-defined function, or circuit or model values. Enclose a complex expression in single quotes to invoke the algebraic processor, *unless* the expression begins with an alphabetic character and contains no spaces. A simple expression consists of a single parameter name. To use an algebraic expression as an output variable in a `.PRINT`, `.PLOT`, or `.PROBE` statement, use the `PAR` keyword or HSPICE RF (except that you cannot use the `.PLOT` statement in HSPICE RF).
- A user-defined function assignment is similar to an algebraic parameter. HSPICE or HSPICE RF extends the algebraic parameter definition to include function parameters, used in the algebraic that defines the function. You can nest user-defined functions up to three deep.
- A pre-defined analysis function. HSPICE or HSPICE RF provides several specialized analysis types, which require a way to control the analysis:
 - Temperature functions (*fn*)
 - Optimization guess/range

HSPICE also supports the following predefined parameter types, that HSPICE RF does *not* support:

- frequency
- time
- Monte Carlo functions

2: Commands in HSPICE Netlists

.PARAM

Argument	Definition
<i>OPTxxx</i>	Optimization parameter reference name. The associated optimization analysis references this name. Must agree with the OPTxxx name in the analysis command associated with an OPTIMIZE keyname.
<i>parameter</i>	Parameter to vary. <ul style="list-style-type: none">• Initial value estimate• Lower limit.• Upper limit. If the optimizer does not find the best solution within these constraints, it attempts to find the best solution without constraints.
<i>delta</i>	The final parameter value is the initial guess $\pm (n\text{-delta})$. If you do not specify <i>delta</i> , the final parameter value is between <i>low_limit</i> and <i>upper_limit</i> . For example, you can use this parameter to optimize transistor drawn widths and lengths, which must be quantized.

.PAT**Syntax**

```
.PAT <PatName>=data <RB=val> <R=repeat>
```

```
.PAT <patName>=[component 1 ... component n] <RB=val>  
+ <R=repeat>
```

Example 1

The following example shows the `.PAT` command used for a b-string:

```
.PAT a1=b1010 r=1 rb=1
```

Example 2

The following example shows how an existing patname is used to define a new patname:

```
.PAT a1=b1010 r=1 rb=1  
.PAT a2=a1
```

Example 3

This example shows a nested structure:

```
.PAT a1=[b1010 r=1 rb=2 b1100]
```

Example 4

This final example shows how a predefined nested structure is used as a component in a new nested structure:

```
.PAT a1=[b1010 r=1 rb=2 b1100] r=1 rb=1  
.PAT a2=[a1 b0m0m] r=2 rb=1
```

Description

When the `.PAT` command is used in an input file, some patnames are predefined and can be used in a pattern source. Patnames can associate a b-string or nested structure (NS), which are two different types of pattern sources. In this case, a b-string is a series of 1, 0, m, and z states. The NS is a combination of a b-string and another NS defined in the `.PAT` command. The `.PAT` command can also be used to define a new patname, which can be a b-string or NS.

You should avoid using a predefined patname to define another patname, which creates a circular definition. A circular definition is created when a patname is defined that depends on another patname, which in turn is defined

2: Commands in HSPICE Netlists

.PAT

by the original patname. HSPICE detects circular definitions and issues an error report.

Nested structures must use brackets “[]”, but HSPICE does not support using multiple brackets in one statement. If you need to use another nested structure as a component in an NS, define the NS in a new .PAT command.

Argument	Definition
data	String of 1, 0, M, or Z that represents a pattern source. The first letter must be “B,” which represents it as a binary bit stream. This series is called b-string. A 1 represents the high voltage or current value, and a 0 is the low voltage or current value. An m represents the value which is equal to $0.5*(v_{hi}+v_{lo})$, and a z represents the high impedance state (only for voltage source).
PatName	Pattern name that has an associated b-string or nested structure.
component	The elements that make up a nested structure. Components can be b-strings or a patnames defined in other .PAT commands.
RB= <i>val</i>	Specifies the starting component of a repetition. The repeat data starts from the component or bit indicated by RB. RB must be an integer. If RB is larger than the length of the NS or b-string, an error is issued. If it is less than 1, it is automatically set to 1.
R= <i>repeat</i>	Specifies how many times the repeating operation is executed. With no argument, the source repeats from the beginning of the NS or b-string. If R=-1, the repeating operation continues forever. The R must be an integer. If it is less than -1, it automatically set to 0.

.PKG

Syntax

```
.PKG pkgname  
+file= 'pkgfilename'  
+model= 'pkgmodelname'
```

Example 1

```
.pkg p_test  
+ file='processor_clk_ff.ibs'  
+ model='FCPGA_FF_PKG'
```

Example 2

The following example shows how pin1 is referenced:

```
p_test_pin1_dia and p_test_pin1
```

The element name becomes:

```
w_p_test_pin1_?? or r_p_test_pin1_?? ...
```

Description

The `.PKG` command provides the IBIS(V 3.2) Package Model feature. It supports both sections and matrixes.

The `.PKG` command automatically creates a series of elements (*W* or *rlc*). The following nodes are referenced in the netlist:

- Nodes on the die side:
`'pkgname'_'pinname'_dia`
- Nodes on the pin side:
`'pkgname'_'pinname'`

See Example 2 for how pin1 is referenced.

Argument	Definition
pkgname	package card name

2: Commands in HSPICE Netlists

.PKG

Argument	Definition
pkgfilename	name of a .pkg or .ibs file that contains package models.
pkgmodelname	working model in the .pkg file

See Also

[.EBD](#)

[.IBIS](#)

.PLOT

Note: This is an obsolete command. You can gain the same functionality by using the [.PRINT](#) command.

Syntax

```
.PLOT antype ov1 <(plo1,phi1)> <ov2> <(plo2,phi2)> ...>
```

Example 1

```
.PLOT DC V(4) V(5) V(1) PAR(`I1(Q1)/I2(Q1)')  
.PLOT TRAN V(17,5) (2,5) I(VIN) V(17) (1,9)  
.PLOT AC VM(5) VM(31,24) VDB(5) VP(5) INOISE
```

- In the first line, `PAR` plots the ratio of the collector current and the base current for the Q1 transistor.
- In the second line, the `VDB` output variable plots the AC analysis results (in decibels) for node 5.
- In the third line, the AC plot can include `NOISE` results and other variables that you specify.

Example 2

```
.PLOT AC ZIN YOUT(P) S11(DB) S12(M) Z11(R)  
.PLOT DISTO HD2 HD3(R) SIM2  
.PLOT TRAN V(5,3) V(4) (0,5) V(7) (0,10)  
.PLOT DC V(1) V(2) (0,0) V(3) V(4) (0,5)
```

In the last line above, HSPICE sets the plot limits for `V(1)` and `V(2)`, but you specify 0 and 5 volts as the plot limits for `V(3)` and `V(4)`.

Description

The `.PLOT` statement plots the output values of one or more variables in a selected HSPICE analysis. Each `.PLOT` statement defines the contents of one plot, which can contain more than one output variable.

If more than one output variable appears on the same plot, HSPICE prints *and* plots the first variable specified. To print out more than one variable, include another `.PLOT` statement.

You can include wildcards in `.PLOT` statements (HSPICE only).

2: Commands in HSPICE Netlists

.PLOT

Argument	Definition
<i>antype</i>	Type of analysis for the specified plots. Analysis types are: DC, AC, TRAN, NOISE, or DISTO.
<i>ov1 ...</i>	Output variables to plot: voltage, current, or element template variables (HSPICE only; HSPICE RF does not support element template output or .PLOT statements), from a DC, AC, TRAN, NOISE, or DISTO analysis. See the next sections for syntax.
<i>plo1, phi1 ...</i>	Lower and upper plot limits. The plot for each output variable uses the first set of plot limits, after the output variable name. Set a new plot limit for each output variable, after the first plot limit. For example to plot all output variables that use the same scale, specify one set of plot limits at the end of the .PLOT statement. If you set the plot limits to (0,0) HSPICE automatically sets the plot limits.

See Also

[.AC](#)
[.DOUT](#)
[.GRAPH](#)
[.MEASURE](#)
[.PRINT](#)
[.PROBE](#)
[.STIM](#)

.PRINT

Syntax

```
.PRINT antype ov1 <ov2 ... >
```

Example 1

```
* CASE 1
.print v(din) i(mxn18)
.dc vdin 0 5.0 0.05
.tran lns 60ns
* CASE 2
.dc vdin 0 5.0 0.05
.tran lns 60ns
.print v(din) i(mxn18)
* CASE 3
.dc vdin 0 5.0 0.05
.print v(din) i(mxn18)
.tran lns 60ns
```

- If you replace the .PRINT statement with:

```
.print TRAN v(din) i(mnx)
```

then all three cases have identical .sw0 and .tr0 files.

- If you replace the .print statement with:

```
.print DC v(din) i(mnx)
```

then the .sw0 and .tr0 files are different.

Example 2

```
.PRINT TRAN V (4) I(VIN) PAR(`V(OUT)/V(IN)')
```

This example prints the results of a transient analysis for the nodal voltage named 4. It also prints the current through the voltage source named VIN. It also prints the ratio of the nodal voltage at the OUT and IN nodes.

Example 3

```
.PRINT AC VM(4,2) VR(7) VP(8,3) II(R1)
```

- Depending on the value of the ACOUT option, VM(4,2) prints the AC magnitude of the voltage difference, or the difference of the voltage magnitudes, between nodes 4 and 2.
- VR(7) prints the real part of the AC voltage, between node 7 and ground.

2: Commands in HSPICE Netlists

.PRINT

- Depending on the ACOUT value, VP(8,3) prints the phase of the voltage difference between nodes 8 and 3, or the difference of the phase of voltage at node 8 and voltage at node 3.
- I(R1) prints the imaginary part of the current, through R1.

Example 4

```
.PRINT AC ZIN YOUT(P) S11(DB) S12(M) Z11(R)
```

This example prints:

- The magnitude of the input impedance.
- The phase of the output admittance.
- Several S and Z parameters.

This statement accompanies a network analysis by using the .AC and .NET analysis statements.

Example 5

```
.PRINT DC V(2) I(VSRC) V(23,17) I1(R1) I1(M1)
```

This example prints the DC analysis results for several different nodal voltages and currents, through:

- The resistor named R1.
- The voltage source named VSRC.
- The drain-to-source current of the MOSFET named M1.

Example 6

```
.PRINT NOISE INOISE
```

This example prints the equivalent input noise.

Example 7

```
.PRINT DISTO HD3 SIM2(DB)
```

This example prints the magnitude of third-order harmonic distortion, and the decibel value of the intermodulation distortion sum, through the load resistor that you specify in the .DISTO statement (HSPICE only; not supported in HSPICE RF).

Example 8

```
.PRINT AC INOISE ONOISE VM(OUT) HD3
```


This statement includes `NOISE`, `DISTO`, and `AC` output variables in the same `.PRINT` statement in HSPICE. HSPICE RF supports `NOISE` and `AC` analysis, but not `DISTO`.

Example 9

```
.PRINT pj1 = par('p(rd) +p(rs)')
```

This statement prints the value of `pj1` with the specified function.

HSPICE or HSPICE RF ignores `.PRINT` statement references to nonexistent netlist part names, and prints those names in a warning.

Example 10

Derivative function:

```
.PRINT der=deriv('v(NodeX)')
```

Integrate function:

```
.PRINT int = integ('v(NodeX)')
```

The parameter can be a node voltage, or a reasonable expression.

Example 11

```
.print p1 = 3  
.print p2 = par("p1*5")
```

You can use `p1` and `p2` as parameters in netlist. The `p1` value is 3; the `p2` value is 15.

Description

The `.PRINT` statement specifies output variables for which HSPICE or HSPICE RF prints values. You can include wildcards in `.PRINT` statements.

You can also use the `ia11` keyword in a `.PRINT` statement to print all branch currents of all diode, BJT, JFET, or MOSFET elements in your circuit design.

2: Commands in HSPICE Netlists

.PRINT

Argument	Definition
<i>antype</i>	Type of analysis for outputs. Antype is one of the following types: DC, AC, TRAN, NOISE, or DISTO (you cannot run DISTO analysis in HSPICE RF).
<i>ov1 ...</i>	Output variables to print. These are voltage, current, or element template (HSPICE only; HSPICE RF does not support element template output) variables, from a DC, AC, TRAN, NOISE, or DISTO analysis (you cannot run DISTO analysis in HSPICE RF).

See Also

- [.AC](#)
- [.DC](#)
- [.OPTION ACOUT](#)
- [.DISTO](#)
- [.DOUT](#)
- [.GRAPH](#)
- [.MEASURE](#)
- [.NOISE](#)
- [.PLOT](#)
- [.PROBE](#)
- [.STIM](#)
- [.TRAN](#)

.PROBE

Syntax

```
.PROBE antype ov1 <ov2 ...>
```

Example 1

```
.PROBE DC V(4) V(5) V(1) beta = PAR(`I1(Q1)/I2(Q1)`)
```

Example 2

```
* Derivative function  
.PROBE der=deriv('v(NodeX)')  
* Integrate function  
.PROBE int = integ('v(NodeX)')
```

Description

The `.PROBE` statement saves output variables into interface and graph data files. The parameter can be a node voltage, or a reasonable expression. You can include wildcards in `.PROBE` statements.

Argument	Definition
<i>antype</i>	Type of analysis for the specified plots. Analysis types are: DC, AC, TRAN, NOISE, or DISTO (you cannot run DISTO analysis in HSPICE RF).
<i>ov1 ...</i>	Output variables to plot: voltage, current, or element template (HSPICE only; HSPICE RF does not support element template output) variables from a DC, DCMATCH, AC, TRAN, NOISE, or DISTO analysis (you cannot run DISTO analysis in HSPICE RF). <code>.PROBE</code> can include more than one output variable.

2: Commands in HSPICE Netlists

.PROBE

See Also

.AC
.DC
.DCMATCH
.DISTO
.DOUT
.GRAPH
.MEASURE
.NOISE
.PLOT
.PRINT
.STIM
.TRAN

.PROTECT

Syntax

```
.PROTECT
```

Description

The `.PROTECT` statement keeps models and cell libraries private. HSPICE RF does not support the `.PROTECT` statement.

- The `.PROTECT` statement suppresses printing text from the list file, such as when you use the BRIEF option.
- The `.UNPROTECT` command restores normal output functions.
- Any elements and models located between a `.PROTECT` and an `.UNPROTECT` statement, inhibit the element and model listing from the LIST option.
- The `.OPTION NODE` nodal cross reference, and the `.OP` operating point printout, do not list any nodes that are contained within the `.PROTECT` and `.UNPROTECT` statements.

See Also

[.UNPROTECT](#)

.PZ

Syntax

```
.PZ output input
```

```
.PZ ov srcname
```

Example

```
.PZ V(10) VIN  
.PZ I(RL) ISORC
```

- In the first pole/zero analysis, the output is the voltage for node 10, and the input is the `VIN` independent voltage source.
- In the second pole/zero analysis, the output is the branch current for the RL branch, and the input is the `ISORC` independent current source.

Description

The `.PZ` command performs pole/zero analysis (you do not need to specify `.OP`, because the simulator automatically invokes an operating point calculation). See “Pole/Zero Analysis” in the *HSPICE Applications Manual* for complete information about pole/zero analysis.

For a description of pole/zero options, see [Chapter 3, Options in HSPICE Netlists](#).

.

Argument	Definition
input	Input source. Can be the name of any independent voltage or current source.
output	Output variables, which can be: <ul style="list-style-type: none">• Any node voltage, <code>V(n)</code>.• Any branch current, <code>I(branch_name)</code>.

Argument	Definition
ov	Output variable: <ul style="list-style-type: none">• a node voltage $V(n)$, or• a branch current $I(element)$
srcnam	Input source: <ul style="list-style-type: none">• an independent voltage or• a current source name

See Also

[.DC](#)

.SAMPLE

Syntax

```
.SAMPLE FS = freq <TOL = val> <NUMF = val>
+ <MAXFLD = val> <BETA = val>
```

Description

To acquire data from analog signals, use the `.SAMPLE` statement with the `.NOISE` and `.AC` statements to analyze data sampling noise in HSPICE or HSPICE RF. The `SAMPLE` analysis performs a noise-folding analysis, at the output node.

.

Argument	Definition
<i>FS = freq</i>	Sample frequency in hertz.
<i>TOL</i>	Sampling-error tolerance: the ratio of the noise power (in the highest folding interval) to the noise power (in baseband). Default = 1.0e-3.
<i>NUMF</i>	Maximum number of frequencies that you can specify. The algorithm requires about ten times this number of internally-generated frequencies so keep this value small. Default = 100.
<i>MAXFLD</i>	Maximum number of folding intervals (default = 10.0). The highest frequency (in hertz) that you can specify is: $F_{MAX} = MAXFLD \cdot FS$
<i>BETA</i>	Optional noise integrator (duty cycle), at the sampling node: <ul style="list-style-type: none"> • $BETA = 0$ no integrator • $BETA = 1$ simple integrator (default) If you clock the integrator (integrates during a fraction of the $1/FS$ sampling interval), then set <code>BETA</code> to the duty cycle of the integrator.

See Also

[.AC](#)
[.NOISE](#)

.SAVE

Syntax

```
.SAVE <TYPE = type_keyword> <FILE = save_file>
+ <LEVEL = level_keyword> <TIME = save_time>
```

Example

```
.TEMP -25 0 25
.SAVE TYPE=NODESET FILE=my_design.ic0 LEVEL=ALL
+ TIME=0
```

This example saves the operating point corresponding to `.TEMP -25` to a file named `my_design.ic0`.

Description

The `.SAVE` statement in HSPICE stores the operating point of a circuit in a file that you specify. HSPICE RF does not support the `.SAVE` statement. For quick DC convergence in subsequent simulations, use the `.LOAD` statement to input the contents of this file. HSPICE saves the operating point by default, even if the HSPICE input file does not contain a `.SAVE` statement. To not save the operating point, specify `.SAVE LEVEL = NONE`.

You can save the operating point data as either an `.IC` or a `.NODESET` statement.

A parameter or temperature sweep saves only the first operating point.

.

Argument	Definition
type_keyword	Storage method for saving the operating point. The type can be one of the following. Default is NODESET. <ul style="list-style-type: none"> <code>.NODESET</code>: Stores the operating point as a <code>.NODESET</code> statement. Later simulations initialize all node voltages to these values, if you use the <code>.LOAD</code> statement. If circuit conditions change incrementally, DC converges within a few iterations. <code>.IC</code>: Stores the operating point as a <code>.IC</code> statement. Later simulations initialize node voltages to these values if the netlist includes the <code>.LOAD</code> statements.
save_file	Name of the file that stores DC operating point data. The file name format is <code><design>.ic#</code> . Default is <code><design>.ic0</code> .

2: Commands in HSPICE Netlists

.SAVE

Argument	Definition
level_keyword	Circuit level, at which you save the operating point. The level can be one of the following. <ul style="list-style-type: none">• ALL (default): Saves all nodes, from the top to the lowest circuit level. This option offers the greatest improvement in simulation time.• TOP: Saves only nodes in the top-level design. Does not save subcircuit nodes.• NONE: Does not save the operating point.
save_time	Time during transient analysis when HSPICE saves the operating point. HSPICE requires a valid transient analysis statement to save a DC operating point. Default = 0.

See Also

[.IC](#)

[.LOAD](#)

[.NODESET](#)

.SENS

Syntax

```
.SENS ov1 <ov2 ...>
```

Example

```
.SENS V(9) V(4,3) V(17) I(VCC)
```

Description

The `.SENS` command obtains DC small-signal sensitivities of output variables for circuit parameters. You can use this command HSPICE, but not in HSPICE RF.

If the input file includes a `.SENS` statement, HSPICE determines DC small-signal sensitivities for each specified output variable, relative to every circuit parameter. The sensitivity measurement is the partial derivative of each output variable for a specified circuit element, measured at the operating point, and normalized to the total change in output magnitude. Therefore, the sum of the sensitivities of all elements is 100%. DC small-signal sensitivities are calculated for:

- resistors
- voltage sources
- current sources
- diodes
- BJTs (including Level 4, the VBIC95 model)
- MOSFETs (Level49 and Level53, Version=3.22).

You can perform only one `.SENS` analysis per simulation. Only the last `.SENS` statement is used in case more than one is present. The others are discarded with warnings.

The amount of output generated from a `.SENS` analysis is dependent on the size of the circuit.

.

Argument	Definition
ov1 ov2 ...	Branch currents, or nodal voltage for DC component-sensitivity analysis

2: Commands in HSPICE Netlists
.SENS

See Also

[.DC](#)

.SHAPE

Syntax

```
.SHAPE sname Shape_Descriptor
```

Description

Use the `.SHAPE` statement to define a shape. The Field Solver uses the shape to describe a cross-section of the conductor.

Argument	Definition
sname	Shape name.
Shape_Descriptor	One of the following: <ul style="list-style-type: none">• Rectangle• Circle• Strip• Polygon

See Also

[.FSOPTIONS](#)
[.LAYERSTACK](#)
[.MATERIAL](#)

2: Commands in HSPICE Netlists

.SHAPE (Defining Rectangles)

.SHAPE (Defining Rectangles)

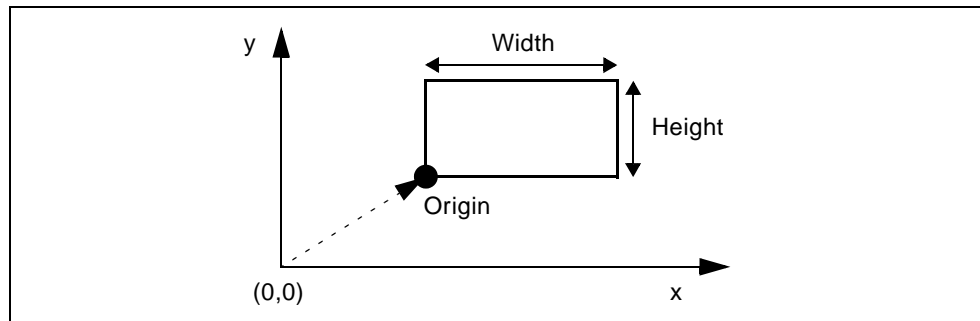
Syntax

```
.SHAPE RECTANGLE WIDTH=val HEIGHT=val <NW=val>  
+ <NH=val>
```

Description

Use the `RECTANGLE` option to define a rectangle. Normally, you do not need to specify the `NW` and `NH` values because the field solver automatically sets these values, depending on the *accuracy* mode. You can specify both values, or specify only one of these values and let the solver determine the other.

Figure 3 Coordinates of a Rectangle



Argument	Definition
WIDTH	Width of the rectangle (size in the x-direction).
HEIGHT	Height of the rectangle (size in the y-direction).
NW	Number of horizontal (x) segments in a rectangle with a specified width.
NH	Number of vertical (y) segments in a rectangle with a specified height.

.SHAPE (Defining Circles)

Syntax

```
.SHAPE CIRCLE RADIUS=val <N=val>
```

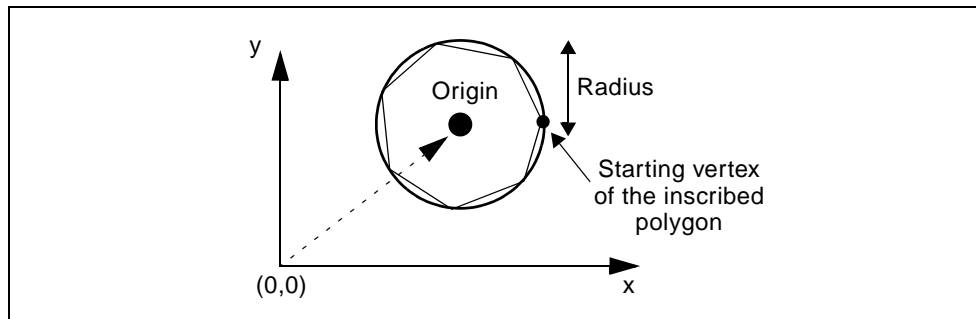
Description

The `CIRCLE` option to defines a circle in the Field Solver. The Field Solver approximates a circle as an inscribed regular polygon with N edges. The more edges, the more accurate the circle approximation is.

Do not use the `CIRCLE` descriptor to model actual polygons; instead use the `POLYGON` descriptor.

Normally, you do not need to specify the N value, because the field solver automatically sets this value, depending on the *accuracy* mode. But you can specify this value if you need to

Figure 4 Coordinates of a Circle



Argument	Definition
RADIUS	Radius of the circle.
N	Number of segments to approximate a circle with a specified radius.

2: Commands in HSPICE Netlists

.SHAPE (Defining Polygons)

.SHAPE (Defining Polygons)

Syntax

```
.SHAPE POLYGON VERTEX=(x1 y1 x2 y2 ...)  
+ <N=(n1,n2,...)>
```

Example 1

The following rectangular polygon uses the default number of segments:

```
.SHAPE POLYGON VERTEX=(1 10 1 11 5 11 5 10)
```

Example 2

The following rectangular polygon uses five segments for each edge:

```
.SHAPE POLYGON VERTEX=(1 10 1 11 5 11 5 10)  
+ N=5
```

Example 3

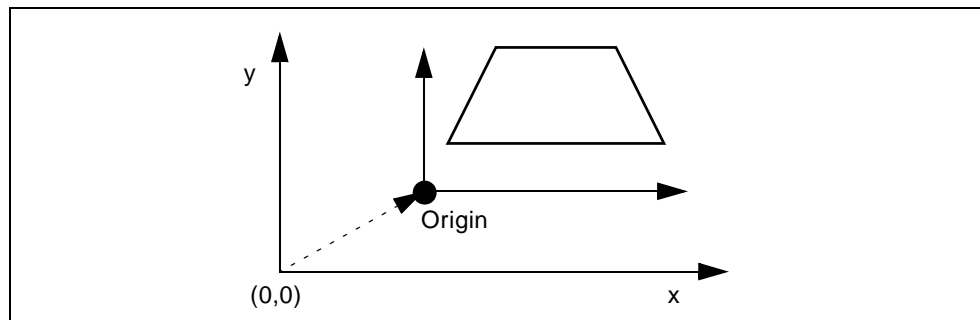
Rectangular polygon by using the different number of segments for each edge:

```
.SHAPE POLYGON VERTEX=(1 10 1 11 5 11 5 10)  
+ N=(5 3 5 3)
```

Description

The `.SHAPE POLYGON` command option defines a polygon in a Field Solver. The specified coordinates are within the local coordinate with respect to the origin of a conductor.

