
Useful Information about Hspice®

By: Dr. Behzad Nouri

Last-Update: February 3, 2019

Contents

1	Introduction	3
2	Hspice® Netlist Simulation Control Options	4
2.1	.OPTION POST	4
2.2	.OPTION INGOLD	5
2.3	.OPTION ACCURATE	5
2.4	.OPTION DVDT	6
2.5	.OPTION RMIN	6
2.6	.OPTION RUNLVL	7
2.7	.OPTION INTERP	7
2.8	.OPTION LVLTIM	8
2.9	.OPTION TRTOL	8
3	HSPICE® Options to Define Threshold Values	9
3.1	.OPTION RESMIN	9
3.2	.OPTION RM_RMIN	9
3.3	.OPTION RM_RMAX	10
3.4	.OPTION RM_CMIN	11
3.5	.OPTION RM_CMAX	11
3.6	.OPTION RM_IMIN	12
4	Notations for Units and Numeric Scale Factor	13
4.1	Units	13
4.2	Numeric Scale Factor	14
4.3	Numeric Scale Factor	16

Bibliography	17
A Appendix: Summary of the Units and Scale-Factors in Hspice® Netlist	18
B Appendix: MATLAB® Script to Write General OPTIONS	20
C Appendix: MATLAB® Script to Write General OPTIONS	22

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 create 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 3)
- 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 generally made available along with the installation of the tool. To access you may check the installation location in the computer (if accessible).

Typical Hspice® documentation.

- 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 Hspice® Netlist Simulation Control 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.

2.1 .OPTION POST

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

Definition of the values for this option:

- 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 for MATLAB” (a set of third-party MATLAB® routines to manipulate and view signals generated by Hspice® simulations) 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).

2.2 .OPTION INGOLD

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

Definition of the values for this option:

- 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:

Table: Summary of Scale Factors in Hspice® Netlist.

A = 1e-18	F = 1e-15	P = 1e-12
N = 1e-9	U = 1e-6	M = 1e-3
K = 1e3	X = 1e6	G = 1e9
T = 1e12		

For more information see Appendix ?? and [2].

2.3 .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].

* * *

2.4 .OPTION DVDT

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

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

Default: 4 (regardless of runlv1 setting)

Definition of the values for this option:

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

The ACCURATE option also increases the accuracy of the results.

For more information see [2].

* * *

2.5 .OPTION RMIN

It sets the minimum value of delta (internal time-step).

HSPICE Syntax: .OPTION RMIN=1.0e-10

Default: 1.0e-9 (if not defined.)

Description:

- The default is 1.0e-9 and the minimum allowable value is 1e-15.
- An internal time-step smaller than $RMIN \times TSTEP$, terminates the transient analysis, and reports an internal “time-step too small” error.
- If the circuit does not converge in IMAX iterations, delta decreases by the amount you set in the FT option.
- Also see: .OPTION FT and .OPTION IMAX.

For more information see [2].

* * *

2.6 .OPTION RUNLVL

For Analog or mixed signal accuracy it should be defined as $RUNLVL = 3 - -5$.

HSPICE Syntax: .OPTION RUNLVL=1|2|3|4|5|6

Default: RUNLVL=3

Description:

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 $(T_{stop}-T_{start})/20$.
- DELMAX automatically sets $(T_{stop}-T_{start})/20$ 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].

2.7 .OPTION INTERP

HSPICE Syntax: .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 time points.
- 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 time-steps, and it also leads to longer runtime.

For more information see [2].

2.8 .OPTION LVLTIM

Hspice® Syntax: .OPTION LVLTIM=0|2|3

Default: 1 *Descriptions:*

- Selects the time-step algorithm for transient analysis.
- LVLTIM=2 uses the local truncation error (LTE) time-step control method.
- You can apply LVLTIM=2 to the trapezoidal 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].

2.9 .OPTION TRTOL

It estimates the amount of error introduced when the time-step algorithm truncates the Taylor series expansion. If you set TRTOL to 1, Hspice® uses a very small time-step. As you increase the TRTOL setting, the time-step size increases.

Hspice® Syntax: .OPTION TRTOL=0.01-100

Typical Values: 1 to 10

Description:

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

For more information see [2].

3 HSPICE® Options to Define Threshold Values

The followings are the thresholds for the important components' values such as Resistors, Capacitors, and Inductors. Sample of the important thresholds is Summarized in below followed by more details.

