

Contents

1	Introduction	1
2	Simulation Options	2
	Bibliography	6
A	Appendix: Scale Factor Notations and Units	7
A.1	Numeric Scale Factor	7
A.2	Units	8

1 Introduction

HSPICE is a circuit simulation program from a company named Synopsys®. “It is the industry’s *gold standard* for accurate circuit simulation and offers foundry-certified MOS device models with state-of-the-art simulation and analysis algorithms” [1]. Hence, it is widely used in the semiconductor industry to design and simulate silicon chips.

“SPICE” stands for Simulation Program with Integrated Circuit Emphasis which was originally developed by the UC Berkeley group, in particular Larry Nagel. HSPICE is one of the commercial variations of the original tool which includes remarkable advancements. The ‘H’ in Hspice stood for the first letter of the family name of Ashawna Hailey (1949–2011) who started a company named “Meta-Software” to created the HSPICE tool. “Meta-Software” eventually became a part of Synopsys. A HSPICE simulation has three primary steps:

- (1) Generating the circuit netlist file (xxx.sp)
- (2) Running the simulation, and
- (3) Displaying, analyzing, and printing the simulation results

HSPICE takes in the netlist (a simple text file with a “.sp” extension), which contains:

- circuit description
- analysis options, (see Section 2)
- analysis commands
- required outputs

The simulation results are recorded in the files such as the one with a “.lis” extension.

The following HSPICE documentation is available through the DOE network:

- HSPICE User Guide: Basic Simulation and Analysis
- HSPICE User Guide: Signal Integrity Modeling and Analysis
- HSPICE User Guide: Advanced Analog Simulation and Analysis
- HSPICE Reference Manual: Commands and Control Options
- HSPICE Reference Manual: MOSFET Models
- HSPICE Reference Manual: Elements and Device Mod

2 Simulation Options

The followings is a sample of options that we commonly use.

```
.OPTION PROBE  
.OPTION POST=1  
.OPTION LIST=0  
.OPTION INGOLD=2
```

*** For tran sim:

```
.OPTION ACCURATE  
.OPTION DVDT=2  
.OPTION LVLTIM=2  
.OPTION TRTOL=10
```

We will use some/all of these options for different simulation tasks. A summary of their descriptions is as follows.

-
- .OPTION POST=1

Saves simulation results for viewing by an interactive waveform viewer.

HSPICE Syntax: .OPTION POST=[0|1|2|3|ASCII|BINARY|CSDF]

Default:

- if option is not specified in the netlist: 0
- if option name is specified without a corresponding value: 1

The defaults for the POST option supply usable data to most parameters:

- POST=0: Does not output simulation results.
 - POST=1, BINARY: (Default if POST is declared without a value) Output format is binary. If you want to use “HSPICE-toolbox” this option should be set as POST=1.
 - POST=2, ASCII: Output format is ASCII.
 - POST=3: Output format is New Wave binary (which enables you to generate .tr0 files that are larger than 2 gigabytes on Linux platforms).
-

-
- .OPTION INGOLD=2

This controls whether HSPICE prints “*.lis” file output in exponential form or engineering notation in HSPICE/HSPICE RF.

HSPICE Syntax: .OPTION INGOLD=[0|1|2]

Default:

- if option is not specified in the netlist: 0
- if option name is specified without a corresponding value: 1
 - INGOLD=0 Engineering Format; defaults 1.234K, 123M
 - INGOLD=1 G Format (fixed and exponential); defaults 1.234e+03, .0123
 - INGOLD=2 E Format (exponential SPICE); defaults 1.234e+03, .123e-1

Use this option to control if HSPICE prints output in exponential form (scientific notation) or engineering notation. Engineering notation provides two to four extra significant digits and aligns columns to facilitate comparison, as:

$$\begin{array}{ccccccc} F = 1e - 15, & M = 1e - 3, & P = 1e - 12, & K = 1e3, \\ N = 1e - 9, & X = 1e6, & U = 1e - 6, & G = 1e9 \end{array}$$

For more information see [\[2\]](#).

-
- .OPTION ACCURATE

The ACCURATE option increases the accuracy of the results. It also selects a time algorithm for circuits such as high-gain comparators.

HSPICE Syntax: .OPTION ACCURATE=[0|1]

Default:

- If option is not specified in the netlist: 0
- If option name is specified without a corresponding value: 1

Use this option to select 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. When set to 1, it sets the above control options; while the default does not set the above control options.

For more information see [\[2\]](#).

-
- .OPTION DVDT=2

It adjusts the time-step, based on rates of change for node voltage. Use this option to adjust the timestep

based on rates of change for node voltage.

HSPICE Syntax: `.OPTION DVDT=0|1|2|3|4`

- 0: Original algorithm
- 1: Fast
- 2: Accurate
- 3,4: Balance speed and accuracy

The ACCURATE option also increases the accuracy of the results.

Default: 4 (regardless of runlvl setting)

For more information see [2].

- `\n.OPTION RMIN=1.0e-10`

It sets the minimum value of delta (internal timestep).

- The default is 1.0e-9. Min value: 1e-15.
- 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.
- See Also: `.OPTION FT` and `.OPTION IMAX`

For more information see [2].