Figure 5 Coordinates of a Polygon



2: Commands in HSPICE Netlists

.SHAPE (Defining Polygons)

Argument	Definition
VERTEX	(x, y) coordinates of vertices. Listed either in clockwise or counter-clockwise direction.
N	<p>Number of segments that define the polygon with the specified X and Y coordinates. You can specify a different N value for each edge. If you specify only one N value, then the Field Solver uses this value for <i>all</i> edges.</p> <p>For example, the first value of N, $n1$, corresponds to the number of segments for the edge from ($x1\ y1$) to ($x2\ y2$).</p>

2: Commands in HSPICE Netlists

.SHAPE (Defining Strip Polygons)

.SHAPE (Defining Strip Polygons)

Syntax

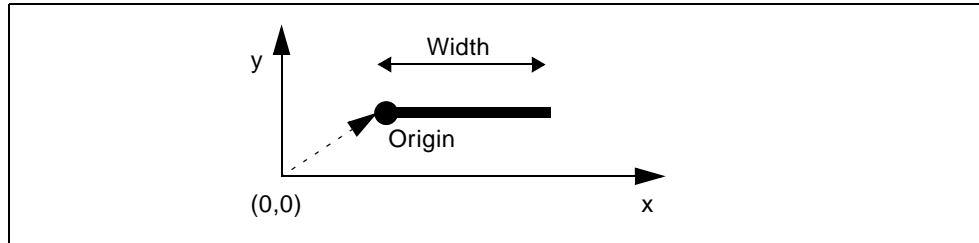
```
.SHAPE STRIP WIDTH=val <N=val>
```

Description

Normally, you do not need to specify the N value, because the field solver automatically sets this value, depending on the *accuracy* mode. But you can specify this value if you need to.

The field solver (filament method) does not support this shape.

Figure 6 Coordinates of a Strip Polygon



Argument	Definition
WIDTH	Width of the strip (size in the x-direction).
N	Number of segments that define the strip shape with the specified width.

.STIM

Syntax

General

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

PWL Source (Transient Analysis Only)

```
.STIM [tran] PWL [filename=output_filename]
+ [name1=] ovar1 [node1=n+] [node2=n-]
+ [[name2=]ovar2 [node1=n+] [node2=n-] ...]
+ [from=val] [to=val] [npoints=val]
```

```
.STIM [tran] PWL [filename=output_filename]
+ [name1=] ovar1 [node1=n+] [node2=n-]
+ [[name2=]ovar2 [node1=n+] [node2=n-] ...]
+ indepvar=[(]t1 [t2 ...[)]]
```

Data Card

```
.STIM [tran | ac | dc] DATA [filename=output_filename]
+ dataname [name1=] ovar1
+ [[name2=]ovar2 ...] [from= val] [to=val]
+ [npoints=val] [indepout=val]
```

```
.STIM [tran | ac | dc] DATA [filename=output_filename]
+ dataname [name1=] ovar1
+ [[name2=]ovar2 ...] indepvar=[(]t1 [t2 ...[)]]
+ [indepout=val]
```

Digital Vector File (Transient Analysis Only)

```
.STIM [tran] VEC [filename=output_filename]
+ vth=val vtl=val [voh=val] [vol=val]
+ [name1=] ovar1 [[name2=] ovar2 ...]
+ [from=val] [to=val] [npoints=val]
```

```
.STIM [tran] VEC [filename=output_filename]
+ vth=val vtl=val [voh=val] [vol=val]
+ [name1=] ovar1 [[name2=] ovar2 ...]
+ indepvar=[(]t1 [t2 ...[)]]
```

2: Commands in HSPICE Netlists

.STIM

Description

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.

PWL Source (Transient Analysis Only):

Argument	Definition
<i>tran</i>	Transient simulation.
<i>filename</i>	Output file name. If you do not specify a file, HSPICE uses the input filename.
<i>namei</i>	PWL Source Name that you specify. The name must start with V (for a voltage source) or I (for a current source).
<i>ovar1</i>	Output variable that you specify. <i>ovar</i> can be: <ul style="list-style-type: none">• Node voltage.• Element current.• Parameter string. If you use a parameter string, you must specify <i>name1</i>. You cannot use character strings as parameter values in HSPICE RF. For example: <code>v(1), i(r1), v(2,1), par('v(1)+v(2)')</code>
<i>node1</i>	Positive terminal node name.
<i>node2</i>	Negative terminal node name.
<i>from</i>	Specifies the time to start output of simulation results. For transient analysis, uses the time units that you specified.
<i>npoints</i>	Number of output time points.

Argument	Definition
<i>to</i>	Specifies the time to end output of simulation results. For transient analysis, uses the time units that you specified. The <i>from</i> value can be greater than the <i>to</i> value.
<i>indepvar</i>	Specifies dispersed (independent-variable) time points. You must specify dispersed time points in <i>increasing</i> order.

Data Card:

Argument	Definition
tran ac dc	Selects the simulation type: transient, AC, or DC.
filename	Output file name. If you do not specify a file, HSPICE uses the input filename.
<i>dataname</i>	Name of the data card to generate.
<i>from</i>	Specifies the time to start output of simulation results. For transient analysis, uses the time units that you specified.
<i>to</i>	Specifies the time to end output of simulation results. For transient analysis, uses the time units that you specified.
<i>namei</i>	Name of a parameter of the data card to generate.
<i>npoints</i>	Number of output independent-variable points.
<i>indepvar</i>	Specifies dispersed independent-variable points.
<i>indepout</i>	Indicates whether to generate the independent variable column. <ul style="list-style-type: none"> • indepout, indepout = 1, or on, produces the independent variable column. You can specify the independent-variables in any order. • indepout= 0 or off (default) does not create an independent variable column. You can place the indepout field anywhere after the ovar1 field.

2: Commands in HSPICE Netlists

.STIM

Argument	Definition
<i>ovar</i>	Output variable that you specify. <i>ovar</i> can be: <ul style="list-style-type: none">• Node voltage.• Element current.• Element templates (HSPICE only).• Parameter string. You cannot use character strings as parameter values in HSPICE RF. For example: <code>v(1), i(r1), v(2,1), par('v(1)+v(2)'), LX1(m1), LX2(m1)</code>

Digital Vector File (Transient Analysis Only):

Argument	Definition
<i>namei</i>	Signal name that you specify.
<i>filename</i>	Output file name. If you do not specify a file, HSPICE uses the input filename.
<i>ovar</i>	Output variable that you specify. <i>ovar</i> can only be a node voltage.
<i>from</i>	Specifies the time to start output of simulation results. For transient analysis, uses the time units that you specified.
<i>to</i>	Time to the end output of simulation results. For transient analysis, uses the specified time units. The <i>from</i> value can be greater than the <i>to</i> value.
<i>npoints</i>	Number of output time points.
<i>indepvar</i>	Specifies dispersed independent-variable points. You must specify dispersed time points in <i>increasing</i> order.
<i>vth</i>	High voltage threshold.
<i>vtl</i>	Low voltage threshold.
<i>voh</i>	Logic-high voltage for each output signal.
<i>vol</i>	Logic-low voltage for each output signal.

See Also

.DOUT
.GRAPH
.MEASURE
.PLOT
.PRINT
.PROBE

2: Commands in HSPICE Netlists

.SUBCKT

.SUBCKT

In HSPICE RF, you cannot replicate output commands within subcircuit (subckt) definitions.

Syntax

```
.SUBCKT subnam n1 <n2 n3 ...> <parnam = val>
.ENDS

.SUBCKT <SubName><PinList>[<SubDefaultsList>]
.ENDS

.SUBCKT subnam n1 <n2 n3 ...> <param=str('string')>
.ENDS
```

Example 1

```
*FILE SUB2.SP TEST OF SUBCIRCUITS
.OPTION LIST ACCT
  V1 1 0 1
.PARAM P5 = 5 P2 = 10
.SUBCKT SUB1 1 2 P4 = 4
  R1 1 0 P4
  R2 2 0 P5
  X1 1 2 SUB2 P6 = 7
  X2 1 2 SUB2
.ENDS
*
.MACRO SUB2 1 2 P6 = 11
  R1 1 2 P6
  R2 2 0 P2
.EOM
  X1 1 2 SUB1 P4 = 6
  X2 3 4 SUB1 P6 = 15
  X3 3 4 SUB2
*
.MODEL DA D CJA = CAJA CJP = CAJP VRB = -20
  IS = 7.62E-18
+ PHI = .5 EXA = .5 EXP = .33
.PARAM CAJA = 2.535E-16 CAJP = 2.53E-16
.END
```

The preceding example defines two subcircuits: SUB1 and SUB2. These are resistor-divider networks, whose resistance values are parameters (variables). The X1, X2, and X3 statements call these subcircuits. Because the resistor values are different in each call, these three calls produce different subcircuits.

Example 2

```
.SUBCKT Inv a y Strength = 3
    Mp1 <MosPinList> pMosMod L = 1.2u
    W = 'Strength * 2u'
    Mn1 <MosPinList> nMosMod L = 1.2u
    W = 'Strength * 1u'
.ENDS
...
xInv0 a y0 Inv $ Default devices: p device = 6u,
    $ n device = 3u
xInv1 a y1 Inv Strength = 5 $ p device = 10u,
    n device = 5u
xInv2 a y2 Inv Strength = 1 $ p device = 2u,
    n device = 1u
...
```

This example implements an inverter that uses a *Strength* parameter. By default, the inverter can drive three devices. Enter a new value for the *Strength* parameter in the element line to select larger or smaller inverters for the application.

Example 3

```
* Using string parameters
.subckt IBIS vccq vss out in
+ IBIS_FILE=str('file.ibs')
+ IBIS_MODEL=str('ibis_model')
ven en 0 vcc
B1 vccq vss out in en v0dq0 vccq vss
+ file= str(IBIS_FILE) model=str(IBIS_MODEL)
.ends
```

This example implements an IBIS model that uses string parameters to specify the IBIS file name and IBIS model name.

Description

You can create a subcircuit description for a commonly-used circuit, and include one or more references to the subcircuit in your netlist.

To define a subcircuit in your netlist, use the `.SUBCKT` statement.

When you use hierarchical subcircuits, you can pick default values for circuit elements in a `.SUBCKT` command. You can use this feature in cell definitions to simulate the circuit with typical values.

2: Commands in HSPICE Netlists

.SUBCKT

Use the `.ENDS` statement to terminate a `.SUBCKT` statement.

Argument	Definition
<i>subnam</i>	Specifies a reference name for the subcircuit model call.
<i>n1 ...</i>	Node numbers for external reference; cannot be the ground node (zero). Any element nodes that are in the subcircuit, but are not in this list, are strictly local with three exceptions: <ul style="list-style-type: none">• Ground node (zero).• Nodes assigned using <code>BULK = node</code> in MOSFET or BJT models.• Nodes assigned using the <code>.GLOBAL</code> statement.
<i>parnam</i>	A parameter name set to a value. Use only in the subcircuit. To override this value, assign it in the subcircuit call, or set a value in a <code>.PARAM</code> statement.
<i>SubDefaultsList</i>	<code><SubParam1>=<Expression></code> <code>[<SubParam2>=<Expression>...]</code>

See Also

[.ENDS](#)
[.EOM](#)
[.MACRO](#)
[.MODEL.MODEL](#)
[.OPTION LIST](#)
[.PARAM](#)

.TEMP

Syntax

```
.TEMP t1 <t2 <t3 ...>>
```

Example 1

```
.TEMP -55.0 25.0 125.0
```

The `.TEMP` statement sets the circuit temperatures for the entire circuit simulation. To simulate the circuit by using individual elements or model temperatures, HSPICE or HSPICE RF uses:

- Temperature as set in the `.TEMP` statement.
- `.OPTION TNOM` setting (or the `TREF` model parameter).
- `DTEMP` element temperature.

Example 2

```
.TEMP 100  
D1 N1 N2 DMOD DTEMP=30  
D2 NA NC DMOD  
R1 NP NN 100 TC1=1 DTEMP=-30  
.MODEL DMOD D IS=1E-15 VJ=0.6 CJA=1.2E-13  
+ CJP=1.3E-14 TREF=60.0
```

In this example:

- The `.TEMP` statement sets the circuit simulation temperature to 100°C.
- You do not specify `.OPTION TNOM` so it defaults to 25°C.
- The temperature of the diode is 30°C above the circuit temperature as set in the `DTEMP` parameter.

That is:

- $D1temp = 100^{\circ}C + 30^{\circ}C = 130^{\circ}C$.
- HSPICE or HSPICE RF simulates the D2 diode at 100°C.
- R1 simulates at 70°C.

2: Commands in HSPICE Netlists

.TEMP

Because the diode model statement specifies `TREF` at 60°C, HSPICE or HSPICE RF derates the specified model parameters by:

- 70°C (130°C - 60°C) for the D1 diode.
- 40°C (100°C - 60°C) for the D2 diode.
- 45°C (70°C - `TNOM`) for the R1 resistor.

Description

To specify the circuit temperature for an HSPICE or HSPICE RF simulation, use the `.TEMP` statement, or the `TEMP` parameter in the `.DC`, `.AC`, and `.TRAN` statements. HSPICE compares the circuit simulation temperature against the reference temperature in the `.OPTION TNOM` control. HSPICE or HSPICE RF uses the difference between the circuit simulation temperature and the `TNOM` reference temperature to define derating factors for component values.

In HSPICE RF, you can use multiple `.TEMP` statements to specify multiple temperatures for different portions of the circuit. HSPICE permits only one temperature for the entire circuit. Multiple definitions of the `.TEMP` statements in a circuit behave as a sweep function.

Note: HSPICE allows multiple `.TEMP` statements in the netlist, and performs multiple DC, AC or TRAN analyses for each temperature. Do not set the temperature to the same value multiple times.

Argument	Definition
<i>t1 t2</i>	Temperatures in xC, at which HSPICE or HSPICE RF simulates the circuit.

See Also

[.AC](#)
[.DC](#)
[.TEMP](#)
[.OPTION TNOM](#)
[.TRAN](#)

.TF

Syntax

```
.TF ov srcnam
```

Example

```
.TF V(5,3) VIN  
.TF I(VLOAD) VIN
```

For the first example, HSPICE or HSPICE RF computes the ratio of V(5,3) to VIN. This is the ratio of small-signal input resistance at VIN to the small-signal output resistance (measured across nodes 5 and 3). If you specify more than one .TF statement in a single simulation, HSPICE or HSPICE RF runs only the last .TF statement.

Description

The transfer function statement (.TF) calculates DC small-signal values for transfer functions (ratio of output variable to input source). You do not need to specify .OP.

The .TF statement defines small-signal output and input for DC small-signal analysis. When you use the .TF statement, HSPICE or HSPICE RF computes:

- DC small-signal value of the transfer function (output/input),.
- Input resistance.
- Output resistance.

.

Argument	Definition
<i>ov</i>	Small-signal output variable.
<i>srcnam</i>	Small-signal input source.

See Also

[.DC](#)

2: Commands in HSPICE Netlists

.TITLE

.TITLE

Syntax

```
.TITLE <string_of_up_to_72_characters>
```

or

```
<string_of_up_to_72_characters>
```

Example

```
.TITLE my-design_netlist
```

Description

You set the simulation title in the first line of the input file. HSPICE or HSPICE RF always reads this line, and uses it as the title of the simulation, regardless of the line's contents. The simulation prints the title verbatim in each section heading of the output listing file.

To set the title, you can place a `.TITLE` statement on the first line of the netlist. However, HSPICE or HSPICE RF does not *require* the `.TITLE` syntax.

In the second form of the syntax, the string is the first line of the input file. The first line of the input file is always the implicit title. If any statement appears as the first line in a file, simulation interprets it as a title, and does not execute it.

An `.ALTER` statement does not support using the `.TITLE` statement. To change a title for a `.ALTER` statement, place the title content in the `.ALTER` statement itself.

Argument	Definition
<i>string</i>	Any character string up to 72 characters long.

.TRAN**Syntax****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>
```

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

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

In HSPICE RF, you can run a parameter sweep around a single analysis, but the parameter sweep cannot change any `.OPTION` value.

Data-Driven Sweep

```
.TRAN DATA = datanm (HSPICE only; HSPICE RF does not support  
the .TRAN DATA statement)
```

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

```
.TRAN DATA = datanm<SWEEP var type np pstart pstop>(HSPICE  
only; HSPICE RF does not support the .TRAN DATA statement)
```

```
.TRAN DATA=datanm <SWEEP var START="param_expr1"  
+STOP="param_expr2" STEP="param_expr3">
```

```
.TRAN DATA=datanm  
+ <SWEEP var start_expr stop_expr step_expr>
```

2: Commands in HSPICE Netlists

.TRAN

HSPICE RF supports the data-driven syntax only for parameter sweeps:

```
.tran AB sweepdata=name
```

Monte Carlo Analysis

```
.TRAN tincr1 tstop1 <tincr2 tstop2 ...tincrN tstopN>  
+ <START = val> <UIC> <SWEEP MONTE = list(<>  
+ <num1:num2> <num3> <num5:num6> <num7> <>> >
```

Optimization

```
.TRAN DATA = datanm OPTIMIZE = opt_par_fun  
+ RESULTS = measnames MODEL = optmod
```

```
.TRAN <DATA=filename> SWEEP OPTIMIZE=OPTxxx  
+ RESULTS=ierr1 ... ierrn MODEL=optmod
```

Example 1

```
.TRAN 1NS 100NS
```

This example performs and prints the transient analysis, every 1 ns for 100 ns.

Example 2

```
.TRAN .1NS 25NS 1NS 40NS START = 10NS
```

This example performs the calculation every 0.1 ns for the first 25 ns; and then every 1 ns, until 40 ns. Printing and plotting begin at 10 ns.

Example 3

```
.TRAN 10NS 1US UIC
```

This example performs the calculation every 10 ns for 1 μ s. This example bypasses the initial DC operating point calculation. It uses the nodal voltages, specified in the .IC statement (or by IC parameters in element statements) to calculate the initial conditions.

Example 4

```
.TRAN 10NS 1US UIC SWEEP TEMP -55 75 10
```

This example increases the temperature by 10 °C, through the range -55 °C to 75 °C. It also performs transient analysis for each temperature.

Example 5

```
.TRAN 10NS 1US SWEEP load POI 3 1pf 5pf 10pf
```


This example analyzes each load parameter value, at 1 pF, 5 pF, and 10 pF.

Example 6

```
.TRAN data = dataname
```

This example is a data-driven time sweep. It uses a data file as the sweep input. If the parameters in the data statement are controlling sources, then a piecewise linear specification must reference them.

Example 7

```
.TRAN 10NS 1US SWEEP MONTE = 10 firstrun = 11
```

This example performs the calculation every 10ns for 1us, from the 11th to 20th Monte Carlo trials.

Example 8

```
.TRAN 10NS 1US SWEEP MONTE = list(10 20:30 35:40 50)
```

This example performs the calculation every 10ns for 1us, at the 10th trial, then from the 20th to the 30th trial, followed by the 35th to the 40th trial, and finally at the 50th Monte Carlo trial.

Description

.TRAN starts a transient analysis, which simulates a circuit at a specific time.

Argument	Definition
<i>DATA = datanm</i>	Data name, referenced in the .TRAN statement (HSPICE only; not supported in HSPICE RF).
<i>MONTE = val</i>	Produces a specified number (<i>val</i>) of randomly-generated values. HSPICE uses them to select parameters from a <i>Gaussian</i> , <i>Uniform</i> , or <i>Random Limit</i> distribution (HSPICE only; not supported in HSPICE RF).
<i>np</i>	Number of points, or number of points per decade or octave, depending on what keyword precedes it.
<i>param_expr...</i>	Expressions you specify: <i>param_expr1...param_exprN</i> .

2: Commands in HSPICE Netlists

.TRAN

Argument	Definition
<i>pincr</i>	Voltage, current, element, or model parameter; or any temperature increment value. If you set the <i>type</i> variation, use <i>np</i> (number of points), not <i>pincr</i> .
<i>pstart</i>	Starting voltage, current, or temperature; or any element or model parameter value. If you set the <i>type</i> variation to POI (list of points), use a list of parameter values, instead of <i>pstart pstop</i> .
<i>pstop</i>	Final voltage, current, or temperature; or element or model parameter value.
<i>START</i>	Time when printing or plotting begins. The <i>START</i> keyword is optional: you can specify a start time without the keyword. If you use <i>.TRAN</i> with <i>.MEASURE</i> , a non-zero <i>START</i> time can cause incorrect <i>.MEASURE</i> results. Do not use non-zero <i>START</i> times in <i>.TRAN</i> statements when you also use <i>.MEASURE</i> .
<i>SWEEP</i>	Indicates that <i>.TRAN</i> specifies a second sweep.
<i>tincr1...</i>	Specifies the printing or plotting increment for printer output, and the suggested computing increment for post-processing.
<i>tstop1...</i>	Time when a transient analysis stops incrementing by the first specified time increment (<i>tincr1</i>). If another <i>tincr-tstop</i> pair follows, analysis continues with a new increment.
<i>UIC</i>	Uses the nodal voltages specified in the <i>.IC</i> statement (or in the <i>IC =</i> parameters of the various element statements) to calculate initial transient conditions, rather than solving for the quiescent operating point.
<i>type</i>	Specifies any of the following keywords: <ul style="list-style-type: none">• DEC – decade variation.• OCT – octave variation (the value of the designated variable is eight times its previous value).• LIN – linear variation.• POI – list of points.

Argument	Definition
<i>var</i>	Name of an independent voltage or current source, any element or model parameter, or the TEMP keyword (indicating a temperature sweep). You can use a source value sweep, referring to the source name (SPICE style). However, if you specify a parameter sweep, a .DATA statement, and a temperature sweep, you must choose a parameter name for the source value, and subsequently refer to it in the .TRAN statement. The parameter name must not start with V or I.
<i>firstrun</i>	The <i>val</i> value specifies the number of Monte Carlo iterations to perform. The <i>firstrun</i> value specifies the desired number of iterations. HSPICE runs from num1 to num1+val-1.
<i>list</i>	The iterations at which HSPICE performs a Monte Carlo analysis. You can write more than one number after <i>list</i> . The colon represents "from ... to ...". Specifying only one number makes HSPICE run at only the specified point.

.UNPROTECT

Syntax

```
.UNPROTECT
```

Description

In HSPICE, the `.UNPROTECT` statement restores normal output functions that a `.PROTECT` statement restricted. HSPICE RF does not support the `.UNPROTECT` statement.

- Any elements and models located between `.PROTECT` and `.UNPROTECT` statements, inhibit the element and model listing from the LIST option.
- Neither the `.OPTION NODE` cross reference, nor the `.OP` operating point printout, list any nodes within the `.PROTECT` and `.UNPROTECT` statements.

See Also

[.PROTECT](#)

.VEC

Syntax

```
.VEC `digital_vector_file`
```

Description

You can call a digital vector file from an HSPICE netlist. A digital vector file consists of three parts:

- Vector Pattern Definition section
- Waveform Characteristics section
- Tabular Data section.

The .VEC file must be a text file. If you transfer it between Unix and Windows, use *text* mode.

2: Commands in HSPICE Netlists

.WIDTH

.WIDTH

Syntax