RESMIN Smaller than this are reset to value e.g. RESMIN=1e-5,
 RM_RMIN Smaller than this are replaced with SHORT-Circuit. It has "*higher priority*" than RESMIN.
 e.g. RM_RMIN=1e-28,
 RM_RMAX Greater than this are replaced with OPEN-Circuit (removed) ((RM_RMAX=0 has no effect),
 RM_CMIN Smaller than this are removed,
 RM_CMAX Greater than this are removed (RM_CMAX=0 has no effect),
 RM_LMIN Smaller than this are removed,

* * *

3.1 .OPTION RESMIN

This specifies the minimum resistance for all resistors. Resistors in the netlist whose values are smaller than specified RESMIN is reset to the RESMIN value by Hspice®.

Hspice® Syntax: .OPTION RESMIN=val

Default value: 1e-05

Use this option to specify the minimum resistance for all resistors. Any resistance (including parasitic, inductive resistors, and those in the transistor models) smaller than the specified RESMIN is reset to the RESMIN value. The default is 1e-05. Users can specify a bigger value up to 10 Ω.

* * *

3.2 .OPTION RM_RMIN

This enables user to set a value for the resistors below which Hspice® ignores resistors.

Hspice® Syntax: .OPTION RM_RMIN=val

Default value: 1e-28

If the value of a resistor is less than this value, it is replaced with short-circuit. The minimum value is 0 and the maximum value is 100.

Use this option to specify a threshold at which resistors are ignored. This option is especially useful with extracted netlists containing very small resistors. Specifying such a threshold helps to speed up simulation. All linear resistors satisfying $|Rvalue| < RM_{RMIN}$ is replaced with a short circuit.

Description:

- Its priority is higher than .OPTION RESMIN,
- To disable the option, set the value to 0,
- If a negative value is set, Hspice® issues a warning message and the simulation ignores the option,
- The minimum value: 0,
- The maximum value: 100,

Examples:

```
.OPTION RM_RMIN=1e-3 RM_RMAX=1e12
```

In the following example, resistors smaller than 1e-3 are shorted (ignored) and the resistors greater than 1e12 are removed from the circuit.

_____ * * * _____

3.3 .OPTION RM_RMAX

Enables you to set a value above which HSPICE removes resistors from the circuit.

Hspice® Syntax: .OPTION RM_RMAX=val

Default value: 0 (Disabled)

Use this option to specify a threshold at which resistors are removed. This option is especially useful with extracted netlists containing numerous resistors. Specifying such a threshold can speed up simulation. All linear resistors that encounter an $|Rvalue| > RM_{RMAX}$ are immediately removed (ignore the resistor). The minimum value is 0 and the maximum value is 1e+20.

Description:

- The priority of .OPTION RM_RMAX is higher than .OPTION RESMIN or .OPTION RM_RMIN,
- If a negative value is set, Hspice® issues a warning message and the simulation ignores the option.
- The minimum value: 0 (disabled, no effect),
- The maximum value: 1e+20,

Examples:

```
.OPTION RM_RMIN=1e-3 RM_RMAX=1e12
```

In the following example, resistors smaller than 1e-3 are shorted (ignored) and the resistors greater than 1e12 are removed from the circuit.

_____ * * * _____

3.4 .OPTION RM_CMIN

Enables you to set a value below which Hspice® ignores capacitors. The minimum value is 0 and the maximum value is 100.

Hspice® Syntax: .OPTION RM_CMIN=val

Default value: 0 (Disabled)

Use this option to specify a threshold at which linear capacitors are ignored. This option is especially useful with extracted netlists containing numerous very small capacitors. Specifying such a threshold helps to speed up simulation.

Description:

- If a negative value is set, Hspice® issues a warning message and the simulation ignores the option.
- The minimum value: 0 (disabled, no effect),
- The maximum value: 100,

Examples:

```
.OPTION RM_CMIN=1e-3
```

In the following example, capacitors less than 1e-3 are removed from the circuit.

_____ * * * _____

3.5 .OPTION RM_CMAX

Enables you to set a value above which HSPICE removes capacitors from the circuit.

Hspice® Syntax: .OPTION RM_CMAX=val

Default value: 0 (Disabled)

Use this option to specify a threshold at which linear capacitors are removed. This option is especially useful with extracted netlists containing numerous capacitors. Specifying such a threshold can speed up simulation. All capacitors that encounter an $|Cvalue| > RM_CMAX$ are immediately removed.