- `.option runlvl=3`

For Analog or mixed signal accuracy RUNLVL=3-5 (**Default:** RUNLVL=3)

The RUNLVL algorithm provides the following characteristics:

- Simplifies accuracy control by setting RUNLVL values between 1 and 6 with 6 discrete settings (1=fastest, 6=most accurate).
- Avoids interpolation error in `.MEASURE` statements by using the interpolating polynomial used by the time integration method.
- Dynamically checks for correct handling of input signals and controlled sources between computed time steps to avoid setting small time steps before transient simulation start.
- Allows HSPICE to take time steps no larger than $(Tstop - Tstart)/20$.
- DELMAX automatically sets $(Tstop - Tstart)/20$ if there is no specific setting of DELMAX. The effect is that, for example, HSPICE can take larger time steps for flat regions.

For more information see [2].

- `.OPTION INTERP=0|1`

Default:

- if option is not specified in the netlist: 0 (engineering notation)
- if option name is specified without a corresponding value: 1

Descriptions:

- The stepsize you define in: `".tran stepsize tmin tmax"` is just used for the plotting purpose.
- If the stepsize Hspice takes is larger than what you asked it provides the output by interpolation.
- By default, HSPICE outputs data at internal timepoints.
- To push Hspice to go with the defined step size we should add in netlist: `.OPTION INTERP`
- When using INTERP, make sure you set TSTEP to the intervals you need the simulation data to be printed at.
- However, in some cases, INTERP produces a much larger design `.tr#` file, especially for smaller timesteps, and it also leads to longer runtime.

For more information see [\[2\]](#).

-
- `.OPTION LVLTIM=2`

HSPICE Syntax: `LVLTIM=0,2,3` Default=1

- Selects the timestep algorithm for transient analysis.
- `LVLTIM=2` uses the local truncation error (LTE) timestep control method.
- You can apply `LVLTIM=2` to the TRAP method.
- 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.

For more information see [\[2\]](#).

-
- `.OPTION TRTOL=10`

It estimates the amount of error introduced when the timestep algorithm truncates the Taylor series expansion.

Description:

- Use this option timestep algorithm for local truncation error (`LVLTIM=2`).
 - HSPICE multiplies `TRTOL` by the internal timestep, which is generated by the timestep algorithm for the local truncation error.
 - `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 timestep 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, HSPICE uses a very small timestep. As you increase the TRTOL setting, the timestep size increases.

For more information see [2].

References

- [1] (2016) The HSPICE page on the Synopsys website. [Online]. Available: <http://www.synopsys.com/Tools/Verification/AMSVerification/CircuitSimulation/HSPICE/Pages/default.aspx>
 - [2] *HSPICE Reference Manual: Commands and Control Options*, Synopsys Inc., Mountain View, CA, Mar. 2013, ver. H-2013.03.
 - [3] *HSPICE Quick Reference Guide*, Synopsys Inc., Mountain View, CA, Version X-2005.09, Sep. 2005.
-

A Appendix: Scale Factor Notations and Units

A.1 Numeric Scale Factor

Number may be an integer (e.g. 0, 10, 1000), floating point number (e.g. 10.5, 3.14159), an integer or floating point number followed by an integer exponent (e.g. 22E3, 5E-9, 1e-14, 2.65e3) or any number followed by one of the scale factors listed below.

Use in Netlist (Suffix)	Multiplying Factor	Description
A, a	= 1e-18	Atto
F, f	= 1e-15	Femto
P, p	= 1e-12	Pico
N, n	= 1e-9	Nano
U, u	= 1e-6	Micro
MI, mi	= 25.4e-6 (or 1/1000 of an inch)	Mil
M, m	= 1e-3	Milli
K, k	= 1e3	Kilo
MEG, meg (or X, x)	= 1e6	Mega
G, g	= 1e9	Giga
T, t	= 1e12	Tera
DB, db	= $20 \log_{10}$	

Notes:

- Scale factors are not accumulative as with other simulators (for example, 1KK does not equal 1MEG)
- Both upper and lower case letters are allowed in HSPICE. So a capacitor of 0.1 nanofarad in the following ways: 0.1n, 0.1N, 0.1PF, 0.1pFarad, 100E-12, 100P, etc.
- Any letters that are not scale factors and immediately follow an entry number are ignored, with the exception of O or I. for example: 153d is the same as 153.
- The letters O and I are not allowed in alphanumeric numbers. If an O or I follows a number, it results in fatal error. Because they are easily confused with the numbers 0 and 1.

A.2 Units

Use in Netlist	Unit	Use in Manual
A	Angstrom	Å
amp	ampere	A
cm	centimeter	cm
deg	degree	deg (degree Centigrade (°C) unless specified as kelvin (K))
H	Henry	H
s	Second	s
V	volt	V
(Default) do not use, confused with exponent	electron volt	eV
(Default) do not use, confused with femto	Farad	F
(Default) do not use, confused with milli	meter	m
(Default) do not use, confused with 0	Ohm	Ohm, Ω

By combining the "scale factor" and "unit" a proper value for a component can be defined in the Hspice netlist. For example time can be defined using any one of the followings.

Use in Netlist	Description
fs	femtosecond (fs)
ps	picosecond (ps)
ns	nanosecond (ns: default)
us	microsecond (μ s)
ms	millisecond (ms)

Note: The reference(s) used for this section: [3]