```
.WIDTH OUT = {80 |132}
```

Example

```
.WIDTH OUT = 132 $ SPICE compatible style  
.OPTION CO = 132 $ preferred style
```

Description

Use the `.WIDTH` statement to define the print-out width in HSPICE.

Permissible values for `OUT` are 80 and 132. You can also use `.OPTION CO` to set the `OUT` value.

Argument	Definition
<i>OUT</i>	Output print width.

3

Options in HSPICE Netlists

Describes the simulation options you can set using various forms of the `.OPTION` command.

You can set a wide variety of HSPICE simulation options using the `.OPTION` command. This chapter provides a list of the various options, arranged by task, followed by detailed descriptions of the individual options.

Options in this chapter fall into the following categories:

- [General Control Options](#)
- [CPU Options](#)
- [Interface Options](#)
- [Analysis Options](#)
- [Error Options](#)
- [Version Option](#)
- [Model Analysis Options](#)
- [DC Operating Point, DC Sweep, and Pole/Zero Options](#)
- [Transient and AC Small Signal Analysis Options](#)
- [Transient Control Options](#)
- [Input/Output Options](#)

3: Options in HSPICE Netlists

General Control Options

- AC Control Options
- Common Model Interface Options
- Verilog-A Options

General Control Options

.OPTION ACCT	.OPTION INGOLD	.OPTION NXX
.OPTION ACOUT	.OPTION LENNAM	.OPTION OPTLST
.OPTION ALT999 or ALT9999	.OPTION LIST	.OPTION OPTS
OPTION ALTCC	.OPTION MEASDGT	.OPTION PATHNUM
.OPTION ALTCHK	.OPTION NODE	.OPTION PLIM
.OPTION BEEP	.OPTION NOELCK	.OPTION POST_VERSION
.OPTION BINPRINT	.OPTION NOMOD	.OPTION SEARCH
.OPTION BRIEF	.OPTION NOPAGE	.OPTION STATFL
.OPTION CO	.OPTION NOTOP	.OPTION VERIFY

CPU Options

.OPTION CPTIME	.OPTION EPSMIN	.OPTION EXPMAX
.OPTION LIMTIM		

Interface Options

.OPTION ARTIST	.OPTION MENTOR	.OPTION PROBE
.OPTION CDS	.OPTION MONTECON	.OPTION PSF
.OPTION CSDF	.OPTION POST	.OPTION SDA
.OPTION DLENCDF	.OPTION POSTLVL	.OPTION ZUKEN
.OPTION MEASOUT	.OPTION POSTTOP	

Analysis Options

.OPTION ASPEC	.OPTION NOISEMINFREQ	.OPTION SEED
.OPTION FFTOUT	.OPTION PARHIER	.OPTION SPICE
.OPTION LIMPTS		

Error Options

.OPTION BADCHR	.OPTION DIAGNOSTIC	.OPTION NOWARN
.OPTION WARNLIMIT		

Version Option

.OPTION H9007

Model Analysis Options

General Model Analysis Options

.OPTION DCAP	.OPTION HIER_SCALE	.OPTION TNOM
.OPTION MODSRH	.OPTION MODMONTE	.OPTION XDTEMP
.OPTION SCALE		

MOSFET Model Analysis Options

.OPTION CVTOL	.OPTION DEFNRD	.OPTION DEFW
.OPTION DEFAD	.OPTION DEFNRS	.OPTION SCALM
.OPTION DEFAS	.OPTION DEFPPD	.OPTION WL
.OPTION DEFL	.OPTION DEFPS	.OPTION WNFLAG

3: Options in HSPICE Netlists

DC Operating Point, DC Sweep, and Pole/Zero Options

Inductor Model Analysis Options

`.OPTION GENK`

`.OPTION KLIM`

BJT and Diode Model Analysis Options

`.OPTION EXPLI`

DC Operating Point, DC Sweep, and Pole/Zero Options

DC Accuracy Options

`.OPTION ABSH`

`.OPTION DI`

`.OPTION RELMOS`

`.OPTION ABSI`

`.OPTION KCLTEST`

`.OPTION RELV`

`.OPTION ABSMOS`

`.OPTION MAXAMP`

`.OPTION RELVDC`

`.OPTION ABSTOL`

`.OPTION RELH`

`.OPTION ABSVDC`

`.OPTION RELI`

DC Matrix Options

`.OPTION ITL1`

`.OPTION PIVOT`

`.OPTION PIVTOL`

`.OPTION ITL2`

`.OPTION PIVREF`

`.OPTION SPARSE`

`.OPTION NOPIV`

`.OPTION PIVREL`

DC Pole/Zero I/O Options

`.OPTION CAPTAB`

`.OPTION DCCAP`

`.OPTION OPFILE`

`.OPTION VFLOOR`

DC Convergence Options

.OPTION CONVERGE	.OPTION DV	.OPTION ITLPTRAN
.OPTION CSHDC	.OPTION GMAX	.OPTION NEWTOL
.OPTION DCFOR	.OPTION GMINDC	.OPTION OFF
.OPTION DCHOLD	.OPTION GRAMP	.OPTION RESMIN
.OPTION DCSTEP	.OPTION GSHDC	.OPTION SYMB
.OPTION DCON	.OPTION GSHUNT	
.OPTION DCTRAN	.OPTION ICSWEEP	

DC Initialization Control Options

.OPTION ABSTOL	.OPTION GDCPATH	.OPTION MAXAMP
.OPTION CAPTAB	.OPTION GRAMP	.OPTION NEWTOL
.OPTION CSHDC	.OPTION GSHDC	.OPTION NOPIV
.OPTION DCCAP	.OPTION GSHUNT	.OPTION OFF
.OPTION DCFOR	.OPTION ICSWEEP	.OPTION PIVOT
.OPTION DCHOLD	.OPTION ITLPTRAN	.OPTION PIVREF
.OPTION DCIC	.OPTION ITL1	.OPTION PIVTOL
.OPTION DCSTEP	.OPTION ITL2	.OPTION RESMIN
.OPTION DV	.OPTION KCLTEST	.OPTION SPARSE

Transient and AC Small Signal Analysis Options

Transient/AC Accuracy Options

.OPTION ABSH	.OPTION DI	.OPTION RELQ
.OPTION ABSV	.OPTION GMIN	.OPTION RELTOL
.OPTION ACCURATE	.OPTION GSHUNT	.OPTION RELV

3: Options in HSPICE Netlists

Transient and AC Small Signal Analysis Options

<code>.OPTION ACOUT</code>	<code>.OPTION MAXAMP</code>	<code>.OPTION RISETIME</code>
<code>.OPTION CHGTOL</code>	<code>.OPTION RELH</code>	<code>.OPTION TRTOL</code>
<code>.OPTION CSHUNT</code>	<code>.OPTION RELI</code>	<code>.OPTION VNTOL</code>

Transient/AC Speed Options

<code>.OPTION AUTOSTOP</code>	<code>.OPTION BYTOL</code>	<code>.OPTION MBYPASS</code>
<code>.OPTION BKPSIZ</code>	<code>.OPTION FAST</code>	<code>.OPTION SCALE</code>
<code>.OPTION BYPASS</code>	<code>.OPTION ITLPZ</code>	

Transient/AC Timestep Options

<code>.OPTION ABSVAR</code>	<code>.OPTION FT</code>	<code>.OPTION ITL4</code>
<code>.OPTION DELMAX</code>	<code>.OPTION IMAX</code>	<code>.OPTION ITL5</code>
<code>.OPTION DVDT</code>	<code>.OPTION IMIN</code>	<code>.OPTION TIMERES</code>
<code>.OPTION FS</code>	<code>.OPTION ITL3</code>	

Transient/AC Algorithm Options

<code>.OPTION DVTR</code>	<code>.OPTION ITL5</code>	<code>.OPTION PURETP</code>
<code>.OPTION IMAX</code>	<code>.OPTION LVLTIM</code>	<code>.OPTION RUNLVL</code>
<code>.OPTION IMIN</code>	<code>.OPTION MAXORD</code>	<code>.OPTION TRCON</code>
<code>.OPTION ITL3</code>	<code>.OPTION METHOD</code>	
<code>.OPTION ITL4</code>	<code>.OPTION MU</code>	

.BIASCHK Options

<code>.OPTION BIASFILE</code>	<code>.OPTION BIAWARN</code>
-------------------------------	------------------------------

Transient Control Options

Transient Control Method Options

.OPTION BYPASS	.OPTION INTERP	.OPTION TRCON
.OPTION CSHUNT	.OPTION ITRPRT	.OPTION WACC
.OPTION DVDT	.OPTION MAXORD	
.OPTION GSHUNT	.OPTION METHOD	

Transient Control Tolerance Options

.OPTION ABSH	.OPTION FAST	.OPTION RELTOL
.OPTION ABSV	.OPTION MAXAMP	.OPTION RELV
.OPTION ABSVAR	.OPTION MBYPASS	.OPTION RELVAR
.OPTION ACCURATE	.OPTION MU	.OPTION SLOPETOL
.OPTION BYTOL	.OPTION RELH	.OPTION TIMERES
.OPTION CHGTOL	.OPTION RELI	.OPTION TRTOL
.OPTION DI	.OPTION RELQ	.OPTION VNTOL

Transient Control Limit Options

.OPTION AUTOSTOP	.OPTION FT	.OPTION ITL4
.OPTION BKPSIZ	.OPTION GMIN	.OPTION ITL5
.OPTION DELMAX	.OPTION IMAX	.OPTION RMAX
.OPTION DVTR	.OPTION IMIN	.OPTION RMIN
.OPTION FS	.OPTION ITL3	.OPTION VFLOOR

Transient Control Matrix Options

`.OPTION GMIN` `.OPTION PIVOT`

Iteration Count Dynamic Timestep Options

`.OPTION IMAX` `.OPTION IMIN`

Input/Output Options

`.OPTION INTERP` `.OPTION MEASFILE` `.OPTION OPTLST`
`.OPTION ITRPRT` `.OPTION MEASOUT` `.OPTION PUTMEAS`
`.OPTION MEASDGT` `.OPTION MEASSORT` `.OPTION UNWRAP`
`.OPTION MEASFAIL` `.OPTION MCBRIEF`

AC Control Options

`.OPTION ABSH` `.OPTION DI` `.OPTION RELH`
`.OPTION ACOUT` `.OPTION MAXAMP` `.OPTION UNWRAP`

Common Model Interface Options

`.OPTION CMIFLAG` `.OPTION CUSTCMI`

Verilog-A Options

`.OPTION SPMODEL` `.OPTION VAMODEL`

.OPTION ABSH

Syntax

```
.OPTION ABSH=x
```

Description

Sets the absolute current change, through voltage-defined branches (voltage sources and inductors). Use this option with options `DI` and `RELH` to check for current convergence. The default is `0.0`.

See Also

[.OPTION DI](#)

[.OPTION RELH](#)

.OPTION ABSI

Syntax

```
.OPTION ABSI=x
```

Description

Sets the absolute error tolerance for branch currents in diodes, BJTs, and JFETs, during DC and transient analysis. Decrease `ABSI`, if accuracy is more important than convergence time.

To analyze currents less than 1 nanoamp, change `ABSI` to a value at least two orders of magnitude smaller than the minimum expected current.

The default is $1e-9$ when `KCLTEST = 0` or $1e-6$ for `KCLTEST = 1`.

See Also

- [.DC](#)
- [.OPTION KCLTEST](#)
- [.TRAN](#)

.OPTION ABSMOS

Syntax

```
.OPTION ABSMOS=x
```

Description

Current error tolerance (for MOSFET devices) in DC or transient analysis. The `ABSMOS` setting determines whether the drain-to-source current solution has converged. The drain-to-source current converged if:

- The difference between the drain-to-source current in the last iteration, versus the present iteration, is less than `ABSMOS`, or
- This difference is greater than `ABSMOS`, but the percent change is less than `RELMOS`.

If other accuracy tolerances also indicate convergence, HSPICE or HSPICE RF solves the circuit at that timepoint, and calculates the next timepoint solution. For low-power circuits, optimization, and single transistor simulations, set `ABSMOS = 1e-12`. Default is `1e-6` (amperes).

See Also

[.DC](#)
[.OPTION RELMOS](#)
[.TRAN](#)

3: Options in HSPICE Netlists

.OPTION ABSTOL

.OPTION ABSTOL

Syntax

```
.OPTION ABSTOL=x
```

Description

Sets the absolute error tolerance for branch currents for DC and transient analysis. Decrease `ABSTOL`, if accuracy is more important than convergence time. `ABSTOL` is the same as `ABSI`.

See Also

[.DC](#)

[.OPTION ABSI](#)

[.TRAN](#)

.OPTION ABSV

Syntax

```
.OPTION ABSV=x
```

Description

Sets absolute minimum voltage for DC and transient analysis. `ABSV` is the same as `VNTOL`.

- If accuracy is more critical than convergence, decrease `ABSV`.
- If you need voltages less than 50 microvolts, reduce `ABSV` to two orders of magnitude less than the smallest desired voltage. This ensures at least two significant digits.

Typically, you do not need to change `ABSV`, except to simulate a high-voltage circuit. A reasonable value for 1000-volt circuits is 5 to 50 millivolts. The default is 50 (microvolts).

You can use `ABSV` in HSPICE, but not HSPICE RF.

See Also

[.DC](#)
[.OPTION VNTOL](#)
[.TRAN](#)

.OPTION ABSVAR

Syntax

```
.OPTION ABSVAR=x
```

Description

Sets the absolute limit for the maximum voltage change, from one time point to the next. Use this option with `.OPTION DVDT`. If the simulator produces a convergent solution that is greater than `ABSVAR`, then HSPICE discards the solution, sets the timestep to a smaller value, and recalculates the solution. This is called a timestep reversal. The default is 0.5 (volts).

For additional information, see section “DVDT Dynamic Timestep Algorithm” in the *HSPICE Simulation and Analysis User Guide*.

You can use `ABSVAR` in HSPICE, but not in HSPICE RF.

See Also

[.OPTION DVDT](#)

.OPTION ABSVDC

Syntax

```
.OPTION ABSVDC=x
```

Description

Sets the minimum voltage for DC and transient analysis. If accuracy is more critical than convergence, decrease `ABSVDC`. If you need voltages less than 50 micro-volts, reduce `ABSVDC` to two orders of magnitude less than the smallest voltage. This ensures at least two digits of significance. Typically, you do not need to change `ABSVDC`, unless you simulate a high-voltage circuit. For 1000-volt circuits, a reasonable value is 5 to 50 millivolts.

The default is the `.OPTION VNTOL` setting (`VNTOL` default = 50 mV).

See Also

- [.DC](#)
- [.OPTION VNTOL](#)
- [.TRAN](#)

.OPTION ACCT

Syntax

```
.OPTION ACCT
```

```
.OPTION ACCT=[ 1 | 2 ]
```

Example 1

```
.OPTION ACCT=2
```

The ratio of `TOT.ITER` to `CONV.ITER` is the best measure of simulator efficiency. The theoretical ratio is 2:1. In this example the ratio was 2.57:1. SPICE generally has a ratio from 3:1 to 7:1.

In transient analysis, the ratio of `CONV.ITER` to `# POINTS` is the measure of the number of points evaluated, to the number of points printed. If this ratio is greater than about 4:1, the convergence and time step control tolerances might be too tight for the simulation.

Description

The `ACCT` option in HSPICE generates a detailed accounting report.

Argument	Definition
<code>.OPTION ACCT</code>	Enables reporting.
<code>.OPTION ACCT = 1</code> (default)	Is the same as <code>ACCT</code> , without arguments.
<code>.OPTION ACCT = 2</code>	Enables reporting, and matrix statistic reporting.

See Also

[.DC](#)
[.TRAN](#)

.OPTION ACCURATE

Syntax

```
.OPTION ACCURATE=x
```

Description

Selects a time algorithm that uses `LVLTIM = 3` and `DVDT = 2` for circuits such as high-gain comparators. Use this option with circuits that combine high gain and large dynamic range to guarantee accurate solutions in HSPICE or HSPICE RF. When set to 1, this option sets these control options:

```
LVLTIM = 3
DVDT = 2
RELVAR = 0.2
ABSVAR = 0.2
FT = 0.2
RELMOS = 0.01
```

The default is 0.

See Also

- [.OPTION ABSVAR](#)
- [.OPTION DVDT](#)
- [.OPTION FT](#)
- [.OPTION LVLTIM](#)
- [.OPTION RELMOS](#)
- [.OPTION RELVAR](#)

3: Options in HSPICE Netlists

.OPTION ACOUT

.OPTION ACOUT

Syntax

```
.OPTION ACOUT=x
```

Description

AC output calculation method for the difference in values of magnitude, phase, and decibels. Use these values for prints and plots. The default is 1.

The default (`ACOUT = 1`) selects the HSPICE method, which calculates the difference of the magnitudes of the values. The SPICE method, `ACOUT = 0`, calculates the magnitude of the differences in HSPICE.

You can use this option in HSPICE, but not in HSPICE RF.

.OPTION ALT999 or ALT9999

Syntax

```
.OPTION ALT999
```

```
.OPTION ALT9999
```

Description

This option was developed to allow the `.GRAPH` statement to create more output files when you ran `.ALTER` simulations.

This option is obsolete starting with version 2003.09. Without this option, HSPICE can now generate up to 10,000 unique files.

See Also

[.ALTER](#)
[.GRAPH](#)

OPTION ALTCC

Syntax

```
.OPTION ALTCC=x
```

Description

Enables HSPICE to only read the input netlist once for multiple `.ALTER` statements.

`ALTCC = 1` enables reading input netlist only once for multiple `.ALTER` statements.

`ALTCC = 0` or `-1` disables : HSPICE or HSPICE RF does not output a warning message during transient analysis. HSPICE or HSPICE RF outputs the results, after this transient analysis.

Note: You can use `.OPTION ALTCC` or `.OPTION ALTCC=1` to ignore parsing of an input netlist before an `.ALTER` statement in the process of standard cell library characterization only when an `.ALTER` statement changes parameters, source stimulus, analysis, or passive elements. Otherwise, this option is ignored.

See Also

[.ALTER](#)

.OPTION ALTCHK

Syntax

```
.OPTION ALTCHK=x
```

Description

By default, HSPICE automatically reports topology errors in the latest elements in your top-level netlist. It also reports errors in elements that you redefine by using the `.ALTER` statement (altered netlist).

To disable topology checking in redefined elements (that is, to check topology only in the top-level netlist, but not in the altered netlist), set:

```
.option altchk=0
```

By default, `.OPTION ALTCHK` is set to 1:

```
.option altchk=1  
.option altchk
```

This enables topology checking in elements that you redefine using the `.ALTER` statement. HSPICE RF does not support `.ALTER` statements.

See Also

[.ALTER](#)

3: Options in HSPICE Netlists

.OPTION ARTIST

.OPTION ARTIST

Syntax

```
.OPTION ARTIST=x
```

Description

ARTIST = 2 enables the Cadence Analog Artist interface. This option requires a specific license.

.OPTION ASPEC

Syntax

```
.OPTION ASPEC=x
```

Description

Sets HSPICE or HSPICE RF to ASPEC-compatibility mode. When you set this option, the simulator reads ASPEC models and netlists, and the results are compatible. The default is 0 (HSPICE mode).

If you set `ASPEC`, the following model parameters default to ASPEC values:

- `ACM = 1`: Changes the default values for `CJ`, `IS`, `NSUB`, `TOX`, `U0`, and `UTRA`.
- *Diode Model*: `TLEV = 1` affects temperature compensation for `PB`.
- *MOSFET Model*: `TLEV = 1` affects `PB`, `PHB`, `VTO`, and `PHI`.
- *SCALM, SCALE*: Sets the model scale factor to microns for length dimensions.
- *WL*: Reverses implicit order for stating width and length in a MOSFET statement. The default (`WL=0`) assigns the length first, then the width.

See Also

[.OPTION SCALE](#)
[.OPTION SCALM](#)
[.OPTION WL](#)

3: Options in HSPICE Netlists

.OPTION AUTOSTOP

.OPTION AUTOSTOP

Syntax

```
.OPTION AUTOSTOP
```

-or-

```
.OPTION AUTOSTOP='expression'
```

Example

```
.option autostop='m1&& m2||m4'
.meas tran m1 trig v(bd_a0) val='ddv/2' fall=1
targ v(re_bd) val='ddv/2' rise=1
.meas tran m2 trig v(bd_a0) val='ddv/2' fall=2
targ v(re_bd) val='ddv/2' rise=2
.meas tran m3 trig v(bd_a0) val='ddv/2' rise=2
targ v(re_bd) val='ddv/2' rise=3
.meas tran m4 trig v(bd_a0) val='ddv/2' fall=3
targ v(re_bd) val='ddv/2' rise=4
.meas tran m5 trig v(bd_a0) val='ddv/2' rise=3 targ
v(re_bd) val='ddv/2' rise=5
```

In this example, when either m1 and m2 are obtained, or just m4 is obtained, the transient analysis ends.

Description

Stops a transient analysis in HSPICE or HSPICE RF, after calculating all TRIG-TARG, FIND-WHEN, and FROM-TO measure functions. This option can substantially reduce CPU time. You can use the AUTOSTOP option with any measure type. You can also use the result of the preceding measurement as the next measured parameter.

When using .OPTION AUTOSTOP='expression', the 'expression' can only involve measure results, a logical AND (&&), or a logical OR(||). Using these types of expressions ends the simulation if any one of a set of .MEASURE statements succeeds, even if the others are not completed.

Also terminates the simulation, after completing all .MEASURE statements. This is of special interest when testing corners.

See Also

[.MEASURE](#)

.OPTION BADCHR

Syntax

```
.OPTION BADCHR
```

Description

Generates a warning, if it finds a non-printable character in an input file.

3: Options in HSPICE Netlists

.OPTION BEEP

.OPTION BEEP

Syntax

```
.OPTION BEEP=x
```

Description

BEEP = 1 sounds an audible tone when simulation returns a message, such as:

```
info: HSPICE job completed.
```

BEEP = 0 turns off the audible tone.

.OPTION BIASFILE

Syntax

```
.OPTION BIASFILE=x
```

Example

```
OPTION BIASFILE='biaschk/mos.bias'
```

Description

If you use this option, HSPICE or HSPICE RF outputs the results of all `.BIASCHK` commands to a file that you specify. If you do not set this option, HSPICE or HSPICE RF outputs the `.BIASCHK` results to the `*.lis` file.

See Also

[.BIASCHK](#)

3: Options in HSPICE Netlists

.OPTION BIAWARN

.OPTION BIAWARN

Syntax

```
.OPTION BIAWARN=x
```

Example

```
.OPTION BIAWARN=1
```

Description

BIAWARN = 1: HSPICE or HSPICE RF immediately outputs a warning message when any local max bias voltage exceeds the limit during transient analysis. After this transient analysis, HSPICE or HSPICE RF outputs the results summary as filtered by noise.

BIAWARN = 0 (default): HSPICE or HSPICE RF does not output a warning message during transient analysis. HSPICE or HSPICE RF outputs the results, after this transient analysis.

See Also

[.TRAN](#)

.OPTION BINPRINT

Syntax

```
.OPTION BINPRINT
```

Description

Outputs the binning parameters of the CMI MOSFET model. Currently available only for Level 57.

3: Options in HSPICE Netlists

.OPTION BKPSIZ

.OPTION BKPSIZ

Syntax

```
.OPTION BKPSIZ=x
```

Description

Sets the size of the breakpoint table. The default is 5000. This is an old option, provided only for backward-compatibility.

.OPTION BRIEF

Syntax

```
.OPTION BRIEF=x
```

Description

Stops printback of the data file, until HSPICE or HSPICE RF finds an `.OPTION BRIEF = 0` or the `.END` statement. It also resets the `LIST`, `NODE`, and `OPTS` options, and sets `NOMOD.BRIEF = 0` enables printback. The `NXX` option is the same as `BRIEF`.

See Also

- [.END](#)
- [.OPTION LIST](#)
- [.OPTION NODE](#)
- [.OPTION NXX](#)
- [.OPTION OPTS](#)

.OPTION BYPASS

Syntax

```
.OPTION BYPASS=x
```

Description

Bypasses model evaluations, if the terminal voltages do not change. Can be 0 (off), 1 (on), or 2 (applies to BSIM3v3 and BSIM4 in special cases). To speed-up simulation, this option does not update the status of latent devices. To enable bypassing, set `.OPTION BYPASS = 1` for MOSFETs, MESFETs, JFETs, BJTs, or diodes. Default = 1.

Use the `BYPASS` algorithm cautiously. Some circuit types might not converge, and might lose accuracy in transient analysis and operating-point calculations.

.OPTION BYTOL

Syntax

```
.OPTION BYTOL=x
```

Description

Specifies a voltage tolerance, at which a MOSFET, MESFET, JFET, BJT, or diode becomes latent. HSPICE does not update status of latent devices. The default = MBYPASS x VNTOL.

You can use this option in HSPICE, but not in HSPICE RF.

See Also

[.OPTION MBYPASS](#)
[.OPTION VNTOL](#)

.OPTION CAPTAB

Syntax

```
.OPTION CAPTAB
```

Description

Prints table of single-plate node capacitances for diodes, BJTs, MOSFETs, JFETs, and passive capacitors, at each operating point.

.OPTION CDS

Syntax

```
.OPTION CDS=x
```

Description

CDS = 2 produces a Cadence WSF (ASCII format) post-analysis file for Opus™. This option requires a specific license. The CDS option is the same as the SDA option.

See Also

[.OPTION SDA](#)

3: Options in HSPICE Netlists

.OPTION CHGTOL

.OPTION CHGTOL

Syntax

```
.OPTION CHGTOL=x
```

Description

Sets a charge error tolerance, if you set `LVLTIM = 2`. Use `CHGTOL` with `RELQ` to set the absolute and relative charge tolerance for all HSPICE capacitances. The default is $1e-15$ (coulomb).

See Also

- [.OPTION CHGTOL](#)
- [.OPTION LVLTIM](#)
- [.OPTION RELQ](#)

.OPTION CMIFLAG

Syntax

```
.OPTION CMIFLAG
```

Description

This option signals to load the dynamically-linked Common Model Interface (CMI) library, libCMImodel.

See Also

[.OPTION CUSTCMI](#)

.OPTION CO

Syntax

```
.OPTION CO=<column_width>
```

Example

```
* Narrow print-out (default)
.OPTION CO=80
* Wide print-out
.OPTION CO=132
```

Description

The number of output variables that print on a single line of output, is a function of the number of columns. Use `.OPTION CO` to set the column width for print-outs in HSPICE.

HSPICE RF does not support the `.OPTION CO` statement.

You can set up to five output variables per 80-column output, and up to eight output variables per 132-column output with twelve characters per column. HSPICE automatically creates additional print statements and tables for all output variables beyond the number that the `CO` option specifies. The default is 80.

Argument	Definition
column_width	The number of characters in a single line of output.

See Also

[.WIDTH](#)

.OPTION CONVERGE

Syntax

`.OPTION CONVERGE=x`

Description

Invokes different methods to solve non-convergence problems.

- `CONVERGE = -1` : Use with `DCON = -1` to disable autoconvergence.
- `CONVERGE = 0` : Autoconvergence (default).
- `CONVERGE = 1` : Uses the Damped Pseudo Transient algorithm. If simulation does not converge within the set CPU time (in the `CPTIME` control option), then simulation halts.
- `CONVERGE = 2` : Uses a combination of `DCSTEP` and `GMINDC` ramping. Not used in the autoconvergence flow.
- `CONVERGE = 3` : Invokes the source-stepping method. Not used in the autoconvergence flow.
- `CONVERGE = 4` : Uses the `gmath` ramping method.

Even you did not set it in an `.OPTION` statement, the `CONVERGE` option activates if a matrix floating-point overflows, or if HSPICE or HSPICE RF reports a *timestep too small* error. The default is 0.

If a matrix floating-point overflows, then `CONVERGE = 1`.

See Also

[.OPTION DCON](#)
[.OPTION DCSTEP](#)
[.OPTION DCTRAN](#)
[.OPTION GMINDC](#)

3: Options in HSPICE Netlists

.OPTION CPTIME

.OPTION CPTIME

Syntax

```
.OPTION CPTIME=x
```

Description

Sets the maximum CPU time, in seconds, allotted for this simulation job. When the time allowed for the job exceeds `CPTIME`, HSPICE prints or plots the results up to that point, and concludes the job. Use this option if you are uncertain how long the simulation will take, especially when you debug new data files. Default is `1e7` (400 days).

See Also

[.OPTION LIMTIM](#)

.OPTION CSDF

Syntax

```
.OPTION CSDF=x
```

Description

Selects Common Simulation Data Format (Viewlogic-compatible graph data file format).

3: Options in HSPICE Netlists

.OPTION CSHDC

.OPTION CSHDC

Syntax

```
.OPTION CSHDC=x
```

Description

The same option as CSHUNT; use only with the CONVERGE option.

You can use the CSHDC option in HSPICE, but not in HSPICE RF.

See Also

[.OPTION CONVERGE](#)

[.OPTION CSHUNT](#)

.OPTION CSHUNT

Syntax

```
.OPTION CSHUNT=x
```

Description

Capacitance added from each node to ground in HSPICE or HSPICE RF. Add a small `CSHUNT` to each node to solve internal timestep too small problems, caused by high-frequency oscillations or numerical noise. The default is 0.

3: Options in HSPICE Netlists

.OPTION CUSTCMI

.OPTION CUSTCMI

Syntax

```
.OPTION CUSTCMI=x
```

Description

You set `.OPTION CUSTCMI=1` jointly with `.OPTION CMIFLAG` to turn on gate direct tunneling current modeling and instance parameter support for customer CMI. You set `.OPTION CUSTCMI=0` to turn off that feature.

See Also

[.OPTION CMIFLAG](#)

.OPTION CVTOL

Syntax

```
.OPTION CVTOL=x
```

Description

Changes the number of numerical integration steps when calculating the gate capacitor charge for a MOSFET by using `CAPOP = 3`. See the discussion of `CAPOP = 3` in the “Overview of MOSFETS” chapter of the *HSPICE Elements and Device Models Manual* for explicit equations and discussion.

You can use the `.OPTION CVTOL` statement in HSPICE, but not in HSPICE RF.

3: Options in HSPICE Netlists

.OPTION D_IBIS

.OPTION D_IBIS

Syntax