Description:

- If a negative value is set, Hspice® issues a warning message and the simulation ignores the option.
- The minimum value: 0 (disabled, no effect),
- The maximum value: 1e+20,

Examples:

```
.OPTION RM_CMAX=1e12
```

In the following example, the capacitors greater than 1e12 are removed from the circuit.

_____ * * * _____

3.6 .OPTION RM_IMIN

For Inductor this parameter was not found in the Hspice® documentation!

```
.OPTION RM_IMIN=0 (disabled, no effect).
```

_____ * * * _____

The above information was summarized from [2, Chapter 3]. For more details [2] can be referred to.

4 Notations for Units and Numeric Scale Factor

4.1 Units

Table: Units in Hspice® Netlist.

Use in Netlist	Unit	Use in Manual
A	Angstrom	Å
amp	ampere	A
cm	centimeter	cm
H	Henry	H
s	Second	s
V	volt	V
deg	degree	deg (degree Centigrade (°C) unless specified as kelvin (K))
(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.

4.2 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 as shortly listed below. A complete list presented in the following table.

Table: Summary of Scale Factors in Hspice® Netlist.

A = 1e-18	F = 1e-15	P = 1e-12
N = 1e-9	U = 1e-6	M = 1e-3
K = 1e3	X = 1e6	G = 1e9
T = 1e12		

Table: Scale Factors in Hspice® Netlist.

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 $\frac{1}{1000}$ inch)	Mil
M, m	= 1e-3	Milli
K, k	= 1e3	Kilo
X, x (or MEG, meg)	= 1e6	Mega
G, g	= 1e9	Giga
T, t	= 1e12	Tera
DB, db	= $20 \log_{10}$	

For example time can be defined using any one of the followings.

Table: Sample Time Scale Factors in Hspice® Netlist.

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

Notes:

- **Non-accumulative:** Scale factors are not accumulative as with other simulators (for example, 1KK does not equal 1MEG)
- **Non-case-sensitive:** Both upper and lower case letters are allowed in Hspice®. So a capacitor of 0.1 nano-farad in the following ways: 0.1n, 0.1N, 0.1PF, 0.1pF, 100E-12, 100P, etc.
- **Undefined scale-factor is ignored:** 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.
- **Fatal-Error:** 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.

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

4.3 Numeric Scale Factor

References

- [1] (2016) The HSPICE page on the Synopsys website. [Online]. Available: <http://www.synopsys.com/Tools/Verification/AMSVerification/CircuitSimulation/HSPICE/Pages/default.aspx> 3
 - [2] *HSPICE Reference Manual: Commands and Control Options*, Synopsys Inc., Mountain View, CA, USA, Mar. 2013, ver. H-2013.03. 5, 6, 7, 8, 12
 - [3] *HSPICE Quick Reference Guide*, Synopsys Inc., Mountain View, CA, USA, Version X-2005.09, Sep. 2005. 15
-

A Appendix: Summary of the Units and Scale-Factors in Hspice® Netlist

Table: Units in Hspice® Netlist.

Use in Netlist	Unit	Use in Manual
A	Angstrom	Å
amp	ampere	A
cm	centimeter	cm
H	Henry	H
s	Second	s
V	volt	V
deg	degree	deg (degree Centigrade (°C) unless specified as kelvin (K))
(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

Table: Summary of Scale Factors in Hspice® Netlist.

A = $1e-18$	F = $1e-15$	P = $1e-12$
N = $1e-9$	U = $1e-6$	M = $1e-3$
K = $1e3$	X = $1e6$	G = $1e9$
T = $1e12$		

Table: Scale Factors in Hspice® Netlist.

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 $\frac{1}{1000}$ inch)	Mil
M, m	= 1e-3	Milli
K, k	= 1e3	Kilo
X, x (or MEG, meg)	= 1e6	Mega
G, g	= 1e9	Giga
T, t	= 1e12	Tera
DB, db	= $20 \log_{10}$	

Table: Sample Time Scale Factors in Hspice® Netlist.