```
.OPTION D_IBIS='ibis_files_directory'
```

Example

```
.OPTION d_ibis='/home/user/ibis/models'
```

Description

The `.OPTION D_IBIS` option specifies the directory containing the IBIS files. If you specify several directories, then the simulation looks for IBIS files in the local directory (the directory from which you run the simulation). It then checks the directories specified through `.OPTION D_IBIS` in the order that `.OPTION` cards appear in the netlist. You can use the `D_IBIS` option to specify up to four directories.

.OPTION DCAP

Syntax

```
.OPTION DCAP
```

Description

Selects equations, which HSPICE or HSPICE RF uses to calculate depletion capacitance for Level 1 and 3 diodes, and BJTs. The *HSPICE Elements and Device Models Manual* describes these equations.

.OPTION DCCAP

Syntax

```
.OPTION DCCAP=x
```

Description

Generates C-V plots. Prints capacitance values of a circuit (both model and element), during a DC analysis. You can use a DC sweep of the capacitor to generate C-V plots. If not set, MOS device or voltage-variable capacitance values will not be evaluated and the printed value will be zero. The default is 0 (off).

See Also

[.DC](#)

.OPTION DCFOR

Syntax

```
.OPTION DCFOR=x
```

Description

Use with `.OPTION DCHOLD` and the `.NODESET` statement to enhance DC convergence.

DCFOR sets the number of iterations to calculate, after a circuit converges in the steady state. The number of iterations after convergence is usually zero so DCFOR adds iterations (and computation time) to the DC circuit solution. DCFOR ensures that a circuit actually, not falsely, converges. The default is 0.

See Also

- [.DC](#)
- [.NODESET](#)
- [.OPTION DCHOLD](#)

.OPTION DCHOLD

Syntax

```
.OPTION DCHOLD=x
```

Description

Use DCFOR and DCHOLD together to initialize DC analysis. You can use the DCHOLD option in HSPICE, but not in HSPICE RF. DCFOR and DCHOLD enhance the convergence properties of a DC simulation. DCFOR and DCHOLD work with the .NODESET statement. The default is 1.

DCHOLD specifies how many iterations to hold a node, at the .NODESET voltage values. The effects of DCHOLD on convergence differ, according to the DCHOLD value, and the number of iterations before DC convergence.

If a circuit converges in the steady state in fewer than DCHOLD iterations, the DC solution includes the values set in .NODESET.

If a circuit requires more than DCHOLD iterations to converge, HSPICE or HSPICE RF ignores the values set in the .NODESET statement, and calculates the DC solution by using the .NODESET fixed-source voltages open circuited.

See Also

[.DC](#)

[.NODESET](#)

[.OPTION DCFOR](#)

.OPTION DCIC

Syntax

```
.OPTION DCIC=x
```

Description

If `DCIC=1` (default), each point in a DC sweep analysis acts like an operating point and all `.IC` commands in the netlist are used.

If `DCIC=0`, `.IC` commands in the netlist are ignored for DC sweep analysis.

See Also

[.IC](#)
[.DC](#)

3: Options in HSPICE Netlists

.OPTION DCON

.OPTION DCON

Syntax

.OPTION DCON=x

Description

If a circuit cannot converge, HSPICE or HSPICE RF automatically sets DCON = 1, and calculates the following:

$$DV = \max\left(0.1, \frac{V_{max}}{50}\right), \text{ if } DV = 1000$$

$$GRAMP = \max\left(6, \log_{10}\left(\frac{I_{max}}{GMINDC}\right)\right) \quad ITL1 = ITL1 + 20 \cdot GRAMP$$

V_{max} is the maximum voltage, and I_{max} is the maximum current.

- If the circuit still cannot converge, HSPICE or HSPICE RF sets DCON = 2, which sets $DV = 1e6$.
- If the circuit uses discontinuous models or uninitialized flip-flops, simulation might not converge. Set DCON = -1 and CONVERGE = -1 to disable autoconvergence. HSPICE lists all non-convergent nodes and devices.

See Also

[.OPTION CONVERGE](#)

[.OPTION DV](#)

.OPTION DCSTEP

Syntax

```
.OPTION DCSTEP=x
```

Description

Converts DC model and element capacitors to a conductance to enhance DC convergence properties. HSPICE divides the value of the element capacitors by `DCSTEP` to model DC conductance. The default is 0 (seconds).

See Also

[.DC](#)

3: Options in HSPICE Netlists

.OPTION DCTRAN

.OPTION DCTRAN

Syntax

```
.OPTION DCTRAN=x
```

Description

Invokes different methods to solve non-convergence problems. DCTRAN is an alias for CONVERGE.

See Also

[.OPTION CONVERGE](#)

.OPTION DEFAD

Syntax

```
.OPTION DEFAD=x
```

Description

The default MOSFET drain diode area in HSPICE. The default is 0.

3: Options in HSPICE Netlists

.OPTION DEFAS

.OPTION DEFAS

Syntax

```
.OPTION DEFAS=x
```

Description

The default MOSFET source diode area in HSPICE. The default is 0.

.OPTION DEFL

Syntax

```
.OPTION DEFL=x
```

Description

The default MOSFET channel length in HSPICE. The default is $1e-4\text{m}$.

.OPTION DEFNRD

Syntax

```
.OPTION DEFNRD=x
```

Description

The default number of squares for the drain resistor on a MOSFET. The default is 0.

.OPTION DEFNRS

Syntax

```
.OPTION DEFNRS=x
```

Description

The default number of squares for the source resistor on a MOSFET. The default is 0.

3: Options in HSPICE Netlists

.OPTION DEFDPD

.OPTION DEFDPD

Syntax

```
.OPTION DEFDPD=x
```

Description

The default MOSFET drain diode perimeter in HSPICE. The default is 0.

.OPTION DEFPS

Syntax

```
.OPTION DEFPS=x
```

Description

The default MOSFET source diode perimeter in HSPICE. The default is 0.

3: Options in HSPICE Netlists

.OPTION DEFW

.OPTION DEFW

Syntax

```
.OPTION DEFW=x
```

Description

The default MOSFET channel width in HSPICE. The default is $1e-4m$.

.OPTION DELMAX

Syntax

```
.OPTION DELMAX=x
```

Description

Sets the maximum Delta of the internal timestep. HSPICE automatically sets the `DELMAX` value, based on timestep control factors. The initial `DELMAX` value, shown in the HSPICE output listing, is generally not the value used for simulation.

You can use the `DELMAX` option in HSPICE, but not in HSPICE RF.

.OPTION DI

Syntax

```
.OPTION DI=x
```

Description

Sets the maximum iteration-to-iteration current change, through voltage-defined branches (voltage sources and inductors). Use this option only if the value of the `ABSH` control option is greater than 0. The default is 0 . 0.

See Also

[.OPTION ABSH](#)

.OPTION DIAGNOSTIC

Syntax

```
.OPTION DIAGNOSTIC
```

Description

Logs the occurrence of negative model conductances.

.OPTION DLENCSDF

Syntax

```
.OPTION DLENCSDF=x
```

Description

If you use the Common Simulation Data Format (Viewlogic graph data file format) as the output format, this digit length option specifies how many digits to include in scientific notation (exponents), or to the right of the decimal point. Valid values are any integer from 1 to 10, and the default is 5.

If you assign a floating decimal point, or if you specify less than 1 or more than 10 digits, HSPICE or HSPICE RF uses the default. For example, it places 5 digits to the right of a decimal point.

.OPTION DV

Syntax

```
.OPTION DV=x
```

Description

Maximum iteration-to-iteration voltage change for all circuit nodes in both DC and transient analysis. High-gain bipolar amplifiers can require values of 0.5 to 5.0 to achieve a stable DC operating point. Large CMOS digital circuits frequently require about 1 volt. The default is 1000 (or 1e6 if DCON = 2).

See Also

[.DC](#)
[.OPTION DCON](#)
[.TRAN](#)

.OPTION DVDT

Syntax

```
.OPTION DVDT=x
```

Description

Adjusts the timestep, based on rates of change for node voltage. The default is 4.

- 0 - original algorithm
- 1 - fast
- 2 - accurate
- 3,4 - balance speed and accuracy
- You can use the DVDT option in HSPICE, but not in HSPICE RF. ACCURATE also increases the accuracy of the results.

For additional information, see section “DVDT Dynamic Timestep Algorithm” in the *HSPICE Simulation and Analysis User Guide*.

See Also

[.OPTION ACCURATE](#)

.OPTION DVTR

Syntax

```
.OPTION DVTR=x
```

Description

Limits voltage in transient analysis. The default is 1000.

3: Options in HSPICE Netlists

.OPTION EPSMIN

.OPTION EPSMIN

Syntax

```
.OPTION EPSMIN=x
```

Description

Specifies the smallest number that a computer can add or subtract, a constant value. The default is $1e-28$.

.OPTION EXPLI

Syntax

```
.OPTION EXPLI=x
```

Description

Current-explosion model parameter. PN junction characteristics, above the explosion current, are linear. HSPICE or HSPICE RF determines the slope at the explosion point. This improves simulation speed and convergence.

The default is 0.0 amp/AREAeff.

3: Options in HSPICE Netlists

.OPTION EXPMAX

.OPTION EXPMAX

Syntax

```
.OPTION EXPMAX=x
```

Description

Specifies the largest exponent that you can use for an exponential, before overflow occurs. Typical value for an IBM platform is 350.

.OPTION FAST

Syntax

```
.OPTION FAST
```

Description

Sets additional options, which increase simulation speed with minimal loss of accuracy.

To speed-up simulation, this option does not update the status of latent devices. Use this option for MOSFETs, MESFETs, JFETs, BJTs, and diodes. The default is 0.

You can use `FAST` in HSPICE, but not HSPICE RF.

A device is latent, if its node voltage variation (from one iteration to the next) is less than the value of either the `BYTOL` control option, or the `BYPASSTOL` element parameter. (If `FAST` is on, HSPICE sets `BYTOL` to different values for different types of device models.)

Besides the `FAST` option, you can also use the `NOTOP` and `NOELCK` options to reduce input pre-processing time. Increasing the value of the `MBYPASS` or `BYTOL` option, also helps simulations to run faster, but can reduce accuracy.

See Also

- [.OPTION BYTOL](#)
- [.OPTION MBYPASS](#)
- [.OPTION NOELCK](#)
- [.OPTION NOTOP](#)

.OPTION FFTOUT

Syntax

```
.OPTION FFTOUT=x
```

Description

Prints 30 harmonic fundamentals, sorted by size, THD, SNR, and SFDR, but only if you specify a `.OPTION FFTOUT` statement and a `.FFT freq=xxx` statement.

You can use the `.OPTION FFTOUT` statement in HSPICE, but not in HSPICE RF.

See Also

[.FFT](#)

.OPTION FS

Syntax

```
.OPTION FS=x
```

Description

Decreases Delta (internal timestep) by the specified fraction of a timestep (TSTEP) for the first time point of a transient. Decreases the FS value to help circuits that have timestep convergence difficulties. DVDT = 3 uses FS to control the timestep.

$$Delta = FS \cdot [MIN(TSTEP, DELMAX, BKPT)]$$

- You specify DELMAX.
- BKPT is related to the breakpoint of the source.
- The .TRAN statement sets TSTEP. The default is 0.25.

You can use .OPTION FS in HSPICE, but not HSPICE RF.

See Also

[.OPTION DELMAX](#)
[.OPTION DVDT](#)
[.TRAN](#)

3: Options in HSPICE Netlists

.OPTION FT

.OPTION FT

Syntax

```
.OPTION FT=x
```

Description

Decreases Delta (the internal timestep), by a specified fraction of a timestep (TSTEP) for an iteration set that does not converge. If DVDT = 2 or DVDT = 4, FT controls the timestep. The default is 0.25.

See Also

[.OPTION DVDT](#)

[.TRAN](#)

.OPTION GDCPATH

Syntax

```
.OPTION GDCPATH[ =x ]
```

Description

Adds conductance to nodes having no DC path to ground. You use this option to help solve no DC path to ground problems. If you specify `GDCPATH` in a netlist without a value, that value is assumed to be $1e-15$ (the default). The default is 0 when not specified.

3: Options in HSPICE Netlists

.OPTION GENK

.OPTION GENK

Syntax

```
.OPTION GENK=x
```

Description

Automatically computes second-order mutual inductance for several coupled inductors. The default is 1, which enables the calculation.

.OPTION GMAX

Syntax

```
.OPTION GMAX=x
```

Description

Conductance in parallel with a current source for `.IC` and `.NODESET` initialization circuitry. Some large bipolar circuits require you to set `GMAX = 1` for convergence. The default is 100 (mho).

You can use `GMAX` in HSPICE, but not in HSPICE RF.

See Also

[.IC](#)
[.NODESET](#)

3: Options in HSPICE Netlists

.OPTION GMIN

.OPTION GMIN

Syntax

```
.OPTION GMIN=x
```

Description

Minimum conductance added to all PN junctions for a time sweep in transient analysis. The default is $1e-12$.

.OPTION GMINDC

Syntax

```
.OPTION GMINDC=x
```

Description

Conductance in parallel to all pn junctions and MOSFET nodes except *gate* for DC analysis. *GMINDC* helps overcome DC convergence problems, caused by low values of off-conductance for pn junctions and MOSFETs. You can use *GRAMP* to reduce *GMINDC*, by one order of magnitude for each step. Set *GMINDC* between $1e-4$ and the *PIVTOL* value. The default is $1e-12$.

Large values of *GMINDC* can cause unreasonable circuit response. If your circuit requires large values to converge, suspect a bad model or circuit. If a matrix floating-point overflows, and if *GMINDC* is $1.0e-12$ or less, HSPICE or HSPICE RF sets it to $1.0e-11$. HSPICE or HSPICE RF manipulates *GMINDC* in auto-converge mode.

See Also

- [.DC](#)
- [.OPTION GRAMP](#)
- [.OPTION PIVTOL](#)

.OPTION GRAMP

Syntax

```
.OPTION GRAMP=x
```

Description

HSPICE sets this value during auto-convergence (default is 0). Use GRAMP with the GMINDC option to find the smallest GMINDC value that results in DC convergence.

You can use GRAMP in HSPICE, but not HSPICE RF.

GRAMP specifies a conductance range, over which DC operating point analysis sweeps GMINDC. HSPICE replaces GMINDC values over this range, simulates each value, and uses the lowest GMINDC value where the circuit converges in a steady state.

If you sweep GMINDC between $1e-12$ mhos (default) and $1e-6$ mhos, GRAMP is 6 (value of the exponent difference, between the default and the maximum conductance limit). In this example:

- HSPICE first sets GMINDC to $1e-6$ mhos, and simulates the circuit.
- If circuit simulation converges, HSPICE sets GMINDC to $1e-7$ mhos, and simulates the circuit.
- The sweep continues until HSPICE simulates all values of the GRAMP ramp.

If the combined GMINDC and GRAMP conductance is greater than $1e-3$ mho, false convergence can occur.

See Also

[.DC](#)

[.OPTION GMINDC](#)

.OPTION GSHDC

Syntax

```
.OPTION GSHDC=x
```

Description

Adds conductance from each node to ground when calculating the DC operating point of the circuit (.OP). The default is 0.

You can use the GSHDC option in HSPICE, but not in HSPICE RF.

See Also

[.OPTION GSHUNT](#)

3: Options in HSPICE Netlists

.OPTION GSHUNT

.OPTION GSHUNT

Syntax

```
.OPTION GSHUNT=x
```

Description

Adds conductance from each node to ground. The default is 0. Add a small `GSHUNT` to each node to help solve Timestep too small problems caused by either high-frequency oscillations or numerical noise.

You can use the `GSHUNT` option in HSPICE, but not in HSPICE RF.

.OPTION H9007

Syntax

```
.OPTION H9007
```

Description

Sets default values for general-control options to correspond to values for HSPICE H9007D. If you set this option, HSPICE does not use the `EXPLI` model parameter.

See Also

[.OPTION EXPLI](#)

.OPTION HIER_SCALE

Syntax

```
.OPTION HIER_SCALE=x
```

Description

If you set the `HIER_SCALE` option, you can use the `S` parameter to scale sub-circuits.

- 0 interprets `S` as a user-defined parameter.
- 1 interprets `S` as a scale parameter.

.OPTION ICSWEEP

Syntax

```
.OPTION ICSWEEP=x
```

Description

Saves the current analysis result of a parameter or temperature sweep as the starting point in the next analysis in the sweep.

- If `ICSWEEP = 1` (default), the next analysis uses the current results.
- If `ICSWEEP = 0`, the next analysis does not use the results of the current analysis.

You can use `ICSWEEP` in HSPICE, but not in HSPICE RF.

.OPTION IMAX

Syntax

```
.OPTION IMAX=x
```

Description

Maximum timestep in timestep algorithms for transient analysis. `IMAX` sets the maximum iterations to obtain a convergent solution at a timepoint. If the number of iterations needed is greater than `IMAX`, the internal timestep (Delta) decreases, by a factor equal to the `FT` transient control option. HSPICE uses the new timestep to calculate a new solution. `IMAX` also works with the `IMIN` transient control option. `IMAX` is the same as `ITL4`. The default is 8.0.

You can use `IMAX` in HSPICE, but not HSPICE RF.

See Also

- [.OPTION FT](#)
- [.OPTION IMIN](#)
- [.OPTION ITL4](#)

.OPTION IMIN

Syntax

```
.OPTION IMIN=x
```

Description

Minimum timestep in timestep algorithms for transient analysis. `IMIN` is the minimum number of iterations required to obtain convergence. If the number of iterations is less than `IMIN`, the internal timestep (Delta) doubles.

Use this option to decrease simulation times in circuits where the nodes are stable most of the time (such as digital circuits). If the number of iterations is greater than `IMIN`, the timestep stays the same, unless the timestep exceeds the `IMAX` option. `IMIN` is the same as `ITL3`. The default is 3.0.

You can use `IMIN` in HSPICE, but not HSPICE RF.

See Also

[.OPTION IMAX](#)

[.OPTION ITL3](#)

3: Options in HSPICE Netlists

.OPTION INGOLD

.OPTION INGOLD

Syntax

```
.OPTION INGOLD=[ 0 | 1 | 2 ]
```

Example

```
.OPTION INGOLD=2
```

Description

By default, HSPICE or HSPICE RF prints variable values in engineering notation:

```
F = 1e-15    M = 1e-3
P = 1e-12    K = 1e3
N = 1e-9     X = 1e6
U = 1e-6     G = 1e9
```

In contrast to exponential form, engineering notation provides two to three extra significant digits, and aligns columns to facilitate comparison. To obtain output in exponential form, specify `.OPTION INGOLD = 1` or `2`.

Argument	Definition	Defaults
INGOLD = 0 (default)	Engineering Format	1.234K 123M
INGOLD = 1	G Format (fixed and exponential)	1.234e+03 .123
INGOLD = 2	E Format (exponential SPICE)	1.234e+03 .123e-1

See Also

[.OPTION MEASDGT](#)

.OPTION INTERP

Syntax

```
.OPTION INTERP=x
```

Description

Limits output for post-analysis tools, such as Cadence or Zuken, to only the `.TRAN` timestep intervals. By default, HSPICE outputs all convergent iterations. `INTERP` typically produces a much smaller design `.tr#` file.

Use `INTERP = 1` with caution when the netlist includes `.MEASURE` statements. To compute measure statements, HSPICE uses the post-processing output. Reducing post-processing output can lead to interpolation errors in measure results.

When you run data-driven transient analysis (`.TRAN DATA`) in an optimization routine, HSPICE forces `INTERP=1`. HSPICE supports `.TRAN DATA`; HSPICE RF does not. All measurement results are at the time points specified in the data-driven sweep. To measure only at converged internal timesteps (for example, to calculate the AVG or RMS), set `ITRPRT=1`.

See Also

- [.MEASURE](#)
- [.OPTION ITRPRT](#)
- [.TRAN](#)

.OPTION ITL1

Syntax

```
.OPTION ITL1=x
```

Description

Maximum DC iteration limit. Increasing this value rarely improves convergence in small circuits. Values as high as 400 have resulted in convergence for some large circuits with feedback (such as operational amplifiers and sense amplifiers). However, to converge, most models do not require more than 100 iterations. Set `.OPTION ACCT` to list how many iterations an operating point requires. The default is 200.

See Also

[.DC](#)

[.OPTION ACCT](#)

.OPTION ITL2

Syntax

```
.OPTION ITL2=x
```

Description

Iteration limit for the DC transfer curve. Increasing this limit improves convergence, only for very large circuits. Default is 50.

See Also

[.DC](#)

.OPTION ITL3

Syntax

```
.OPTION ITL3=x
```

Description

Minimum timestep in timestep algorithms for transient analysis. `ITL3` is the minimum number of iterations required to obtain convergence. If the number of iterations is less than `ITL3`, the internal timestep (Delta) doubles.

Use this option to decrease simulation times in circuits where the nodes are stable most of the time (such as digital circuits). If the number of iterations is greater than `IMIN`, the timestep stays the same, unless the timestep exceeds the `IMAX` option. `ITL3` is the same as `IMIN`. The default is 3.0.

You can use `IMIN` in HSPICE, but not HSPICE RF.

See Also

[.OPTION IMAX](#)

[.OPTION IMIN](#)

.OPTION ITL4

Syntax

```
.OPTION ITL4=x
```

Description

Maximum timestep in timestep algorithms for transient analysis. `ITL4` sets the maximum iterations to obtain a convergent solution at a timepoint. If the number of iterations needed is greater than `ITL4`, the internal timestep (Delta) decreases, by a factor equal to the `FT` transient control option. HSPICE uses the new timestep to calculate a new solution. `ITL4` also works with the `IMIN` transient control option. `ITL4` is the same as `IMAX`. The default is 8.0.

You can use `IMAX` in HSPICE, but not HSPICE RF.

See Also

- [.OPTION FT](#)
- [.OPTION IMAX](#)
- [.OPTION IMIN](#)

3: Options in HSPICE Netlists

.OPTION ITL5

.OPTION ITL5

Syntax

```
.OPTION ITL5=x
```

Description

Sets an iteration limit for transient analysis. If a circuit uses more than `ITL5` iterations, the program prints all results, up to that point. The default is `0.0`. `0.0` allows an infinite number of iterations.

You can use the `ITL5` option in HSPICE, but not in HSPICE RF.

.OPTION ITLPTRAN

Syntax

```
.OPTION ITLPTRAN=x
```

Description

Controls the iteration limit used in the final try of the pseudo-transient method in OP or DC analysis. If simulation fails in the final try of the pseudo-transient method, enlarge this option. The default is 30.

See Also

[.DC](#)
[.OP](#)

3: Options in HSPICE Netlists

.OPTION ITLPZ

.OPTION ITLPZ

Syntax

```
.OPTION ITLPZ=x
```

Description

Sets the iteration limit for Pole/Zero analysis. The default is 100.

You can use `ITLPZ` in HSPICE, but not in HSPICE RF.

.OPTION ITRPRT

Syntax

```
.OPTION ITRPRT
```

Description

Prints output variables at their internal time points. This option might generate a long output list.

.OPTION KCLTEST

Syntax

```
.OPTION KCLTEST=x
```

Description

Activates KCL (Kirchhoff's Current Law) test. increases simulation time, especially for large circuits, but very accurately checks the solution. The default is 0.

If you set this value to 1, HSPICE or HSPICE RF sets these options:

- Sets RELMOS and ABSMOS options to 0 (off).
- Sets ABSI to $1e-6$ A.
- Sets RELI to $1e-6$.

To satisfy the KCL test, each node must satisfy this condition:

$$|\sum i_b| < RELI \cdot \sum |i_b| + ABSI$$

In this equation, the i_b s are the node currents.

See Also

[.OPTION ABSI](#)
[.OPTION ABSMOS](#)
[.OPTION RELI](#)
[.OPTION RELMOS](#)

.OPTION KLIM

Syntax

```
.OPTION KLIM=x
```

Description

This option sets the minimum mutual inductance, below which automatic second-order mutual inductance calculation no longer proceeds. `KLIM` is unitless (analogous to coupling strength, specified in the K Element). Typical `KLIM` values are between .5 and 0.0. The default is 0.01.

3: Options in HSPICE Netlists

.OPTION LENNAM

.OPTION LENNAM

Syntax

```
.OPTION LENNAM=x
```

Description

Maximum length of names in the printout of operating point analysis results.
Default is 8, and the maximum x value=1024.

.OPTION LIMPTS

Syntax

```
.OPTION LIMPTS=x
```

Description

Number of points to print or plot in AC analysis. You do not need to set `LIMPTS` for DC or transient analysis. HSPICE spools the output file to disk. The default is 2001.

See Also

[.AC](#)
[.DC](#)
[.TRAN](#)

3: Options in HSPICE Netlists

.OPTION LIMTIM

.OPTION LIMTIM

Syntax

```
.OPTION LIMTIM=x
```

Description

Amount of CPU time reserved to generate prints and plots, if a CPU time limit (`CPTIME = x`) terminates simulation. The default is 2 (seconds), normally sufficient for short printouts and plots.

See Also

[.OPTION CPTIME](#)

.OPTION LIST

Syntax

`.OPTION LIST`

Description

This option produces an element summary of the input data to print, and calculates effective sizes of elements and the key values. The `BRIEF` option suppresses the `LIST` option.

See Also

[.OPTION BRIEF](#)
[.OPTION UNWRAP](#)
[.OPTION VFLOOR](#)

.OPTION LVLTIM

Syntax

```
.OPTION LVLTIM=x
```

Description

Selects the timestep algorithm for transient analysis.

- `LVLTIM = 1` (default) uses the DVDT timestep control algorithm.
- `LVLTIM = 2` uses the local truncation error (LTE) timestep control method. You can apply `LVLTIM = 2` to the TRAP method.
- `LVLTIM = 3` uses the DVDT timestep control method with timestep reversal.

The local truncation algorithm `LVLTIM = 2` (LTE) provides a higher degree of accuracy than `LVLTIM = 1` or `3` (DVDT). If you use this option, errors do not propagate from time point to time point, which can result in an unstable solution.

Selecting the GEAR method changes the value of `LVLTIM` to `2` automatically.

See Also

- [.OPTION CHGTOL](#)
- [.OPTION DVDT](#)
- [.OPTION FS](#)
- [.OPTION FT](#)
- [.OPTION RELQ](#)

.OPTION MAXAMP

Syntax

```
.OPTION MAXAMP=x
```

Description

Sets the maximum current, through voltage-defined branches (voltage sources and inductors). If the current exceeds the `MAXAMP` value, HSPICE or HSPICE RF reports an error. The default is 0.0.

.OPTION MAXORD

Syntax

```
.OPTION MAXORD=x
```

Description

Maximum order of integration for the GEAR method in HSPICE. The x value can be either 1 or 2.

- MAXORD = 1 uses the backward Euler integration method.
- MAXORD = 2 (default) is more stable, accurate, and practical.

See Also

[.OPTION METHOD](#)

.OPTION MBYPASS

Syntax

```
.OPTION MBYPASS=x
```

Description

Computes the default value of the BYTOL control option:

```
BYTOL = MBYPASSxVNTOL
```

Also multiplies the RELV voltage tolerance. Set MBYPASS to about 0.1 for precision analog circuits.

- Default is 1 for DVDT = 0, 1, 2, or 3.
- Default is 2 for DVDT = 4.

See Also

[.OPTION BYTOL](#)
[.OPTION DVDT](#)
[.OPTION RELV](#)

3: Options in HSPICE Netlists

.OPTION MCBRIEF

.OPTION MCBRIEF

Syntax

```
.OPTION MCBRIEF=x
```

Description

Controls how HSPICE outputs Monte Carlo parameters.

- MCBRIEF=0: Outputs all Monte Carlo parameters (default)
- MCBRIEF=1: Does not output the Monte Carlo parameters
- MCBRIEF=2: Outputs the Monte Carlo parameters into a *.lis* file only.
- MCBRIEF=3: Outputs the Monte Carlo parameters into the measure files only.

.OPTION MEASDGT

Syntax

```
.OPTION MEASDGT=x
```

Description

Formats the `.MEASURE` statement output in both the listing file and the `.MEASURE` output files (`.ma0`, `.mt0`, `.ms0`, and so on).

The value of `x` is typically between 1 and 7, although you can set it as high as 10. The default is 4.0.

For example, if `MEASDGT = 5`, then `.MEASURE` displays numbers as:

- Five decimal digits for numbers in scientific notation.
- Five digits to the right of the decimal for numbers between 0.1 and 999.

In the listing (`.lis`) file, all `.MEASURE` output values are in scientific notation, so `.OPTION MEASDGT=5` results in five decimal digits.

Use `MEASDGT` with `.OPTION INGOLD=x` to control the output data format.

See Also

[.OPTION INGOLD](#)
[.MEASURE](#)

.OPTION MEASFAIL

Syntax

```
.OPTION MEASFAIL=0 | 1
```

Description

You can assign this option the following values:

- `MEASFAIL=0`, outputs 0 into the `.mt#`, `.ms#`, or `.ma#` file, and prints failed to the listing file.
- `MEASFAIL=1` (default), prints failed into the `.mt#`, `.ms#`, or `.ma#` file, and into the listing file.

See Also

[.MEASURE](#)

.OPTION MEASFILE

Syntax

```
.OPTION MEASFILE=x
```

Description

Controls whether measure information outputs to single or multiple files when an `.ALTER` statement is present in the netlist. You can assign this option the following values:

- `MEASFILE=0`, outputs measure information to several files.
- `MEASFILE=1` (default), outputs measure information to a single file.

See Also

[.ALTER](#)
[.MEASURE](#)

.OPTION MEASSORT

Syntax

```
.OPTION MEASSORT=x
```

Description

In versions of HSPICE before 2003.09, to automatically sort large numbers of `.MEASURE` statements, you could use the `.OPTION MEASSORT` statement.

- `.OPTION MEASSORT=0` (default; did not sort `.MEASURE` statements).
- `.OPTION MEASSORT=1` (internally sorted `.MEASURE` statements).

You needed to set this option to 1 only if you used a large number of `.MEASURE` statements, where you needed to list similar variables together (to reduce simulation time). For a small number of `.MEASURE` statements, turning on internal sorting sometimes slowed-down simulation while sorting, compared to not sorting first.

Starting in version 2003.09, this option is obsolete. Now the measure performance is order independent, and HSPICE ignores this option.

See Also

[.MEASURE](#)

.OPTION MEASOUT

Syntax

```
.OPTION MEASOUT=x
```

Description

This option outputs `.MEASURE` statement values and sweep parameters into an ASCII file. Post-analysis processing (AvanWaves or other analysis tools) uses this `<design>.mt#` file, where `#` increments for each `.TEMP` or `.ALTER` block.

For example, for a parameter sweep of an output load, which measures the delay, the `.mt#` file contains data for a delay-versus-fanout plot. The default is 1. You can set this option to 0 (off) in the `hspice.ini` file.

See Also

- [.ALTER](#)
- [.MEASURE](#)
- [.TEMP](#)

3: Options in HSPICE Netlists

.OPTION MENTOR

.OPTION MENTOR

Syntax

```
.OPTION MENTOR=x
```

Description

MENTOR = 2 enables the Mentor MSPICE-compatible (ASCII) interface. This option requires a specific license.

.OPTION METHOD

Syntax

```
.OPTION METHOD=GEAR | TRAP
```

Description

Sets the numerical integration method for a transient analysis to either GEAR or TRAP.

- To use GEAR, set `METHOD = GEAR`, which sets `LVLTIM = 2`.
- To change `LVLTIM` from 2 to 1 or 3, set `LVLTIM = 1` or `3`, after the `METHOD = GEAR` option. This overrides `METHOD = GEAR`, which sets `LVLTIM = 2`.

TRAP (trapezoidal) integration usually reduces program execution time with more accurate results. However, this method can introduce an apparent oscillation on printed or plotted nodes, which might not result from circuit behavior. To test this, run a transient analysis by using a small timestep. If oscillation disappears, the cause was the trapezoidal method.

The GEAR method is a filter, removing oscillations that occur in the trapezoidal method. Highly non-linear circuits (such as operational amplifiers) can require very long execution times when you use the GEAR method.

Circuits that do not converge in trapezoidal integration, often converge if you use GEAR. Default is TRAP (trapezoidal).

Gear algorithm:

```
OPTION METHOD = GEAR
```

Backward-Euler:

```
OPTION METHOD = GEAR MU = 0
```

Trapezoidal algorithm (default):

```
OPTION METHOD = TRAP
```

See Also

[.OPTION LVLTIM](#)
[.OPTION MU](#)

.OPTION MODMONTE

Syntax

```
.OPTION MODMONTE=x
```

Description

If `MODMONTE=1`, then within a single simulation run, each device that shares the same model card and is in the same Monte Carlo index receives a different random value for parameters that have a Monte Carlo definition.

If `MODMONTE=0` (default), then within a single simulation run, each device that shares the same model card and is in the same Monte Carlo index, receives the same random value for its parameters that have a Monte Carlo definition.

.OPTION MODSRH

Syntax

```
.OPTION MODSRH=x
```

Example

```
example.sp:  
* modsrh used incorrectly  
.option post modsrh=1  
xil net8 b c t6  
xi0 a b net8 t6  
v1 a 0 pulse 3.3 0.0 10E-6 1E-9 1E-9  
+ 25E-6 50E-6  
v2 b 0 2  
v3 c 0 3  
.model nch nmos level=49 version=3.2  
.end
```

This input file automatically searches for t6.inc. If t6.inc includes the nch model, and you set MODSRH to 1, HSPICE or HSPICE RF does not load nch. Do not set MODSRH=1 in this type of file call. Use this option in front of the .MODEL card definition.

Description

If MODSRH=1, HSPICE or HSPICE RF does not load or reference a model described in a .MODEL statement, if the netlist does not use that model. This option shortens simulation run time when the netlist references many models, but no element in the netlist calls those models. The default is MODSRH=0. If MODSRH=1, then the read-in time increases slightly.

See Also

[.MODEL](#)

.OPTION MONTECON

Syntax

```
.OPTION MONTECON=x
```

Description

Continues a Monte Carlo analysis in HSPICE (not supported in HSPICE RF). Retrieves the next random value, even if non-convergence occurs. A random value can be too large or too small to cause convergence to fail. Other types of analysis can use this Monte Carlo random value.

.OPTION MU

Syntax

```
.OPTION MU=x
```

Description

This option defines the coefficient for trapezoidal integration. The value range is 0.0 to 0.5, and the default is 0.5.

3: Options in HSPICE Netlists

.OPTION NEWTOL

.OPTION NEWTOL

Syntax

```
.OPTION NEWTOL=x
```

Description

Calculates one or more iterations past convergence for every calculated DC solution and timepoint circuit solution. If you do not set `NEWTOL`, after HSPICE determines convergence, the convergence routine ends, and the next program step begins. The default is 0.

You can use `NEWTOL` in HSPICE, but not in HSPICE RF.

.OPTION NODE

Syntax

```
.OPTION NODE=x
```

Example

```
1 M1:B D2:+ Q4:B
```

This sample part of a cross reference line indicates that the bulk of M1, the anode of D2, and the base of Q4, all connect to node 1.

Description

Prints a node cross reference table. The `BRIEF` option suppresses `NODE`. The table lists each node and all elements connected to it. A code indicates the terminal of each element. A colon (:) separates the code from the element name.

The codes are:

- + Diode anode
- Diode cathode
- B BJT base
- B MOSFET or JFET bulk
- C BJT collector
- D MOSFET or JFET drain
- E BJT emitter
- G MOSFET or JFET gate
- S BJT substrate
- S MOSFET or JFET source

See Also

[.OPTION BRIEF](#)

.OPTION NOELCK

Syntax

```
.OPTION NOELCK
```

Description

No element check; bypasses element checking to reduce pre-processing time for very large files.

.OPTION NOISEMINFREQ

Syntax

```
.OPTION NOISEMINFREQ=x
```

Description

The `.OPTION NOISEMINFREQ` command option specifies the minimum frequency of noise analysis. The default is $1e-5$. If the frequency of noise analysis is smaller than the minimum frequency, HSPICE automatically sets the frequency for `NOISEMINFREQ` in noise analysis.

.OPTION NOMOD

Syntax

```
.OPTION NOMOD
```

Description

Suppresses the printout of model parameters.

.OPTION NOPAGE

Syntax

```
.OPTION NOPAGE
```

Description

Suppresses page ejects for title headings.

3: Options in HSPICE Netlists

.OPTION NOPIV

.OPTION NOPIV

Syntax

```
.OPTION NOPIV=x
```

Description

Prevents HSPICE or HSPICE RF from automatically switching to pivoting matrix factors, if a nodal conductance is less than `PIVTOL`. `NOPIV` inhibits pivoting.

See Also

[.OPTION PIVTOL](#)

.OPTION NOTOP

Syntax

```
.OPTION NOTOP
```

Description

Suppresses topology checks to increase the speed for pre-processing very large files.

.OPTION NOWARN

Syntax

```
.OPTION NOWARN
```

Description

Suppresses all warning messages, except those generated from statements in `.ALTER` blocks.

See Also

[.ALTER](#)

.OPTION NUMDGT

Syntax

```
.OPTION NUMDGT=x
```

Description

This option controls the listing printout (*.lis*) accuracy. The value of *x* is typically between 1 and 7, although you can set it as high as 10. The default is 4.0. This option does not affect the accuracy of the simulation.

This option does affect the results files (ASCII and binary) if you use the `.OPTION POST_VERSION = 2001` setting. The default setting of results files for printout accuracy is 5 digits.

See Also

[.OPTION POST_VERSION](#)

.OPTION NXX

Syntax

```
.OPTION NXX
```

Description

Stops printback of the data file, until HSPICE or HSPICE RF finds an `.OPTION BRIEF=0` or the `.END` statement. It also resets the `LIST`, `NODE` and `OPTS` options, and sets `NOMOD`. When `BRIEF=0`, it enables printback. `NXX` is the same as `BRIEF`.

See Also

- [.OPTION BRIEF](#)
- [.OPTION LIST](#)
- [.OPTION NODE](#)
- [.OPTION OPTS](#)

.OPTION OFF

Syntax

```
.OPTION OFF=x
```

Description

For all active devices, initializes terminal voltages to zero, if you did not initialize them to other values. For example, if you did not initialize both drain and source nodes of a transistor (using `.NODESET`, `.IC` statements, or connecting them to sources), then `OFF` initializes all nodes of the transistor to 0.

HSPICE or HSPICE RF checks the `OFF` option, before element `IC` parameters. If you assigned an element `IC` parameter to a node, simulation initializes the node to the element `IC` parameter value, even if the `OFF` option previously set it to 0.

You can use the `OFF` element parameter to initialize terminal voltages to 0 for specific active devices. Use the `OFF` option to help find exact DC operating-point solutions for large circuits.

See Also

[.DC](#)

[.IC](#)

[.NODESET](#)

3: Options in HSPICE Netlists

.OPTION OPFILE

.OPTION OPFILE

Syntax

```
.OPTION OPFILE=value
```

Description

The `OPFILE` option outputs the operating point information to a file. *value* can be 0 or 1.

- If *value* is 1, operating point information is output to a file named `<design>.dp#`.
- If *value* is 0, the operating point information outputs to stdout.

.OPTION OPTLST

Syntax

```
.OPTION OPTLIST=x
```

Description

Outputs additional optimization information:

- OPTLIST=0: No information (default).
- OPTLIST=1: Prints parameter, Broyden update, and bisection results information.
- OPTLIST=2: Prints gradient, error, Hessian, and iteration information.
- OPTLIST=3: Prints all of the above and Jacobian.

.OPTION OPTS

Syntax

`.OPTION OPTS`

Description

Prints the current settings for all control options. If you change any of the default values of the options, the `OPTS` option prints the values that the simulation actually uses. The `BRIEF` option suppresses `OPTS`.

See Also

[.OPTION BRIEF](#)

.OPTION PARHIER

Syntax

```
.OPTION PARHIER = < GLOBAL | LOCAL >
```

Example

```
.OPTION parhier=<global | local>
.PARAM DefPwid = 1u
.SUBCKT Inv a y DefPwid = 2u DefNwid = 1u
    Mpl <MosPinList> pMosMod L = 1.2u W = DefPwid
    Mnl <MosPinList> nMosMod L = 1.2u W = DefNwid
.ENDS
```

This example explicitly shows the difference between local and global scoping for using parameters in sub-circuits.

Description

Use the .OPTION OPTLST parameter to specify scoping rules.

The default setting is GLOBAL.

See Also

[.OPTION OPTLST](#)

.OPTION PATHNUM

Syntax

```
.OPTION PATHNUM
```

Description

Prints subcircuit path numbers, instead of path names.

.OPTION PIVOT

Syntax

```
.OPTION PIVOT=x
```

Description

Selects a pivot algorithms. Use these algorithms to reduce simulation time, and to achieve convergence in circuits that produce hard-to-solve matrix equations. To select the pivot algorithm, set `PIVOT` as follows:

- `PIVOT=0`: Original non-pivoting algorithm.
- `PIVOT=1`: Original pivoting algorithm.
- `PIVOT=2`: Picks the largest pivot in the row.
- `PIVOT=3`: Picks the best pivot in a row.
- `PIVOT=10` (default): Fast, non-pivoting algorithm; requires more memory.
- `PIVOT=11`: Fast, pivoting algorithm; requires more memory than `PIVOT` values less than 11.
- `PIVOT=12`: Picks the largest pivot in the row; requires more memory than `PIVOT` values less than 12.
- `PIVOT=13`: Fast, best pivot: faster; requires more memory than `PIVOT` values less than 13.

The fastest algorithm is `PIVOT = 13`, which can improve simulation time up to ten times, on very large circuits. However, `PIVOT = 13` requires substantially more memory for simulation.

Some circuits with large conductance ratios, such as switching regulator circuits, might require pivoting.

If `PIVTOL = 0`, HSPICE or HSPICE RF automatically changes from non-pivoting to a row-pivot strategy, if it detects any diagonal-matrix entry less than `PIVTOL`. This strategy provides the time and memory advantages of non-pivoting inversion, and avoids unstable simulations and incorrect results. Use `.OPTION NOPIV` to prevent HSPICE or HSPICE RF from pivoting. For very large circuits, `PIVOT = 10, 11, 12, or 13`, can require excessive memory.

If HSPICE or HSPICE RF switches to pivoting during a simulation, it displays the message followed by the node numbers that cause the problem:

```
pivot change on the fly
```

3: Options in HSPICE Netlists

.OPTION PIVOT

Use `.OPTION NODE` to cross-reference a node to an element. The `SPARSE` option is the same as `PIVOT`.

See Also

- [.OPTION NODE](#)
- [.OPTION NOPIV](#)
- [.OPTION PIVREF](#)
- [.OPTION PIVREL](#)
- [.OPTION PIVTOL](#)
- [.OPTION SPARSE](#)

.OPTION PIVREF

Syntax

```
.OPTION PIVREF=x
```

Description

Pivot reference. Use PIVREF in PIVOT = 11, 12, or 13 to limit the size of the matrix. The default is 1e+8.

See Also

[.OPTION PIVOT](#)

3: Options in HSPICE Netlists

.OPTION PIVREL

.OPTION PIVREL

Syntax

```
.OPTION PIVREL=x
```

Description

Sets the maximum and minimum ratio of a row or matrix. Use only if `PIVOT = 1`. Large values for `PIVREL` can result in very long matrix pivot times. If the value is too small, however, no pivoting occurs. Start with small values of `PIVREL` by using an adequate (but not excessive) value for convergence and accuracy. The default is `1E-20` (max = `1e-20`, min = `1`).

See Also

[.OPTION PIVOT](#)

.OPTION PIVTOL

Syntax

```
.OPTION PIVTOL=x
```

Description

Absolute minimum value for which HSPICE or HSPICE RF accepts a matrix entry as a pivot. If `PIVOT=0`, `PIVTOL` is the minimum conductance in the matrix. The default is `1.0e-15`.

`PIVTOL` must be less than `GMIN` or `GMINDC`. Values that approach 1 increase the pivot.

See Also

- [.OPTION GMIN](#)
- [.OPTION GMINDC](#)
- [.OPTION PIVOT](#)

.OPTION PLIM

Syntax

```
.OPTION PLIM
```

Description

Specifies plot size limits for current and voltage plots:

- Finds a common plot limit, and plots all variables on one graph, at the same scale
- Enables SPICE-type plots in HSPICE, which create a separate scale and axis for each plot variable

You can use SPICE-compatibility mode in HSPICE, but not in HSPICE RF.

This option does not affect postprocessing of graph data.

.OPTION POST

Syntax

```
.OPTION POST=[ 0 | 1 | 2 | 3 | ASCII | BINARY ]
```

Example

```
.OPTION POST=2
```

Description

Use an `.OPTION POST` statement to display high-resolution AvanWaves plots of simulation results, on either a graphics terminal or a high-resolution laser printer. Use `.OPTION POST` to provide output, without specifying other parameters. `POST` has defaults, which supply usable data to most parameters.

- `POST = 0`: Does not output simulation results.
- `POST = 1, BINARY`: (Default) Output format is binary.
- `POST = 2, ASCII`: Output format is ASCII.
- `POST = 3`: Output format is New Wave binary.

See Also

[.OPTION POST_VERSION](#)

3: Options in HSPICE Netlists

.OPTION POSTLVL

.OPTION POSTLVL

Syntax

```
.OPTION POSTLVL=x
```

Example

```
.OPTION POSTLVL=2
```

This example limits the data written to the waveform file to only the second-level nodes.

Description

The `.OPTION POSTLVL` option limits the data to only the x level nodes, which is written to your waveform file.

.OPTION POST_VERSION

Syntax

```
.OPTION POST_VERSION=x
```

Description

Sets the post-processing output version:

- `x = 9007` truncates the node name in the post-processor output file to a maximum of 16 characters.
- `x = 9601` (default) sets the node name length for the output file consistent with the input restrictions (1024 characters), and limits the number of output variables to 9999.
- `x = 2001` shows the new output file header, which includes the right number of output variables rather than **** when the number exceeds 9999. This option also changes the number of digits precision in results files to match the value of `.OPTION NUMDGT` (when `< 5`).

If you set `.OPTION POST_VERSION = 2001 POST= 2` in the netlist, then HSPICE or HSPICE RF returns more-accurate ASCII results.

```
.option post_version=2001
```

To use binary values (with double precision) in the output file, include the following in the input file:

```
*****  
.option post (or post=1) post_version=2001  
*****
```

For more accurate simulation results, comment this format.

See Also

[.OPTION NUMDGT](#)
[.OPTION POST](#)

.OPTION POSTTOP

Syntax

```
.OPTION POSTTOP=n
```

Example

```
POSTTOP = 1
```

This example limits the data written to the waveform file to only the top-level nodes.

Description

The `.OPTION POSTTOP` option limits the data to only the data from the top `n` level nodes, which is written to your waveform file. If you do not specify either the `.OPTION PROBE` or the `.OPTION POSTTOP` options, then HSPICE outputs all levels.

To enable the waveform display interface, you also need the `.OPTION POST` option.

See Also

[.OPTION POST](#)
[.OPTION PROBE](#)

.OPTION PROBE

Syntax

```
.OPTION PROBE=x
```

Description

Limits post-analysis output to only variables specified in `.PROBE`, `.PRINT`, `.PLOT`, and `.GRAPH` statements. HSPICE RF supports `.PROBE` and `.PRINT` statements, but does not support `.PLOT` and `.GRAPH` statements. By default, HSPICE or HSPICE RF outputs all voltages and power supply currents in addition to variables listed in `.PROBE`, `.PRINT`, `.PLOT`, and `.GRAPH` statements. `PROBE` significantly decreases the size of simulation output files.

See Also

- [.GRAPH](#)
- [.PLOT](#)
- [.PRINT](#)
- [.PROBE](#)

.OPTION PSF

Syntax

```
.OPTION PSF=x
```

Description

Specifies whether HSPICE or HSPICE RF outputs binary or ASCII data when you run an HSPICE simulation from Cadence Analog Artist.

The value of *x* can be 1 or 2.

- If *x* is 2, HSPICE or HSPICE RF produces ASCII output.
- If `.OPTION ARTIST PSF = 1`, HSPICE produces binary output.

See Also

[.OPTION ARTIST](#)

.OPTION PURETP

Syntax

```
.OPTION PURETP=x
```

Description

Integration method to use for reversal time point. The default is 0. If you set `PURETP=1`, then if HSPICE finds non-convergence, it uses TRAP (instead of B.E) for the reversed time point. Use this option with the `method=TRAP` statement to help some oscillating circuits to oscillate, if the default simulation process cannot satisfy the result.

.OPTION PUTMEAS

Syntax

```
.OPTION PUTMEAS=0 | 1
```

Description

The `.OPTION PUTMEAS` option controls the output variables, listed in the `.MEASURE` statement.

- 0: Does not save variable values, which are listed in the `.MEASURE` statement, into the corresponding output file (such as `.tr#`, `.ac#` or `.sw#`). This option decreases the size of the output file.
- 1: Default. Saves variable values, which are listed in the `.MEASURE` statement, into the corresponding output file (such as `.tr#`, `.ac#` or `.sw#`). This option is similar to the output of HSPICE 2000.4.

See Also

[.MEASURE](#)

.OPTION RELH

Syntax

```
.OPTION RELH=x
```

Description

Relative current tolerance, through voltage-defined branches (voltage sources and inductors). Use `RELH` to check current convergence, but only if the value of the `ABSH` control option is greater than zero. The default is 0.05.

You can use `RELH` in HSPICE, but not in HSPICE RF.

See Also

[.OPTION ABSH](#)

.OPTION RELI

Syntax

```
.OPTION RELI=x
```

Description

Sets the relative error/tolerance change, from iteration to iteration. This parameter determines convergence for all currents in diode, BJT, and JFET devices. (RELMOS sets tolerance for MOSFETs). This is the change in current, from the value calculated at the previous timepoint.

- Default = 0.01 for .OPTION KCLTEST = 0.
- Default = 1e-6 for .OPTION KCLTEST = 1.

See Also

[.OPTION RELMOS](#)
[.OPTION KCLTEST](#)

.OPTION RELMOS

Syntax

```
.OPTION RELMOS=x
```

Description

Sets the relative error tolerance (percent) for drain-to-source current, from iteration-to-iteration. This parameter determines convergence for currents in MOSFET devices. (.OPTION RELI sets the tolerance for other active devices.) Sets the change in current, from the value calculated at the previous timepoint. HSPICE or HSPICE RF uses the .OPTION RELMOS value, only if the current is greater than the .OPTION ABSMOS floor value. The default is 0.05.

See Also

- [.OPTION ABSMOS](#)
- [.OPTION RELI](#)
- [.OPTION RELMOS](#)

3: Options in HSPICE Netlists

.OPTION RELQ

.OPTION RELQ

Syntax

```
.OPTION RELQ=x
```

Description

Used in the timestep algorithm for local truncation error (LVLTIM = 2). RELQ changes the timestep size. If the capacitor charge calculation (in the present iteration) exceeds that of the past iteration by a percentage greater than the RELQ value, then HSPICE reduces the internal timestep (Delta). The default is 0.01.

You can use RELQ in HSPICE, but not in HSPICE RF.

See Also

[.OPTION LVLTIM](#)

.OPTION RELTOL

Syntax

```
.OPTION RELTOL=x
```

Description

Relative error tolerance for voltages. Use RELTOL with the ABSV control option to determine voltage convergence. Increasing RELTOL increases the relative error. RELTOL is the same as RELV. RELI and RELVDC options default to the RELTOL value. The default is $1e-3$.

You can use the RELTOL and RELV options in HSPICE, but not in HSPICE RF.

See Also

- [.OPTION ABSV](#)
- [.OPTION RELI](#)
- [.OPTION RELV](#)
- [.OPTION RELVDC](#)

3: Options in HSPICE Netlists

.OPTION RELV

.OPTION RELV

Syntax

```
.OPTION RELV=x
```

Description

Sets the relative error tolerance for voltages. If voltage or current exceeds the absolute tolerances, a RELV test determines convergence. Increasing RELV increases the relative error. You should generally maintain RELV at its default value. RELV conserves simulator charge. For voltages, RELV is the same as RELTOL. The default is $1e-3$.

See Also

[.OPTION RELTOL](#)

.OPTION RELVAR

Syntax

```
.OPTION RELVAR=x
```

Description

Use this option with ABSVAR, and the DVDT timestep algorithm. RELVAR sets the relative voltage change for LVLTIM = 1 or 3. If the node voltage at the current time point exceeds the node voltage at the previous time point by RELVAR, then HSPICE reduces the timestep, and calculates a new solution at a new time point. The default is 0.30 (30%).

For additional information, see section “DVDT Dynamic Timestep Algorithm” in the *HSPICE Simulation and Analysis User Guide*.

You can use the RELVAR option in HSPICE, but not in HSPICE RF.

See Also

- [.OPTION ABSVAR](#)
- [.OPTION DVDT](#)
- [.OPTION LVLTIM](#)

.OPTION RELVDC

Syntax

```
.OPTION RELVDC=x
```

Description

Sets the relative error tolerance for voltages. If voltages or currents exceed their absolute tolerances, the RELVDC test determines convergence. Increasing RELVDC increases the relative error. You should generally maintain RELVDC at its default value. RELVDC conserves simulator charge. Default is RELTOL (RELTOL default = $1e-3$).

See Also

[.OPTION RELTOL](#)

.OPTION RESMIN

Syntax

```
.OPTION RESMIN=x
```

Description

Minimum resistance for all resistors, including parasitic and inductive resistances. The default is $1e-5$ (ohm), and the range is $1e-15$ to 10 ohm.

3: Options in HSPICE Netlists

.OPTION RISETIME

.OPTION RISETIME

Syntax

.OPTION RISETIME=*x*

Description

Smallest risetime of a signal. Use this option only in transmission line models or HSPICE RF. In the U element, this equation determines the number of lumps:

$$\text{MIN}\left[20, 1 + \left(\frac{T_{\text{Def}}}{\text{RISETIME}}\right) \cdot 20\right]$$

TDef is the end-to-end delay in a transmission line. The W element uses RISETIME, only if Rs or Gd is non-zero. In such cases, RISETIME determines the maximum signal frequency.

.OPTION RMAX

Syntax

```
.OPTION RMAX=x
```

Description

Sets the TSTEP multiplier, which controls the maximum value (DELMAX) for the Delta of the internal timestep:

$$\text{DELMAX} = \text{TSTEP} \times \text{RMAX}$$

- The default is 5, if DVDT is 4 and LVLTIM is 1.
- Otherwise, the default is 2.

The maximum value is $1e+9$, the minimum value is $1e-9$. The recommended maximum value is $1e+5$. Supported in HSPICE and HSPICE RF.

For a discussion about timestep control, see section “Timestep Control for Accuracy” in the *HSPICE Simulation and Analysis User Guide*.

See Also

[.OPTION DELMAX](#)
[.OPTION DVDT](#)
[.OPTION LVLTIM](#)

.OPTION RMIN

Syntax

```
.OPTION RMIN=x
```

Description

Sets the minimum value of Delta (internal timestep). An internal timestep smaller than $RMIN \times TSTEP$, terminates the transient analysis, and reports an internal timestep too small error. If the circuit does not converge in $IMAX$ iterations, Delta decreases by the amount you set in the FT option. The default is $1.0e-9$.

You can use $RMIN$ in HSPICE, but not HSPICE RF.

See Also

[.OPTION FT](#)

[.OPTION IMAX](#)

.OPTION RUNLVL

Syntax

```
.OPTION RUNLVL=x
```

Description

The value for the `.OPTION RUNLVL` option controls the speed and accuracy trade-off. Higher values of `RUNLVL` result in higher accuracy and longer simulation times, while lower values give lower accuracy and faster simulation runtimes. The value of `RUNLVL` can be set to 0, 1, 2, 3, 4, 5, or 6.

The `RUNLVL` option setting controls the scaling of all simulator tolerances simulatenously and affects timestep control, convergence, and model bypass tolerances all at once. Higher values of `RUNLVL` result in smaller timestep sizes, and could result in more Newton-Raphson iterations in order to meet stricter error tolerances. The mode activated with `RUNLVL` affects only transient analysis.

When `RUNLVL` is set to

- 0, the algorithm turns off.
- 1, the simulation runs at the lowest simulation runtime.
- 3, is the default value.
- 5 or 6, corresponds to the HSPICE standard accurate mode. For most circuits, `RUNLVL = 5` is similar to HSPICE standard accurate mode. `RUNLVL = 6` has the highest accuracy.

If `.OPTION ACCURATE` is specified in the netlist together with `RUNLVL`, then the value of `RUNLVL` is limited to 5. In this case, specifying `RUNLVL` with a value smaller than 5 results in simulation running with `RUNLVL = 5`.

The `RUNLVL` option interacts with other options as follows:

1. The `RUNLVL` option, regardless of its position in the netlist, overrides the `LVLTIM` and `DVDT` timestep control mode options.
2. When `RUNLVL` is specified in the netlist, the default value of the `BYPASS` option is 1. Setting `BYPASS = 0` disables model bypass, regardless of the order in which `BYPASS` and `RUNLVL` are set.
3. If `.OPTION ACCURATE` is set, then the `RUNLVL` value is limited to 5, and the default value of `BYPASS` is set to 0. This behavior is independent of the order of the `RUNLVL`, `BYPASS`, and `ACCURATE` options.

3: Options in HSPICE Netlists

.OPTION RUNLVL

4. The `tstep` value specified with the `.TRAN` command affects timestep control when a `RUNLVL` option is used. Timestep values larger than `tstep*RMAX` use a tighter timestep control tolerance.

See Also

[.OPTION ACCURATE](#)
[.OPTION BYPASS](#)
[.OPTION DVDT](#)
[.OPTION LVLTIM](#)
[.OPTION RELTOL](#)
[.TRAN](#)

.OPTION SCALE

Syntax

```
.OPTION SCALE=x
```

Description

Element scaling factor in HSPICE or HSPICE RF. Scales parameters in element cards, by their value. The default is 1.

3: Options in HSPICE Netlists

.OPTION SCALM

.OPTION SCALM

Syntax

```
.OPTION SCALM=x
```

Description

Model scaling factor in HSPICE or HSPICE RF. Scales model parameters by their value. The default is 1. See the *HSPICE Elements and Device Models Manual* for parameters this option scales.

.OPTION SDA

Syntax

```
.OPTION SDA=x
```

Description

SDA = 2 produces a Cadence WSF (ASCII format) post-analysis file for Opus™. This option requires a specific license. The SDA is the same as the CDS option.

See Also

[.OPTION CDS](#)

.OPTION SEARCH

Syntax

```
.OPTION SEARCH = 'directory_path'
```

Example

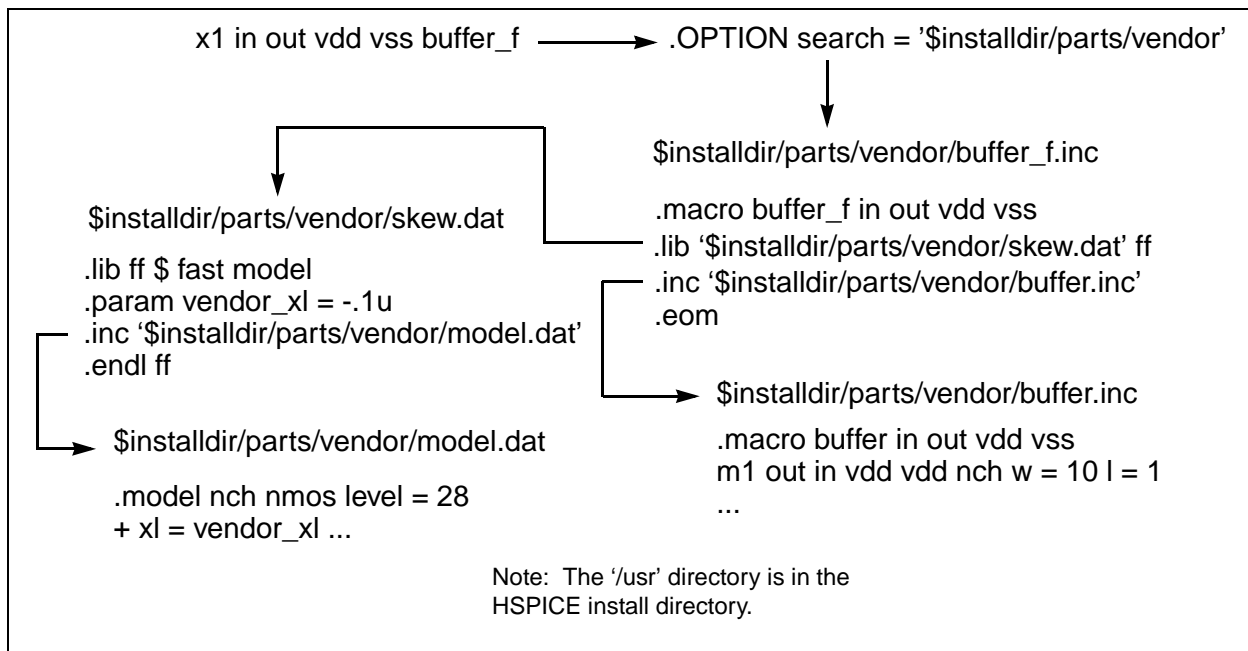
```
.OPTION SEARCH = '$installdir/parts/vendor'
```

Description

Use the `.OPTION SEARCH` statement to automatically access a library.

This example searches for models in the *vendor* subdirectory, under the `<$installdir>/parts` installation directory (see Figure 7). The *parts* directory contains the DDL subdirectories.

Figure 7 Vendor Library Usage



.OPTION SEED

Syntax

```
.OPTION SEED=x
```

Description

Starting seed for random-number generator in HSPICE Monte Carlo analysis (HSPICE RF does not support Monte Carlo analysis or the `.OPTION SEED` statement). The minimum value is 1; the maximum value is 259200.

.OPTION SLOPETOL

Syntax

```
.OPTION SLOPETOL=x
```

Description

Minimum value for breakpoint table entries in a piecewise linear (PWL) analysis. If the difference in the slopes of two consecutive PWL segments is less than the `SLOPETOL` value, HSPICE or HSPICE RF ignores the breakpoint for the point between the segments. The default is 0.75.

.OPTION SPARSE

Syntax

```
.OPTION SPARSE=x
```

Description

The `SPARSE` option is the same as `PIVOT`.

See Also

[.OPTION PIVOT](#)

3: Options in HSPICE Netlists

.OPTION SPICE

.OPTION SPICE

Syntax

```
.OPTION SPICE=x
```

Example 1

Example of general parameters, used with .OPTION SPICE:

```
TNOM = 27 DEFNRD = 1 DEFNRS = 1 INGOLD = 2
ACOUT = 0 DC
PIVOT PIVTOL = 1E-13 PIVREL = 1E-3 RELTOL = 1E-3
ITL1 = 100
ABSMOS = 1E-6 RELMOS = 1E-3 ABSTOL = 1E-12
VNTOL = 1E-6
ABSVDVDC = 1E-6 RELVDC = 1E-3 RELI = 1E-3
```

Example 2

Example of transient parameters, used with .OPTION SPICE:

```
DCAP = 1 RELQ = 1E-3 CHGTOL=1E-14 ITL3 = 4 ITL4 = 10
ITL5 = 5000 FS = 0.125 FT = 0.125
```

Example 3

Example of model parameters, used with .OPTION SPICE:

```
For BJT: MJS = 0
For MOSFET, CAPOP = 0
LD = 0 if not user-specified UTRA = 0 not used by SPICE for
LEVEL = 2 NSUB must be specified NLEV = 0 for SPICE noise
equation
```

Description

Makes HSPICE compatible with Berkeley SPICE. HSPICE RF is C-based and not Fortran-based so it is not compatible with Berkeley SPICE. You can use .OPTION SPICE in HSPICE, but not in HSPICE RF. If you set this option, HSPICE uses the options and model parameters explained in the examples.

.OPTION SPMODEL

Syntax

```
.OPTION SPMODEL [= name]
```

Example 1

```
.option spmodel
```

This example disables the previous `.OPTION VAMODEL`, but has no effect on the other `VAMODEL` options if they are specified for the individual cells. For example, if `.OPTION VAMODEL = vco` has been set, the `vco` cell uses the Verilog-A definition whenever it is available until `.OPTION SPMODEL = vco` disables it.

Example 2

```
.option spmodel=chargepump
```

This example disables the previous `.OPTION VAMODEL = chargepump`, which causes all instantiations of `chargepump` to now use the subcircuit definition again.

Description

This option is for use in HSPICE with Verilog-A only. In this option, the name is the cell name that uses a SPICE definition. Each `SPMODEL` option can take no more than one name. Multiple names need multiple `SPMODEL` options.

.OPTION STATFL

Syntax

`.OPTION STATFL=x`

Description

Controls whether HSPICE creates a *.st0* file.

- `STATFL = 0` (default) outputs a *.st0* file.
- `STATFL = 1` suppresses the *.st0* file.

.OPTION SYMB

Syntax

```
.OPTION SYMB=x
```

Description

If you set the `SYMB` option to 1, HSPICE operates with a symbolic operating point algorithm to get initial guesses before calculating operating points. The default is 0.

.OPTION TIMERES

Syntax

```
.OPTION TIMERES=x
```

Description

Minimum separation between breakpoint values for the breakpoint table. If two breakpoints are closer together (in time) than the `TIMERES` value, HSPICE enters only one of them in the breakpoint table. The default is 1 ps.

You can use `TIMERES` in HSPICE, but not in HSPICE RF.

.OPTION TNOM

Syntax

```
.OPTION TNOM=x
```

Description

Reference temperature for HSPICE or HSPICE RF simulation. At this temperature, component derating is zero. The default is 25 °C. If you enable `.OPTION SPICE` (HSPICE only; HSPICE RF does not support this option), the default is 27 °C.

Note:

The reference temperature defaults to the analysis temperature if you do not explicitly specify a reference temperature.

See Also

[.OPTION SPICE](#)
[.TEMP](#)

.OPTION TRCON

Syntax

```
.OPTION TRCON=x
```

Description

Controls the speed of some special circuits. For some large non-linear circuits with large TSTOP/TSTEP values, analysis might run for an excessively long time. In this case, HSPICE might automatically set a new and bigger RMAX value to speed up the analysis for primary reference. In most cases, however, HSPICE does not activate this type of autospeedup process.

For autospeedup to occur, all three of the following conditions must occur:

- N1 (Number of Nodes) > 1,000
- N2 (TSTOP/TSTEP) >= 10,000
- N3 (Total Number of Diode, BJTs, JFETs and MOSFETs) > 300

Autospeedup is most likely to occur if the circuit also meets either of the following conditions:

- N2 >= 1e+8, and N3 > 500, or
- N2 >= 2e+5, and N3 > 1e+4
- TRCON = 3: enable auto-speedup only. HSPICE invokes auto-speed up if:
 - there are more than 1000 nodes, or
 - there are more than 300 active devices, or
 - Tstop/Tstep (as defined in .TRAN) > 1e8.

When auto-speedup is active, RMAX increases, and HSPICE can take larger timesteps.

- TRCON = 2: enables auto-convergence only.
 - HSPICE invokes auto-convergence if you use the default integration method (trapezoidal), and if HSPICE fails to converge, an “internal timestep too small” error is issued.
 - Auto-convergence sets METHOD = gear, LVLTIM = 2, and starts the transient simulation again from time=0.
- TRCON = 1: enables both auto-convergence and auto-speedup.

- TRCON = 0: disables both auto-convergence and auto-speedup (default).
- TRCON = -1: same as TRCON = 0.

TRCON also controls the automatic convergence process (autoconvergence) as well as the automatic speedup (autospeedup) processes in HSPICE. You cannot use TRCON in HSPICE RF. HSPICE also uses autoconvergence in DC analysis, if the Newton-Raphson (N-R) method fails to converge.

If the circuit fails to converge using the trapezoidal (TRAP) numerical integration method (for example, because of trapezoidal oscillation), HSPICE uses the GEAR method and LTE timestep algorithm to run the transient analysis again from `time=0`. This process is called autoconvergence.

Autoconvergence sets options to their default values before the second try:

```
METHOD=GEAR, LVLTIM=2, MBYPASS=1.0,  
+ BYPASS=0.0, SLOPETOL=0.5,  
+ BYTOL= min{mbypas*vntol and reltol}
```

RMAX = 2.0 if it was 5.0 in the first run; otherwise RMAX does not change.

See Also

- [.OPTION BYPASS](#)
- [.OPTION BYTOL](#)
- [.OPTION MBYPASS](#)
- [.OPTION RMAX](#)
- [.OPTION SLOPETOL](#)

.OPTION TRTOL

Syntax

```
.OPTION TRTOL=x
```

Description

Used in the timestep algorithm for local truncation error (LVLTIM = 2). HSPICE multiplies TRTOL by the internal timestep, which the timestep algorithm for the local truncation error generates. TRTOL reduces simulation time, and maintains accuracy. It estimates the amount of error introduced when the algorithm truncates the Taylor series expansion. This error reflects the minimum time-step to reduce simulation time and maintain accuracy. The range of TRTOL is 0.01 to 100; typical values are 1 to 10. If you set TRTOL to 1 (the minimum value), HSPICE uses a very small timestep. As you increase the TRTOL setting, the timestep size increases. The default is 7.0.

You can use TRTOL in HSPICE, but not HSPICE RF.

See Also

[.OPTION LVLTIM](#)

.OPTION UNWRAP

Syntax

```
.OPTION UNWRAP
```

Description

Displays phase results for AC analysis in unwrapped form (with a continuous phase plot).HSPICE uses these results to accurately calculate group delay (HSPICE RF does not support group time delays in AC analysis output). It also uses unwrapped phase results to compute group delay, even if you do not set UNWRAP.

.OPTION VAMODEL

Syntax

```
.OPTION VAMODEL [=name]
```

Example 1

```
.option vamodel=vco
```

This example specifies a Verilog-A definition for all instantiations of the cell `vco`.

Example 2

```
.option vamodel=vco vamodel=chargepump
```

This example specifies a Verilog-A definition for all instantiations of the `vco` and `chargepump` cells.

Example 3

```
.option vamodel
```

This example instructs HSPICE to always use the Verilog-A definition whenever it is available.

Description

This option is for use in HSPICE with Verilog-A only. This option specifies that the *name* is the cell name that uses a Verilog-A definition rather than the subcircuit definition when both exist. Each `VAMODEL` option can take no more than one name. Multiple names need multiple `VAMODEL` options.

If a name is not provided for the `VAMODEL` option, HSPICE uses the Verilog-A definition whenever it is available. The `VAMODEL` option works on cell-based instances only. Instance-based overriding is not allowed.

.OPTION VERIFY

Syntax

```
.OPTION VERIFY=x
```

Description

This option is an alias for `.OPTION LIST`.

See Also

[.OPTION LIST](#)

3: Options in HSPICE Netlists

.OPTION VFLOOR

.OPTION VFLOOR

Syntax

```
.OPTION VFLOOR=x
```

Description

Minimum voltage to print in output listing. All voltages lower than VFLOOR, print as 0. Affects only the output listing: VNTOL (ABSV) sets minimum voltage to use in a simulation.

See Also

[.OPTION ABSV](#)

[.OPTION VNTOL](#)

.OPTION VNTOL

Syntax

```
.OPTION VNTOL=x
```

Description

The VNTOL option is the same as the ABSV option.

See Also

[.OPTION ABSV](#)

.OPTION WACC

Syntax

```
.OPTION WACC=x
```

Description

This option is used to activate the dynamic step control algorithm for a W element transient analysis. `WACC` is a non-negative real value, which can be set between 0.0 and 10.0.

When `WACC` is positive, the dynamic step control algorithm is activated. Larger values result in higher performance with lower accuracy, while smaller values result in lower performance with better accuracy.

Use `WACC = 1.0` for normal simulation and `WACC = 0.1` for an accurate simulation. When `WACC = 0.0`, the original step control method is used with predetermined static breakpoints. Currently the default value is 0.0. For HSPICE RF, the default value for `WACC` is 0.5. If `WACC` is set as 0.0, no control is added.

.OPTION WNFLAG

Syntax

```
.OPTION WNFLAG=[ 0 | 1 ]
```

Description

This option only applies to BSIM4 models. You use this option to select a bin model.

When an `.OPTION WNFLAG` instance parameter

- is not specified, the bin model specified by this option is used.
- is specified, its value is used.

Use `WNFLAG=1` (default) to select the bin model based on `W` (BSIM4 MOSFET channel width) per `NF` (number of device fingers) parameters.

Use `WNFLAG=0` to select the bin model based on total `W`.

.OPTION WARNLIMIT

Syntax

```
.OPTION WARNLIMIT=x
```

Description

Limits how many times certain warnings appear in the output listing. This reduces the output listing file size. *x* is the maximum number of warnings for each warning type. This limit applies to the following warning messages:

- MOSFET has negative conductance.
- Node conductance is zero.
- Saturation current is too small.
- Inductance or capacitance is too large.

The default is 1.

.OPTION WL

Syntax

```
.OPTION WL=x
```

Description

Reverses the order of the MOS element `VSIZE`. Default order is length-width; changes the order to width-length. The default is 0.

.OPTION XDTEMP

Syntax

```
.OPTION XDTEMP=value
```

Example

```
.OPTION XDTEMP
X1 2 0 SUB1 DTEMP=2
.SUBCKT SUB1 A B
R1 A B 1K DTEMP=3
C1 A B 1P
X2 A B sub2 DTEMP=4
.ENDS
.SUBCKT SUB2 A B
R2 A B 1K
.ENDS
```

In this example:

- X1 sets a temperature difference (2 degrees Celsius) between the elements within the subcircuit SUB1.
- X2 (a subcircuit instance of X1) sets a temperature difference by the DTEMP value of both X1 and X2 (2+4=6 degrees Celsius) between the elements within the SUB2 subcircuit. Finally, the DTEMP value of each element in this example is:

```
Elements DTEMP Value (Celsius)
X1 2
X1.R1 2+3 =5
X1.C1 2
X2 2+4=6
X2.R2 6
```

Description

The .OPTION XDTEMP statement defines how HSPICE interprets the DTEMP parameter, where value is either:

- 0 (the default), indicating a user-defined parameter, or
- 1 indicates a temperature difference parameter.

If you set .OPTION XDTEMP to 1, HSPICE adds the DTEMP value in the subcircuit call statement to all elements within the subcircuit, that use the DTEMP keyword syntax.

The DTEMP parameter is cumulative throughout the design hierarchy.

.OPTION ZUKEN

Syntax

```
.OPTION ZUKEN=x
```

Description

This option enables or disables the Zuken interface.

- If x is 2, enables the Zuken interactive interface.
- If x is 1 (default), disables this interface.

3: Options in HSPICE Netlists

.OPTION ZUKEN

3: Options in HSPICE Netlists
.OPTION ZUKEN

3: Options in HSPICE Netlists
.OPTION ZUKEN

3: Options in HSPICE Netlists

.OPTION ZUKEN

3: Options in HSPICE Netlists

.OPTION ZUKEN

3: Options in HSPICE Netlists
.OPTION ZUKEN

3: Options in HSPICE Netlists

.OPTION ZUKEN

4

Commands in Digital Vector Files

Contains an alphabetical listing of the commands you can use in a digital vector file.

You can use the following commands in a digital vector file.

ENABLE	TDELAY	VIL
IO	TFALL	VNAME
ODELAY	TRISE	VOH
OUT or OUTZ	TRIZ	VOL
PERIOD	TSKIP	VREF
RADIX	TUNIT	VTH
SLOPE	VIH	

ENABLE**Syntax**

```
ENABLE controlling_signalname mask
```

Argument	Definition
<i>controlling_signalname</i>	Controlling signal for bidirectional signals. Must be an input signal with a radix of 1. The bidirectional signals become output when the controlling signal is at state 1 (or high). To reverse this default control logic, start the control signal name with a tilde (~).
<i>mask</i>	Defines the bidirectional signals to which <code>ENABLE</code> applies.

Example

```
radix 144
io ibb
vname a x[[3:0]] y[[3:0]]
enable a 0 F 0
enable ~a 0 0 F
```

In this example, the *x* and *y* signals are bidirectional as defined by the *b* in the *io* line.

- The first enable statement indicates that *x* (as defined by the position of *F*) becomes output when the *a* signal is 1.
- The second enable specifies that the *y* bidirectional bus becomes output when the *a* signal is 0.

Description

The `ENABLE` statement specifies the controlling signal(s) for bidirectional signals. All bidirectional signals require an `ENABLE` statement. If you specify more than one `ENABLE` statement, the last statement overrules the previous statement, and HSPICE or HSPICE RF issues a warning message:

```
[Warning]:[line 6] resetting enable signal to WENB for bit 'XYZ'
```

IDELAY

Syntax

IDELAY *delay_value mask*

Argument	Definition
<i>delay_value</i>	Time delay to apply to the signals.
mask	Signals to which the delay applies. If you do not provide a mask, the delay value applies to all signals.

Example

```
RADIX 1 1 4 1234 11111111
IO i i o iib iiiiii
VNAME V1 V2 VX[[3:0]] V4 V5[[1:0]] V6[[0:2]] V7[[0:3]]
+ V8 V9 V10 V11 V12 V13 V14 V15
TDELAY 1.0
TDELAY -1.2 0 1 F 0000 00000000
TDELAY 1.5 0 0 0 1370 00000000
IDELAY 2.0 0 0 0 000F 00000000
ODELAY 3.0 0 0 0 000F 00000000
```

This example does not specify the TUNIT statement so HSPICE or HSPICE RF uses the default, ns, as the time unit for this example. The first TDELAY statement indicates that all signals have the same delay time of 1.0ns. Subsequent TDELAY, IDELAY, or ODELAY statements overrule the delay time of some signals.

- The delay time for the V2 and Vx signals is -1.2.
- The delay time for the V4, V5[0:1], and V6[0:2] signals is 1.5.
- The input delay time for the V7[0:3] signals is 2.0, and the output delay time is 3.0.

Description

Defines an input delay time for bidirectional signals, relative to the absolute time of each row in the Tabular Data section.

HSPICE or HSPICE RF ignores IDELAY settings on output signals, and issues a warning message.

You can specify more than one TDELAY, IDELAY, or ODELAY statement.

4: Commands in Digital Vector Files

IDELAY

- If you apply more than one TDELAY (IDELAY, ODELAY) statement to a signal, the last statement overrides the previous statements, and HSPICE or HSPICE RF issues a warning.
- If you do not specify the signal delays in a TDELAY, IDELAY, or ODELAY statement, HSPICE or HSPICE RF defaults to zero.

See Also

[ODELAY](#)

[TDELAY](#)

[TUNIT](#)

IO

Syntax

```
IO I | O | B | U      [I | O | B | U ...]
```

Argument	Definition
i	Input, which HSPICE or HSPICE RF uses to stimulate the circuit.
o	Expected output, which HSPICE or HSPICE RF compares with the simulated outputs.
b	Bidirectional vector.
u	Unused vector, which HSPICE or HSPICE RF ignores.

Example

```
io i i i bbbb iiiioouu
```

Description

The IO statement defines the type for each vector. The line starts with the IO keyword followed by a string of i, b, o, or u definitions. These definitions indicate whether each corresponding vector is an input (i), bidirectional (b), output (o), or unused (u) vector.

- If you do not specify the IO statement, HSPICE or HSPICE RF assumes that all signals are input signals.
- If you define more than one IO statement, the last statement overrules previous statements.

ODELAY

Syntax

```
ODELAY delay_value mask
```

Argument	Definition
<i>delay_value</i>	Time delay to apply to the signals.
mask	Signals to which the delay applies. If you do not provide a mask, the delay value applies to all signals.

Example

```

RADIX 1 1 4 1234 11111111
IO i i o iiib iiiiiii
VNAME V1 V2 VX[[3:0]] V4 V5[[1:0]] V6[[0:2]] V7[[0:3]]
+ V8 V9 V10 V11 V12 V13 V14 V15
TDELAY 1.0
TDELAY -1.2 0 1 F 0000 00000000
TDELAY 1.5 0 0 0 1370 00000000
IDELAY 2.0 0 0 0 000F 00000000
ODELAY 3.0 0 0 0 000F 00000000

```

This example does not specify the `TUNIT` statement so HSPICE or HSPICE RF uses the default, ns, as the time unit for this example. The first `TDELAY` statement indicates that all signals have the same delay time of 1.0ns. Subsequent `TDELAY`, `IDELAY`, or `ODELAY` statements overrule the delay time of some signals.

- The delay time for the V2 and Vx signals is -1.2.
- The delay time for the V4, V5[0:1], and V6[0:2] signals is 1.5.
- The input delay time for the V7[0:3] signals is 2.0, and the output delay time is 3.0.

Description

Defines an output delay time for bidirectional signals relative to the absolute time of each row in the Tabular Data section.

HSPICE or HSPICE RF ignores `ODELAY` settings on input signals and issues a warning message.

You can specify more than one `TDELAY`, `IDELAY`, or `ODELAY` statement.

- If you apply more than one TDELAY (IDELAY, ODELAY) statement to a signal, the last statement overrides the previous statements, and HSPICE or HSPICE RF issues a warning.
- If you do not specify the signal delays in a TDELAY, IDELAY, or ODELAY statement, HSPICE or HSPICE RF defaults to zero.

See Also

[IDELAY](#)
[TDELAY](#)
[TUNIT](#)

4: Commands in Digital Vector Files

OUT or OUTZ

OUT or OUTZ

Syntax

```
OUT <output_resistance> mask
```

Argument	Definition
<output_resistance>	Output resistance for an input signal. Default=0.
mask	Signals to which the output resistance applies. If you do not provide a mask, the output resistance value applies to all input signals.

Example

```
OUT 15.1
OUT 150 1 1 1 0000 00000000
OUTZ 50.5 0 0 0 137F 00000000
```

The first `OUT` statement in this example creates a 15.1 ohm resistor to place in series with all vector inputs. The next `OUT` statement sets the resistance to 150 ohms for vectors 1 to 3. The `OUTZ` statement changes the resistance to 50.5 ohms for vectors 4 through 7.

Description

The `OUT` and `OUTZ` keywords are equivalent, and specify output resistance for each signal (for which the mask applies); `OUT` (or `OUTZ`) applies only to input signals.

- If you do not specify the output resistance of a signal in an `OUT` (or `OUTZ`) statement, HSPICE or HSPICE RF uses the default (zero).
- If you specify more than one `OUT` (or `OUTZ`) statement for a signal, the last statement overrules the previous statements, and HSPICE or HSPICE RF issues a warning message.

The `OUT` (or `OUTZ`) statements have no effect on the expected output signals.

PERIOD

Syntax

```
PERIOD time_interval
```

Argument	Definition
<code>time_interval</code>	Time interval for the Tabular Data.

Example

```
radix 1111 1111  
period 10  
1000 1000  
1100 1100  
1010 1001
```

- The first row of the tabular data (1000 1000) is at time 0ns.
- The second row (1100 1100) is at 10ns.
- The third row (1010 1001) is at 20ns.

Description

The `PERIOD` statement defines the time interval for the Tabular Data section. You do not need to specify the absolute time at every time point. If you use a `PERIOD` statement without the `TSKIP` statement, the Tabular Data section contains only signal values, not absolute times. The `TUNIT` statement defines the time unit of the `PERIOD`.

4: Commands in Digital Vector Files

RADIX

RADIX

Syntax

```
RADIX <number_of_bits> [<number_of_bits>...]
```

Argument	Definition
<number_of_bits>	Specifies the number of bits in one vector in the digital vector file. You must include a separate <number_of_bits> argument in the RADIX statement for each vector listed in the file.

Example

```
; start of Vector Pattern Definition section
RADIX 1 1 4 1234 1111 1111
VNAME A B C[[3:0]] I9 I[[8:7]] I[[6:4]] I[[3:0]] 07 06 05 04
+ 03 02 01 00
IO I I I IIII 0000 0000
```

This example illustrates two 1-bit signals followed by a 4-bit signal, followed by one each 1-bit, 2-bit, 3-bit, and 4-bit signals, and finally eight 1-bit signals.

Description

The RADIX statement specifies the number of bits associated with each vector. Valid values for the number of bits range from 1 to 4.

Table 1 Valid Values for the RADIX Statement

# bits	Radix	Number System	Valid Digits
1	2	Binary	0, 1
2	4	–	0 – 3
3	8	Octal	0 – 7
4	16	Hexadecimal	0 – F

A digital vector file must contain only one RADIX command, and it must be the first non-comment line in the file.

SLOPE

Syntax

```
SLOPE [<input_rise_time> | <input_fall_time>] mask
```

Argument	Definition
<input_rise_time>	Rise time of the input signal.
<input_fall_time>	Fall time of the input signal.
mask	Name of a signal to which the SLOPE statement applies. If you do not specify a mask, the SLOPE statement applies to all signals.

Example 1

```
SLOPE 1.2
```

In this example, the rising and falling times of all signals are 1.2 ns.

Example 2

```
SLOPE 1.1 1100 0110
```

In this example, the rising/falling time is 1.1 ns for the first, second, sixth, and seventh signals.

Description

The SLOPE statement specifies the rise/fall time for the input signal. Use the TUNIT statement to define the time unit for this statement.

- If you do not specify the SLOPE statement, the default slope value is 0.1 ns.
- If you specify more than one SLOPE statement, the last statement overrules the previous statements, and HSPICE or HSPICE RF issues a warning message.

The SLOPE statement has no effect on the expected output signals. You can specify the optional TRISE and TFALL statements to overrule the rise time and fall time of a signal.

TDELAY**Syntax**

```
TDELAY delay_value mask
```

Argument	Definition
<i>delay_value</i>	Time delay to apply to the signals.
<i>mask</i>	Signals to which the delay applies. If you do not provide a mask, the delay value applies to all signals.

Example

```
RADIX 1 1 4 1234 11111111
IO i i o iib iiiiii
VNAME V1 V2 VX[[3:0]] V4 V5[[1:0]] V6[[0:2]] V7[[0:3]]
+ V8 V9 V10 V11 V12 V13 V14 V15
TDELAY 1.0
TDELAY -1.2 0 1 F 0000 00000000
TDELAY 1.5 0 0 0 1370 00000000
IDELAY 2.0 0 0 0 000F 00000000
ODELAY 3.0 0 0 0 000F 00000000
```

This example does not specify the `TUNIT` statement so HSPICE or HSPICE RF uses the default, ns, as the time unit for this example. The first `TDELAY` statement indicates that all signals have the same delay time of 1.0ns. Subsequent `TDELAY`, `IDELAY`, or `ODELAY` statements overrule the delay time of some signals.

- The delay time for the V2 and Vx signals is -1.2.
- The delay time for the V4, V5[0:1], and V6[0:2] signals is 1.5.
- The input delay time for the V7[0:3] signals is 2.0, and the output delay time is 3.0.

Description

Defines the delay time of both input and output signals relative to the absolute time of each row in the Tabular Data section.

You can specify more than one `TDELAY`, `IDELAY`, or `ODELAY` statement.

- If you apply more than one TDELAY (IDELAY, ODELAY) statement to a signal, the last statement overrides the previous statements, and HSPICE or HSPICE RF issues a warning.
- If you do not specify the signal delays in a TDELAY, IDELAY, or ODELAY statement, HSPICE or HSPICE RF defaults to zero.

See Also

IDELAY
ODELAY
TUNIT

TFALL

Syntax

```
TFALL <input_fall_time> mask
```

Argument	Definition
<input_fall_time>	Fall time of the input signal.
mask	Name of a signal to which the TFALL statement applies. If you do not specify a mask, the TFALL statement applies to all input signals.

Example

In the following example, the TFALL statement assigns a fall time of 0.5 time units to all vectors.

```
TFALL 0.5
```

In the following example, the TFALL statement assigns a fall time of 0.3 time units, overriding the older setting of 0.5 to vectors 2, 3, and 4 to 7.

```
TFALL 0.3 0 1 1 137F 00000000
```

In the following example, the TFALL statement assigns a fall time of 0.9 time units to vectors 8 through 11.

```
TFALL 0.9 0 0 0 0000 11110000
```

Description

The TFALL statement specifies the fall time of each input signal for which the mask applies. The TUNIT statement defines the time unit of TFALL.

- If you do not use any TFALL statement to specify the fall time of the signals, HSPICE or HSPICE RF uses the value defined in the *slope* statement.
- If you apply more than one TFALL statement to a signal, the last statement overrides the previous statements, and HSPICE or HSPICE RF issues a warning message.

TFALL statements have no effect on the expected output signals.

TRISE

Syntax

```
TRISE <input_rise_time> mask
```

Argument	Definition
<input_rise_time>	Rise time of the input signal.
mask	Name of a signal to which the TRISE statement applies. If you do not specify a mask, the TRISE statement applies to all input signals.

Example 1

```
TRISE 0.3
```

In this example, the TRISE statement assigns a rise time of 0.3 time units to all vectors.

Example 2

```
TRISE 0.5 0 1 1 137F 00000000
```

In this example, the TRISE statement assigns a rise time of 0.5 time units, overriding the older setting of 0.3 in at least some of the bits in vectors 2, 3, and 4 through 7.

Example 3

```
TRISE 0.8 0 0 0 0000 11110000
```

In this example, the TRISE statement assigns a rise time of 0.8 time units to vectors 8 through 11.

Description

The TRISE statement specifies the rise time of each input signal for which the mask applies. The TUNIT statement defines the time unit of TRISE.

- If you do not use any TRISE statement to specify the rising time of the signals, HSPICE or HSPICE RF uses the value defined in the *slope* statement.

4: Commands in Digital Vector Files

TRISE

- If you apply more than one `TRISE` statement to a signal, the last statement overrides the previous statements, and HSPICE or HSPICE RF issues a warning message.

`TRISE` statements have no effect on the expected output signals.

TRIZ

Syntax

```
TRIZ <output_impedance>
```

Argument	Definition
<output_impedance>	Output impedance of the input signal.
mask	Name of a signal to which the TRIZ statement applies. If you do not specify a mask, the TRIZ statement applies to all input signals.

Example

```
TRIZ 15.1Meg  
TRIZ 150Meg 1 1 1 0000 00000000  
TRIZ 50.5Meg 0 0 0 137F 00000000
```

- The first TRIZ statement sets the high impedance resistance globally, at 15.1 Mohms.
- The second TRIZ statement increases the value to 150 Mohms, for vectors 1 to 3.
- The last TRIZ statement increases the value to 50.5 Mohms, for vectors 4 through 7.

Description

The TRIZ statement specifies the output impedance, when the signal (for which the mask applies) is in *tristate*; TRIZ applies only to the input signals.

- If you do not specify the *tristate* impedance of a signal, in a TRIZ statement, HSPICE or HSPICE RF assumes 1000M.
- If you apply more than one TRIZ statement to a signal, the last statement overrules the previous statements, and HSPICE or HSPICE RF issues a warning.

TRIZ statements have no effect on the expected output signals.

TSKIP**Syntax**

```
TSKIP <absolute_time> <tabular_data> ...
```

Argument	Definition
<absolute_time>	Absolute time.
<tabular_data>	Data captured at <absolute_time>.

Example

```
radix 1111 1111
period 10
tskip
11.0 1000 1000
20.0 1100 1100
33.0 1010 1001
```

HSPICE or HSPICE RF ignores the absolute times 11.0, 20.0 and 33.0, but HSPICE does process the tabular data on the same lines as those absolute times.

Description

The `TSKIP` statement specifies to ignore the absolute time field in the tabular data. You can then keep, but ignore, the absolute time field of each row in the tabular data, when you use the `PERIOD` statement.

You might do this, for example, if the absolute times are not perfectly periodic for testing reasons. Another reason might be that a path in the circuit does not meet timing, but you might still use it as part of a test bench. Initially, HSPICE or HSPICE RF writes to the vector file, using absolute time. After you fix the circuit, you might want to use periodic data.

TUNIT

```
TUNIT [ fs | ps | ns | us | ms ]
```

Argument	Definition
fs	femtosecond
ps	picosecond
ns	nanosecond (this is the default)
us	microsecond
ms	millisecond

Example

```
TUNIT ns
11.0 1000 1000
20.0 1100 1100
33.0 1010 1001
```

The `TUNIT` statement in this example specifies that the absolute times in the Tabular Data section are 11.0ns, 20.0ns, and 33.0ns.

Description

The `TUNIT` statement defines the time unit in the digital vector file, for `PERIOD`, `TDELAY`, `IDELAY`, `ODELAY`, `SLOPE`, `TRISE`, `TFALL`, and absolute time.

- If you do not specify the `TUNIT` statement, the default time unit value is ns.
- If you define more than one `TUNIT` statement, the last statement overrules the previous statement.

See Also

[IDELAY](#)
[ODELAY](#)
[TDELAY](#)

VIH

Syntax

```
VIH <logic-high_voltage> mask
```

Argument	Definition
<logic-high_voltage>	Logic-high voltage for an input signal. Default=3.3.
mask	Name of a signal to which the VIH statement applies. If you do not specify a mask, the VIH statement applies to all input signals.

Example

```
VIH 5.0  
VIH 3.5 0 0 0 0000 11111111
```

- The first VIH statement sets all input vectors to 5V, when they are high.
- The last VIH statement changes the logic-high voltage from 5V to 3.5V, for the last eight vectors.

Description

The VIH statement specifies the logic-high voltage, for each input signal to which the mask applies.

- If you do not specify the logic high voltage of the signals, in a VIH statement, HSPICE or HSPICE RF assumes 3.3.
- If you use more than one VIH statement for a signal, the last statement overrules previous statements. HSPICE or HSPICE RF issues a warning.

VIH statements have no effect on the expected output signals.

See Also

[VIL](#)
[VOH](#)
[VOL](#)
[VTH](#)

VIL

Syntax

```
VIL <logic-low_voltage>
```

Argument	Definition
<logic-low_voltage>	Logic-low voltage for an input signal. Default=0.0.
mask	Name of a signal to which the VIL statement applies. If you do not specify a mask, the VIL statement applies to all input signals.

Example

```
VIL 0.0  
VIL 0.5 0 0 0 0000 11111111
```

- The first VIL statement sets the logic-low voltage to 0V, for all vectors.
- The second VIL statement changes the logic-low voltage to 0.5V, for the last eight vectors.

Description

The VIL statement specifies the logic-low voltage, for each input signal to which the mask applies.

- If you do not specify the logic-low voltage of the signals, in a VIL statement, HSPICE or HSPICE RF assumes 0.0.
- If you use more than one VIL statement for a signal, the last statement overrules previous statements. HSPICE or HSPICE RF issues a warning.

VIL statements have no effect on the expected output signals.

See Also

[VIH](#)
[VOH](#)
[VOL](#)
[VTH](#)

4: Commands in Digital Vector Files

VNAME

VNAME

Syntax

```
VNAME vector_name[[starting_index : ending_index]]
```

Argument	Definition
<i><vector_name></i>	Name of the vector, or range of vectors.
<i>starting_index</i>	First bit in a range of vector names.
<i>ending_index</i>	Last bit in a range of vector names. You can associate a single name with multiple bits (such as bus notation). The opening and closing brackets and the colon are required; they indicate that this is a range. The vector name must correlate with the number of bits available. You can nest the bus definition inside other grouping symbols, such as {}, (), [], and so on. The bus indices expand in the specified order

Example 1

```
RADIX 1 1 1 1 1 1 1 1 1 1 1 1 1  
VNAME V1 V2 V3 V4 V5 V6 V7 V8 V9 V10 V11 V12
```

Example 2

```
VNAME a[[0:3]]
```

This example represents a0, a1, a2, and a3, in that order. HSPICE or HSPICE RF does not reverse the order to make a3 the first bit.

The bit order is MSB:LSB, which means most significant bit to least significant bit. For example, you can represent a 5-bit bus such as: {a4 a3 a2 a1 a0}, using this notation: a[[4:0]]. The high bit is a4, which represents 2⁴. It is the largest value, and therefore is the MSB.

Example 3

```
RADIX 2 4  
VNAME VA[[0:1]] VB[[4:1]]
```

HSPICE or HSPICE RF generates voltage sources with the following names:

```
VA0 VA1 VB4 VB3 VB2 VB1
```

- *VA0* and *VB4* are the MSBs.
- *VA1* and *VB1* are the LSBs.

Example 4

```
VNAME VA[[0:1]] VB<[4:1]>
```

HSPICE or HSPICE RF generates voltage sources with the following names:

```
VA[0] VA[1] VB<4> VB<3> VB<2> VB<1>
```

Example 5

```
VNAME VA[[2:2]]
```

This example specifies a single bit of a bus. This range creates a voltage source named:

```
VA[2]
```

Example 6

```
RADIX 444444  
VNAME A[[0:23]]
```

This example generates signals named A0, A1, A2, ... A23.

Description

The `VNAME` statement defines the name of each vector. If you do not specify `VNAME`, HSPICE or HSPICE RF assigns a default name to each signal: V1, V2, V3, and so on. If you define more than one `VNAME` statement, the last statement overrules the previous statement.

VOH

Syntax

```
VOH <logic-high_voltage> mask
```

Argument	Definition
<logic-high_voltage>	Logic-high voltage for an output vector. Default=2.66.
mask	Name of a signal to which the VOH statement applies. If you do not specify a mask, the VOH statement applies to all output signals.

Example

```
VOH 4.75
VOH 4.5 1 1 1 137F 00000000
VOH 3.5 0 0 0 0000 11111111
```

- The first line tries to set a logic-high output voltage of 4.75V, but it is redundant.
- The second line changes the voltage level to 4.5V, for the first seven vectors.
- The last line changes the last eight vectors to a 3.5V logic-high output.

These second and third lines completely override the first VOH statement.

If you do not define either VOH or VOL, HSPICE or HSPICE RF uses VTH (default or defined).

Description

The VOH statement specifies the logic-high voltage, for each output signal to which the mask applies.

- If you do not specify the logic-high voltage in a VOH statement, HSPICE or HSPICE RF assumes 2.64.
- If you apply more than one VOH statement to a signal, the last statement overrules the previous statements, and HSPICE or HSPICE RF issues a warning.

VOH statements have no effect on input signals.

See Also

[VIH](#)
[VIL](#)
[VOL](#)
[VTH](#)

VOL**Syntax**

```
VOL <logic-low_voltage> mask
```

Argument	Definition
<logic-low_voltage>	Logic-low voltage for an output vector. Default=0.64.
mask	Name of a signal to which the VOL statement applies. If you do not specify a mask, the VOL statement applies to all output signals.

Example

```
VOL 0.0
VOL 0.2 0 0 0 137F 00000000
VOL 0.5 1 1 1 0000 00000000
```

- The first VOL statement sets the logic-low output to 0V.
- The second VOL statement sets the output voltage to 0.2V, for the fourth through seventh vectors.
- The last statement increases the voltage further to 0.5V, for the first three vectors.

These second and third lines completely override the first VOL statement.

If you do not define either VOH or VOL, HSPICE or HSPICE RF uses VTH (default or defined).

Description

The VOL statement specifies the logic-low voltage, for each output signal to which the mask applies.

- If you do not specify the logic-low voltage, in a VOL statement, HSPICE or HSPICE RF assumes 0.66.
- If you apply more than one VOL statement to a signal, the last statement overrules the previous statements, and HSPICE or HSPICE RF issues a warning.

See Also

VIH
VIL
VOH
VTH

VREF

Syntax

```
VREF <reference_voltage>
```

Argument	Definition
<reference_voltage>	Reference voltage for each input vector. Default=0.

Example

```
VNAME v1 v2 v3 v4 v5[[1:0]] v6[[2:0]] v7[[0:3]] v8 v9 v10
VREF 0
VREF 0 111 137F 000
VREF vss 0 0 0 0000 111
```

When HSPICE or HSPICE RF implements these statements into the netlist, the voltage source realizes *v1*:

```
v1 V1 0 pwl(.....)
```

as well as *v2*, *v3*, *v4*, *v5*, *v6*, and *v7*.

However, *v8* is realized by

```
V8 V8 vss pwl(.....)
```

v9 and *v10* use a syntax similar to *v8*.

Description

Similar to the `TDELAY` statement, the `VREF` statement specifies the name of the reference voltage, for each input vector to which the mask applies. `VREF` applies only to input signals.

- If you do not specify the reference voltage name of the signals, in a `VREF` statement, HSPICE or HSPICE RF assumes 0.
- If you apply more than one `VREF` statement, the last statement overrides the previous statements, and HSPICE or HSPICE RF issues a warning.

`VREF` statements have no effect on the output signals.

VTH

Syntax

```
VTH <logic-threshold_voltage>
```

Argument	Definition
<logic-threshold_voltage>	Logic-threshold voltage for an output vector. Default=1.65.

Example

```
VTH 1.75  
VTH 2.5 1 1 1 137F 00000000  
VTH 1.75 0 0 0 0000 11111111
```

- The first VTH statement sets the logic threshold voltage at 1.75V.
- The next line changes that threshold to 2.5V, for the first 7 vectors.
- The last line changes that threshold to 1.75V, for the last 8 vectors.

All of these examples apply the same vector pattern, and both output and input control statements, so the vectors are all bidirectional.

Description

Similar to the TDELAY statement, the VTH statement specifies the logic threshold voltage, for each output signal to which the mask applies. The threshold voltage determines the logic state of output signals, for comparison with the expected output signals.

- If you do not specify the threshold voltage of the signals, in a VTH statement, HSPICE or HSPICE RF assumes 1.65.
- If you apply more than one VTH statement to a signal, the last statement overrules the previous statements, and HSPICE or HSPICE RF issues a warning.

VTH statements have no effect on the input signals.

4: Commands in Digital Vector Files

VTH

See Also

[VIH](#)
[VIL](#)
[VOH](#)
[VOL](#)

A

- ABSH option 195
- ABSI option 196, 286
- ABSMOS option 197, 286
- ABSTOL option 198
- ABSV option 199
- ABSVAR option 200
- ABSVDC option 201
- AC analysis
 - magnitude 204
 - optimization 9
 - output 204
 - phase 204
- .AC command 9
 - external data 29
- ACCURATE option 203
- ACOUT option 204
- algorithms
 - DVDT 200, 292
 - local truncation error 292, 342, 368
 - pivoting 325
 - timestep control 254
 - transient analysis timestep 292
 - trapezoidal integration 303
- .ALIAS command 14
- ALL keyword 134, 158
- ALT9999 option 205
- ALTCC option 206
- ALTCHK option 207
- alter block commands 1
- .ALTER command 16, 44
- Analog Artist interface 336
 - See also Artist
- Analysis commands 1
- analysis, network 129
- arithmetic expression 111

- ARTIST option 208, 336
- ASCII output data 221, 301, 355
- ASPEC option 209
- AT keyword 109
- autoconvergence 238
- AUTOSTOP option 210
- average measurements, with .MEASURE 105
- average nodal voltage, with .MEASURE 112
- average value, measuring 112
- AVG keyword 113

B

- BADCHR option 211, 212
- BETA keyword 156
- .BIASCHK command 18
- BIASFILE option 213
- BIAWARN option 214
- BINPRINT option 215
- bisection
 - pushout 121
- BKPSIZ option 216
- branch current error 196
- breakpoint table, size 216
- BRIEF option 134, 135, 217, 291, 309, 318, 322
- BSIM model, LEVEL 13 128
- BSIM2 model, LEVEL 39 128
- bus notation 414
- BYPASS option 218
- BYTOL option 219

C

- Cadence
 - Opus 221, 355
 - WSF format 221, 355

Index

C

capacitance
 charge tolerance, setting 222
 CSHUNT node-to-ground 229
 table of values 220
capacitor, models 124
CAPTAB option 220
CDS option 221
CENDIF optimization parameter 125
characterization of models 35
charge tolerance, setting 222
CHGTOL option 222
CLOSE optimization parameter 125
CMIFLAG option 223
CO option 181, 186, 224
column laminated data 28
commands
 .AC 9
 .ALIAS 14
 .ALTER 16, 44
 alter block 1
 analysis 1
 .BIASCHK 18
 .CONNECT 23
 .DATA 25
 .DC 32
 .DCMATCH 38
 .DCVOLT 40
 .DEL LIB 42
 .DISTO 46
 .DOUT 49
 .EBD 52
 .ELSE 54
 .ELSEIF 55
 .END 56
 .ENDDATA 57
 .ENDIF 58
 .ENDL 59
 .ENDS 60
 .EOM 61
 .FFT 62
 .FOUR 65
 .FSOPTIONS 66
 .GLOBAL 68
 .GRAPH 69
 .HDL 71
 .IBIS 72
 .IC 76
 .ICM 78

.IF 79
.INCLUDE 81
.LAYERSTACK 82
.LIB 84
.LOAD 91
.MACRO 93
.MALIAS 96
.MATERIAL 98
.MEASURE 100
.MODEL 123
.NET 129
.NODESET 131
.NOISE 132
.OP 133
.PARAM 137
.PAT 141
.PKG 143
.PLOT 145
.PRINT 147
.PROBE 151
.PROTECT 153
.PZ 154
.SAVE 157
.SENS 159
.SHAPE 161
.STIM 167
subcircuit 5
.SUBCKT 172
.TEMP 175
.TF 177
.TITLE 178
.TRAN 179
.UNPROTECT 184
.VEC 185
Verilog-A 5
.WIDTH 186
Common Simulation Data Format 252
concatenated data files 27
Conditional Block 2
conductance
 current source, initialization 265
 minimum, setting 266
 models 239
 MOSFETs 267
 negative, logging 251
 node-to-ground 270
 sweeping 268
.CONNECT command 23

- control options
 - printing 322
 - setting 135
 - transient analysis
 - limit 372
 - CONVERGE option 225, 240
 - convergence
 - for optimization 127
 - problems
 - causes 218
 - changing integration algorithm 303
 - CONVERGE option 225, 240
 - DCON setting 238
 - decreasing the timestep 261
 - .NODESET statement 131
 - nonconvergent node listing 238
 - operating point Debug mode 134
 - setting DCON 238
 - steady state 268
 - CPTIME option 226
 - CPU time, reducing 310
 - CROSS keyword 108
 - CSDF option 227
 - CSHDC option 228
 - CSHUNT option 229
 - current
 - ABSMOS floor value for convergence 341
 - branch 196
 - operating point table 134
 - CURRENT keyword 134
 - CUSTCMI option 230
 - CUT optimization parameter 125
 - CVTOL option 231
- D**
- D_IBIS option 232
 - .DATA command 25, 26
 - datanames 29
 - external file 25
 - for sweep data 29
 - inline data 29
 - data files, disabling printout 217, 318
 - DATA keyword 11, 28, 35, 181
 - datanames 29, 169
 - DC
 - analysis
 - decade variation 36
 - initialization 236
 - iteration limit 278
 - linear variation 36
 - list of points 36
 - octave variation 36
 - optimization 32
 - .DC command 32, 35
 - external data with .DATA 29
 - DCAP option 233
 - DCCAP option 234
 - DCFOR option 235
 - DCHOLD option 236
 - DCIC option 237
 - .DCMATCH command 38
 - DCON option 238
 - DCSTEP option 239
 - DCTRAN option 240
 - .DCVOLT command 40, 76
 - DEBUG keyword 134
 - DEC keyword 12, 36, 182
 - DEFAD option 241
 - DEFAS option 242
 - DEFL option 243
 - DEFNRD option 244
 - DEFNRS option 245
 - DEFPD option 246
 - DEFPS option 247
 - DEFW option 248
 - .DEL LIB command 42
 - with .ALTER 44
 - with .LIB 44
 - delays
 - group 369
 - DELMAX option 249, 349
 - DELTA internal timestep 249
 - See also timestep
 - derivative function 116
 - DERIVATIVE keyword 117
 - derivatives, measuring 108
 - DI option 250
 - DIAGNOSTIC option 251
 - DIFSIZ optimization parameters 126
 - DIM2 distortion measure 47
 - DIM3 distortion measure 47
 - diode models 124
 - .DISTO command 46

Index

E

- distortion
 - HD2 47
 - HD3 48
- distortion measures
 - DIM2 47
 - DIM3 47
- DLENCSDF option 252
- .DOUT command 49
- DV option 238, 253
- DVDT
 - algorithm 200, 345
 - option 254, 292
- DVDT option 254
- DVTR option 255

E

- .EBD command 52
- element
 - checking, suppression of 310
 - OFF parameter 319
- .ELSE command 54
- .ELSE statement 54
- .ELSEIF command 55
- Encryption 2
- .END command 56
 - for multiple HSPICE runs 56
 - location 56
- .ENDDATA command 57
- ENDDATA keyword 25, 27, 30
- .ENDIF command 58
- .ENDL command 59, 86
- .ENDS command 60
- .EOM command 61
- EPSMIN option 256
- equation 111
- ERR function 119
- ERR1 function 119
- ERR2 function 119
- ERR3 function 119
- error function 119
- errors
 - branch current 196
 - function 119
 - internal timestep too small 229, 350
 - optimization goal 103
 - tolerances

- ABSMOS 197
 - branch current 196
 - RELMOS 197
 - voltage 343, 344, 346
- example, subcircuit test 93, 172
- EXPLI option 257
- EXPMAX option 258
- expression, arithmetic 111
- external data files 30

F

- FALL keyword 108
- FAST option 259
- .FFT command 62
- FFTOUT option 260
- FIL keyword 30
- files
 - column lamination 28
 - concatenated data files 27
 - filenames 30
 - hspice.ini 301
 - include files 81, 85
 - multiple simulation runs 56
- FIND keyword 108
- FIND, using with .MEASURE 107
- floating point overflow
 - CONVERGE setting 225
 - setting GMINDC 267
- .FOUR command 65
- frequency
 - ratio 47
 - sweep 10
- FROM parameter 120
- FS option 156, 261
- .FSOPTIONS command 66
- FT option 262
- functions
 - ERR 119
 - ERR1 119
 - ERR2 119
 - ERR3 119
 - error 119

G

- GDCPATH option 263
- GENK option 264

.GLOBAL command 68
 global node names 68
 GMAX option 265
 GMIN option 266, 267
 GMINDC option 267
 GOAL keyword 113
 GRAD optimization parameter 126
 GRAMP
 calculation 238
 option 268
 .GRAPH command 69
 graph data file (Viewlogic format) 252
 group delay, calculating 369
 GSHDC option 269
 GSHUNT option 270

H

H9007 option 271
 harmonic distortion 47
 HD2 distortion 47
 HD3 distortion 48
 .HDL command 71
 HIER_SCALE option 272
 HSPICE
 job statistics report ??–202
 version
 H9007 compatibility 271
 parameter 128
 hspice.ini file 301

I

.IBIS command 72
 IBIS commands 3
 .IC command 40, 76
 from .SAVE 157
 IC parameter 40, 76, 157
 .ICM command 78
 ICSWEEP option 273
 .IF command 79
 IGNOR keyword 119
 IMAX option 274, 281
 IMIN option 275, 280
 .INCLUDE command 81
 include files 81, 85
 indepout 169

indepvar 169, 170
 inductors, mutual model 124
 INGOLD option 276, 297
 initial conditions
 saving and reusing 273
 transient 182
 initialization 319
 inline data 29
 inner sweep 26
 input
 data
 adding library data 44
 column laminated 28
 concatenated data files 27
 deleting library data 44
 external, with .DATA statement 29
 filenames on networks 28
 formats 28, 29
 include files 81
 printing 291
 suppressing printout 291
 netlist file 56
 INTEG keyword 113, 115
 used with .MEASURE 112
 integral function 115
 integration
 backward Euler method 294
 order of 294
 interfaces
 Analog Artist 336
 Mentor 302
 MSPICE 302
 ZUKEN 379
 intermodulation distortion 47
 INTERP option 277
 iterations
 limit 278
 maximum number of 282
 ITL1 option 278
 ITL2 option 279
 ITL3 option 280
 ITL4 option 281
 ITL5 option 282
 ITPTRAN option 283
 ITPZ option 284
 ITROPT optimization parameter 126
 ITRPRT option 285

Index

J

J

Jacobian data, printing 321

K

KCLTEST option 286

keywords

.AC statement parameter 11

ALL 134, 158

AT 109

AVG 113

BETA 156

CROSS 108

CURRENT 134

DATA 11, 28, 35, 181

.DATA command parameter 28

.DC command parameter 35

DEBUG 134

DEC 12, 36, 182

DERIVATIVE 117

ENDDATA 25, 27, 30

FALL 108

FIL 30

FIND 108

FS 156

IGNOR 119

INTEG 112, 113, 115

LAM 28, 30

LAST 109

LIN 12, 36, 182

MAXFLD 156

.MEASUREMENT command parameter 113

MER 27, 28, 30

MINVAL 120

MODEL 35

.MODEL statement parameters 123

MONTE 12, 181

NONE 134, 158

NUMF 156

OCT 12, 36, 182

OPTIMIZE 35

PLOT 123

POI 12, 36, 182

PP 112, 113

RESULTS 35

RIN 130

RISE 108

START 182

SWEEP 12, 36, 182

target syntax 109

TO 113, 120

TOL 156

TOP 158

.TRAN command parameter 181

TRIG 101

VOLTAGE 134

WEIGHT 113, 120

weight 113

WHEN 108

Kirchhoff's Current Law (KCL) test 286

KLIM option 287

L

LAM keyword 28, 30

laminated data 28

LAST keyword 109

latent devices

BYPASS option 218

excluding 259

.LAYERSTACK command 82

LENNAM option 288

LEVEL 13 BSIM model 128

LEVEL parameter 126

.LIB command 84

call statement 86

in .ALTER blocks 86

nesting 86

with .DEL LIB 44

libraries

adding with .LIB 44

building 86

DDL 356

defining macros 86

deleting 42

private 153

protecting 153

Library Management 3

LIMPTS option 289

LIMTIM option 290

LIN keyword 12, 36, 182

LIST option 291

listing, suppressing 153

.LOAD command 91

local truncation error algorithm 292, 342, 368

LVLTIM option 292, 303, 345, 368

M

.MACRO command 93

macros 44, 86

magnetic core models 124

.MALIAS command 96

.MATERIAL command 98

Material Properties 3

matrix

 minimum pivot values 329

 parameters 129

 row/matrix ratio 328

 size limitation 327

MAX 112

MAX parameter 113, 126

MAXAMP option 293

MAXFLD keyword 156

maximum value, measuring 112

MAXORD option 294

MBYPASS option 295

MCBRIEF option 296

MEASDGT option 297

MEASFAIL option 298

MEASFILE option 299

MEASOUT option 301

MEASSORT option 300

.MEASURE command 100, 297, 301

 average measurements 105

 average nodal voltage 112

 expression 111

 propagation delay 101

measuring average values 112

measuring derivatives 108

Mentor interface 302

MENTOR option 302

MER keyword 27, 28, 30

messages

 See *also* errors, warnings

messages, pivot change 326

METHOD option 303

MIN 112

MIN parameter 113

minimum value, measuring 112

MINVAL keyword 120

.MODEL command 123

 CENDIF 125

 CLOSE 125

 CUT 125

 DEV 125

 DIFSIZ 126

 distribution 126

 GRAD 126

 HSPICE version parameter 128

 ITROPT 126

 keyword 126

 LEVEL 126

 LOT 126

 MAX 126

 model name 124

 PARMIN 127

 PLOT 127

 RELIN 127

 RELOUT 127

 type 124

 VERSION 128

MODEL keyword 35

model parameters

 LEVEL 126

 suppressing printout of 312

 TEMP 176

models

 BJTs 124

 BSIM LEVEL 13 128

 BSIM2 LEVEL 39 128

 capacitors 124

 characterization 35

 diode 124

 JFETs 124

 magnetic core 124

 MOSFETs 124

 mutual inductors 124

 names 124

 npn BJT 124

 op-amps 124

 optimization 124

 plot 124

 private 153

 protecting 153

 simulator access 86

 types 124

models, diode 124

MODMONTE option 304

Index

N

MODSRH option 305
Monte Carlo
 AC analysis 10
 DC analysis 32
 .MODEL parameters 126
 time analysis 180
MONTE keyword 12, 181
MONTECON option 306
MSPICE simulator interface 302
MU option 307

N

namei 168, 169, 170
n-channel, MOSFET's models 124
negative conductance, logging 251
nested library calls 86
.NET comamnd 129
network
 analysis 129
 filenames 28
network analysis 129
NEWTOL option 308
Node Naming 4
NODE option 309
nodes
 cross-reference table 309
 global versus local 68
 printing 309
.NODESET command 131, 235
 DC operating point initialization 131
 from .SAVE 157
NODESET keyword 157
node-to-element list 326
NOELCK option 310
noise
 folding 156
 numerical 229
 sampling 156
.NOISE command 132
NOISEMINFREQ option 311
NOMOD option 312
NONE keyword 134, 158
NOPAGE option 313
NOPIV option 314
NOTOP option 315
NOWARN option 316

npn BJT models 124
npoints 168, 169, 170
NUMDGT option 317
numerical integration algorithms 303
numerical noise 229, 270
NUMF keyword 156
NXX option 318

O

OCT keyword 12, 36, 182
OFF option 319
.OP command 133
op-amps model, names 124
operating point
 capacitance 220
 .IC statement initialization 40, 76
 .NODESET statement initialization 131
 restoring 92
 solution 319
 voltage table 134
OPFILE option 320
optimization
 AC analysis 9
 algorithm 126
 DC analysis 32
 error function 103
 iterations 126
 models 124
 time
 analysis 180
 required 125
optimization parameter, DIFSIZ 126
OPTIMIZE keyword 35
.OPTION 135
 SEARCH 356
.OPTION ABSH 195
.OPTION ABSI 196
.OPTION ABSMOS 197
.OPTION ABSTOL 198
.OPTION ABSV 199
.OPTION ABSVAR 200
.OPTION ABSVDC 201
.OPTION ACCT 201
.OPTION ACCURATE 203
.OPTION ACOUT 204
.OPTION ALT999 203

.OPTION ALT9999 205
.OPTION ALTCC 206
.OPTION ALTCHK 207
.OPTION ARTIST 208, 336
.OPTION ASPEC 209
.OPTION AUTOSTOP 210
.OPTION BADCHR 211, 212
.OPTION BIASFILE 213
.OPTION BIAWARN 214
.OPTION BINPRINT 215
.OPTION BKPSIZ 216
.OPTION BRIEF 134, 135, 217, 291, 309, 318, 322
.OPTION BYPASS 218
.OPTION BYTOL 219
.OPTION CAPTAB 220
.OPTION CDS 221
.OPTION CHGTOL 222
.OPTION CMIFLAG 223
.OPTION CO 181, 186, 224
.OPTION CONVERGE 225
.OPTION CPTIME 226
.OPTION CSDF 227
.OPTION CSHDC 228
.OPTION CSHUNT 229
.OPTION CUSTCMI 230
.OPTION CVTOL 231
.OPTION D_IBIS 232
.OPTION DCAP 233
.OPTION DCCAP 234
.OPTION DCFOR 235
.OPTION DCHOLD 236
.OPTION DCIC 237
.OPTION DCON 238
.OPTION DCSTEP 239
.OPTION DCTRAN 240
.OPTION DEFAD 241
.OPTION DEFAS 242
.OPTION DEFL 243
.OPTION DEFNRD 244
.OPTION DEFNRS 245
.OPTION DEFPPD 246
.OPTION DEFPS 247
.OPTION DEFW 248
.OPTION DELMAX 249
.OPTION DI 250
.OPTION DIAGNOSTIC 251
.OPTION DLENCSDF 252
.OPTION DV 253
.OPTION DVDT 254
.OPTION DVTR 255
.OPTION EPSMIN 256
.OPTION EXPLI 257
.OPTION EXPMAX 258
.OPTION FAST 259
.OPTION FFTOUT 260
.OPTION FS 261
.OPTION FT 262
.OPTION GDCPATH 263
.OPTION GENK 264
.OPTION GMAX 265
.OPTION GMIN 266
.OPTION GMINDC 267
.OPTION GRAMP 268
.OPTION GSHDC 269
.OPTION GSHUNT 270
.OPTION H9007 271
.OPTION HIER_SCALE 272
.OPTION ICSWEEP 273
.OPTION IMAX 274
.OPTION IMIN 275
.OPTION INGOLD 276
.OPTION INTERP 277
.OPTION ITL1 278
.OPTION ITL2 279
.OPTION ITL3 280
.OPTION ITL4 281
.OPTION ITL5 282
.OPTION ITLPTRAN 283
.OPTION ITLPZ 284
.OPTION ITRPRT 285
.OPTION KCLTEST 286
.OPTION KLIM 287
.OPTION LENNAM 288
.OPTION LIMPTS 289
.OPTION LIMITIM 290
.OPTION LIST 291
.OPTION LVLTIM 292
.OPTION MAXAMP 293
.OPTION MAXORD 294

Index

O

.OPTION MBYPASS 295
.OPTION MCBRIEF 296
.OPTION MEASDGT 297
.OPTION MEASFAIL 298
.OPTION MEASFILE 299
.OPTION MEASOUT 301
.OPTION MEASSORT 300
.OPTION MENTOR 302
.OPTION METHOD 303
.OPTION MODMONTE 304
.OPTION MODSRH 305
.OPTION MONTECON 306
.OPTION MU 307
.OPTION NEWTOL 308
.OPTION NODE 309
.OPTION NOELCK 310
.OPTION NOISEMINFREQ 311
.OPTION NOMOD 312
.OPTION NOPAGE 313
.OPTION NOPIV 314
.OPTION NOTOP 315
.OPTION NOWARN 316
.OPTION NUMDGT 317
.OPTION NXX 318
.OPTION OFF 319
.OPTION OPFILE 320
.OPTION OPTLST 321
.OPTION OPTS 322
.OPTION PARHIER 323
.OPTION PATHNUM 324
.OPTION PIVOT 325
.OPTION PIVREF 327
.OPTION PIVREL 328
.OPTION PIVTOL 329
.OPTION PLIM 330
.OPTION POST 331
.OPTION POST_VERSION 333
.OPTION POSTTOP 334
.OPTION PROBE 335
.OPTION PSF 336
.OPTION PURETP 337
.OPTION PUTMEAS 338
.OPTION RELH 339
.OPTION RELI 340
.OPTION RELMOS 341
.OPTION RELQ 342
.OPTION RELTOL 343
.OPTION RELV 344
.OPTION RELVAR 345
.OPTION RELVDC 346
.OPTION RESMIN 347
.OPTION RMAX 349
.OPTION RMIN 350
.OPTION RUNLVL 351
.OPTION SCALE 353
.OPTION SCALM 354
.OPTION SDA 355
.OPTION SEARCH 356
.OPTION SEED 357
.OPTION SLOPETOL 358
.OPTION SPARSE 359
.OPTION SPICE 360
.OPTION SPMODEL 361
.OPTION STATFL 362
.OPTION SYMB 363
.OPTION TIMERES 364
.OPTION TNOM 365
.OPTION TRCON 366
.OPTION TRTOL 368
.OPTION UNWRAP 369
.OPTION VAMODEL 370
.OPTION VERIFY 371
.OPTION VFLOOR 372
.OPTION VNTOL 373
.OPTION WACC 374
.OPTION WARNLIMIT 376
.OPTION WL 377
.OPTION WNFLAG 375
.OPTION XDTEMP 378
.OPTION ZUKEN 379
OPTLST option 321
OPTS option 322
Opus 221, 355
oscillation, eliminating 303
outer sweep 26
Output 4
output
 data
 format 297, 336
 limiting 277

- significant digits specification 317
 - specifying 289
 - storing 301
- files
 - reducing size of 376
- .MEASURE results 100
- plotting 145
- printing 147–150
- printout format 276
- variables
 - printing 285
 - probing 151
 - specifying significant digits for 317
- ovari 168, 170

P

- .PARAM command 137
- parameters
 - AC sweep 9
 - DC sweep 32
 - defaults 323
 - FROM 120
 - IC 40, 76
 - inheritance 323
 - ITROPT optimization 126
 - LEVEL 126
 - matrix 129
 - names
 - .MODEL command
 - parameter name 127
 - simulator access 86
 - skew, assigning 85
 - UIC 40, 76
- PARHIER option 323
- PARMIN optimization parameter 127
- .PAT command 141
- path names 324
- path numbers, printing 324
- PATHNUM option 324
- p-channel
 - JFETs models 124
 - MOSFET's models 124
- Peak 105
- peak measurement 105
- peak-to-peak value 115
 - measuring 112
- PERIOD statement 401, 411
- pivot
 - algorithm, selecting 325
 - change message 326
 - reference 327
- PIVOT option 325
- PIVREF option 327
- PIVREL option 328
- PIVTOL option 325, 329
- .PKG command 143
- PLIM option 330
- plot
 - models 124
 - value calculation method 204
- .PLOT command 145
 - in .ALTER block 16
- PLOT keyword 123
- pnp BJT models 124
- POI keyword 12, 36, 182
- pole/zero analysis, maximum iterations 284
- polygon, defining 166
- POST_VERSION option 333
- POSTTOP option 334
- power operating point table 134
- PP 112, 115
- PP keyword 112, 113
- .PRINT command 147
 - in .ALTER 16
- printing
 - Jacobian data 321
- printout
 - disabling 217, 318
 - suppressing 153
 - value calculation method 204
- .PROBE command 151
- PROBE option 335
- propagation delays
 - measuring 102
 - with .MEASURE 101
- .PROTECT command 153
- protecting data 153
- PSF option 336
- PURETP option 337
- pushout bisection 121
- PUTMEAS option 338
- .PZ command 154

Index

R

R

- reference temperature 176
- RELH option 339
- RELI option 286, 340
- RELIN optimization parameter 127
- RELMOS option 197, 286, 341
- RELOUT optimization parameter 127
- RELQ option 342
- RELTOL option 222, 343
- RELTOLoption 343
- RELV option 259, 295, 344
- RELVAR option 345
- RELVDC option 344, 346
- resistance 347
- RESMIN option 347
- RESULTS keyword 35
- RIN keyword 130
- Rise 101
- rise and fall times 102
- RISE keyword 108
- rise time
 - specify 406, 407
- RISETIME option 348
- RMAX option 349
- RMIN option 350
- RMS
 - measurement 105
 - used with .MEASURE 105
- RMS keyword 113
- ROUT keyword 130
- row/matrix ratio 328
- RUNLVL option 351

S

- S parameter, model type 124
- .SAMPLE 156
- .SAMPLE statement 156
- sampling noise 156
- .SAVE command 157
- SCALE option 353
- SCALM option 354
- Schmitt trigger example 34
- SDA option 355
- SEARCH option 356

- SEED option 357
- .SENS command 159
- Setup 4
- .SHAPE command 161
 - Defining Circles 163
 - Defining Polygons 164
 - Defining Rectangles 162
 - Defining Strip Polygons 166
- SIM2 distortion measure 48
- simulation
 - accuracy 203, 292
 - improvement 254
 - multiple analyses, .ALTER statement 16
 - multiple runs 56
 - reducing time 29, 210, 218, 254, 275, 280, 358, 368
 - results
 - plotting 145
 - printing 147
 - specifying 100
 - title 178
- Simulation Runs 5
- skew, parameters 85
- SLOPE statement 411
- SLOPETOL option 358
- small-signal, DC sensitivity 159
- source
 - AC sweep 9
 - DC sweep 32
- SPARSE option 359
- SPICE
 - compatibility 360
 - AC output 204
 - plot 330
- SPICE option 360
- SPMODEL option 361
- START keyword 182
- statement
 - PERIOD 401, 411
 - SLOPE 411
 - TDELAY 411
 - TFALL 411
 - TRISE 411
 - TSKIP 401
 - TUNIT, with TRISE statement 406, 407
- statements
 - .AC 9
 - .ALIAS 14

.ALTER 16, 44
 alter block 1
 .BIASCHK 18
 .CONNECT 23
 .DATA 25
 external file 25
 inline 25
 .DC 32, 35
 .DCMATCH 38
 .DCVOLT 40, 76
 .DEL LIB 42
 .DISTO 46
 .DOUT 49
 .EBD 52
 .ELSE 54
 .ELSEIF 55
 .END 56
 .ENDDATA 57
 .ENDIF 58
 .ENDL 59, 86
 .ENDS 60, 61
 .EOM 61
 .FFT 62
 .FOUR 65
 .FSOPTIONS 66
 .GLOBAL 68
 .GRAPH 69
 .HDL 71
 .IBIS 72
 .IC 40, 76
 .ICM 78
 .IF 79
 .INCLUDE 54, 56, 79, 81, 157
 .LAYERSTACK 82
 .LIB 84, 86
 nesting 86
 .LOAD 91
 .MACRO 93
 .MALIAS 96
 .MATERIAL 98
 .MEASURE 100, 297, 301
 .MODEL 123
 .NET 129
 .NODESET 131, 235
 .NOISE 132
 .OP 133
 .OPTION SEARCH 356
 .PARAM 137

.PAT 141
 .PKG 143
 .PLOT 145
 .PRINT 147
 .PROBE 151
 .PROTECT 153
 .PZ 154
 .SAMPLE 156
 .SAVE 157
 .SENS 159
 .SHAPE 161
 .STIM 167
 .SUBCKT 172
 .TEMP 175, 176
 .TF 177
 .TITLE 178
 .TRAN 179
 .UNPROTECT 184
 .VEC 185
 .WIDTH 186
 STATFL option 362
 statistics, listing 202
 statements
 ELSE 54
 .STIM command 167
 subcircuit commands 5
 subcircuits
 calling 93, 172
 global versus local nodes 68
 names 94, 174
 node numbers 94, 174
 parameter 60, 61, 93, 94, 172, 174
 printing path numbers 324
 test example 93, 172
 .SUBCKT command 172
 sweep
 data 26, 301
 frequency 10
 inner 26
 outer 26
 SWEEP keyword 12, 36, 182
 SYMB option 363

T

Tabular Data section
 time interval 401
 TARG_SPEC 101

Index

U

- target specification 102
- TDELAY statement 411
- TEMP
 - keyword 12, 36
 - model parameter 176
- .TEMP command 175
- temperature
 - AC sweep 9
 - DC sweep 32, 33
 - derating 176
 - reference 176
- .TF command 177
- TFALL statement 411
- threshold voltage 49
- time 134
 - See also CPU time
- TIMERES option 364
- timestep
 - algorithms 254
 - calculation for DVDT=3 261
 - changing size 342
 - control 261, 345, 368
 - internal 249
 - maximum 274, 281, 349
 - minimum 275, 280, 350
 - reversal 200
 - setting initial 249
 - transient analysis algorithm 292
 - variation by HSPICE 249
- .TITLE command 178
- title for simulation 178
- TNOM option 176, 365
- TO keyword 113, 120
- TOL keyword 156
- TOP keyword 158
- .TRAN command 179
- transient analysis
 - Fourier analysis 65
 - initial conditions 40, 76
 - number of iterations 282
- TRAP algorithm
 - See trapezoidal integration
- trapezoidal integration
 - coefficient 307
- TRCON option 366
- TRIG keyword 101
- TRIG_SPEC 101
- trigger specification 102

- TRISE statement 406, 407, 411
- TRTOL option 368
- TSKIP statement 401
- TSTEP
 - multiplier 349, 350
 - option 349, 350
- TUNIT statement 411
 - with TRISE statement 406, 407

U

- U Element, transmission line model 124
- UIC
 - keyword 182
 - parameter 40, 76
- .UNPROTECT command 184
- UNWRAP option 369

V

- VAMODEL option 370
- .VEC command 185
- VERIFY option 371
- Verilog-A commands 5
- version
 - H9007 compatibility 271
 - HSPICE 128
- Version Options 369, 370
- VFLOOR option 372
- Viewlogic graph data file 252
- VIH statement 412
- VIL statement 413
- VNTOL option 259, 373
- VOH statement 416, 418
- voltage
 - error tolerance
 - DC analysis 344, 346
 - transient analysis 343
 - initial conditions 40, 76
 - iteration-to-iteration change 253
 - logic high 412, 416, 418
 - logic low 413
 - maximum change 200
 - minimum
 - DC analysis 201
 - listing 372
 - transient analysis 199
 - operating point table 134

- relative change, setting 345
- tolerance
 - MBYPASS multiplier 295
 - value for BYPASS 219
- VOLTAGE keyword 134
- VREF statement 420
- VTH statement 421

W

- W Elements transmission line model 124
- WACC option 374
- warnings
 - limiting repetitions 376
 - misuse of VERSION parameter 128
 - suppressing 316
- WARNLIMIT option 376
- WEIGHT keyword 113, 120

- WHEN keyword 108
- WHEN, using with .MEASURE 107
- .WIDTH command 186
- WL option 377
- WNFLAG option 375
- WSF output data 221, 355

X

- XDTEMP option 378

Y

- YMAX parameter 120
- YMIN parameter 119

Z

- ZUKEN option 379

Index
Z