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

B Appendix: MATLAB® Script to Write General OPTIONS

By running this MATLAB® code a set of more general options is written in a file to be included in your Hspice® netlist.

```
% Write Hspice Simulation Options
% By: Dr. Behzad NOURI
% Last Update: 2019/02/20
%-----
close all; clc; clear; %Intialization

FN = 'xoption.sp';
%-----
[OptFid,errmsg] = fopen(FN,'wt');
if OptFid<0, error(errmsg); end

fprintf(OptFid, '\n***Options:');
fprintf(OptFid, '\n.OPTION PROBE');

%-----
%POST = x Stores simulation results for analysis by using AvanWaves
...interface or other methods.
...POST = 1 saves results in binary. (for Hspice-toolbox)
...POST = 2 saves results in ASCII.
...POST = 3 saves results in New Wave binary format.
fprintf(OptFid, '\n.OPTION POST=1\t\t\t\t$POST=1 is for Hspice-toolbox');
%-----

%-----
%LIST=0: Do NOT Print ckt element list(0=None). Setting it to 1 Prints all
... ckt element summary table and para definition. Default is
fprintf(OptFid, '\n.OPTION LIST=0');
%-----

fprintf(OptFid, '\n.OPTION INGOLD=2') ;%$2=Numbers in exp. format');
fprintf(OptFid, '\n');

%fprintf(OptFid, '\n\n***For Freq. sim:');
%fprintf(OptFid, '\n');

fprintf(OptFid, '\n\n***For tran. sim:');

%-----
%ACCURATE: Selects a time algorithm; uses LVLTIM=3 and DVDT = 2 for
...circuits such as high-gain comparators. Default is 0.
fprintf(OptFid, '\n.OPTION ACCURATE');
%-----

fprintf(OptFid, '\n.OPTION DVDT=2');

%-----
%LVLTIM: Selects the timestep algorithm for transient analysis.
...If LVLTIM = 1 (default), HSPICE uses the DVDT timestep algorithm.
...If LVLTIM = 2, HSPICE uses the local truncation error (LTE) timestep control method.
...If LVLTIM = 3, HSPICE uses the DVDT timestep algorithm with timestep reversal.
fprintf(OptFid, '\n.OPTION LVLTIM=2') ;%if 2, use local truncation error (LTE) ...
time-step control method
%-----

fprintf(OptFid, '\n.OPTION TRTOL=10') ;%Range:0.01-to-100, Use this when ...
LVLTIM=2, As TRTOL increases,the time-step increases
fprintf(OptFid, '\n*.OPTION RMIN=1.0e-10') ;%Min of time-step, default-val:1.0e-9. ...
Min-val:1e-15
```

```

fprintf(OptFid, '\n');

%-----
%INTERP: Default is 0. By setting it to 1, Hspice limits output to only the
...timestep intervals in .TRAN command (faster post processing)and enables
...printing of output variables at their internal time points
fprintf(OptFid, '\n*.OPTION INTERP=1');
%-----
fprintf(OptFid, '\n');

fprintf(OptFid, '\n\n***For Selecting Integration Method:');
%-----
%RUNLVL: Controls the speed and accuracy trade-off. It can be set to
...0,1,2,3,4,5,6. Higher values of RUNLVL result in higher accuracy and
...longer simulation times, while lower values give lower accuracy and
...faster simulation runtimes.
...RUNLVL=0 turns off this algorithm.
...RUNLVL=1 is the lowest simulation runtime.
...RUNLVL=3 is the default (similar to original HSPICE default mode).
...RUNLVL=5, 6 correspond to the HSPICE standard accurate mode. For most circuits,
...RUNLVL=5 is similar to the HSPICE standard accurate mode.
fprintf(OptFid, '\n*.OPTION RUNLVL=5')           ;%5 or 6 are the most accurate
%-----

fprintf(OptFid, '\n*.OPTION METHOD=TRAP PURETP')   ;%sets pure trapezoidal method ...
integration. No Gear-2 or Backward-Euler is mixed in.
                                                    %...e.g. use this setting when ...
                                                    you simulate harmonic ...
                                                    oscillators.
fprintf(OptFid, '\n*.option method=gear maxord=1') ;%sets pure Backward-Euler ...
integration
fprintf(OptFid, '\n*.option method=gear')         ;%sets pure Gear-2 integration
fprintf(OptFid, '\n*.option method=bdf')         ;%Sets the higher order backward ...
differentiation formulation integration for supported models.

fprintf(OptFid, '\n');

fclose(OptFid);

%End-of-File

```

C Appendix: MATLAB® Script to Write General OPTIONS

Sample xoption.sp file generated by running the MATLAB® code in Appendix B.

```
***Options:
.OPTION PROBE
.OPTION POST=1          $POST=1 is for Hspice-toolbox
.OPTION LIST=0
.OPTION INGOLD=2

***For tran. sim:
.OPTION ACCURATE
.OPTION DVDT=2
.OPTION LVLTIM=2
.OPTION TRTOL=10
*.OPTION RMIN=1.0e-10

*.OPTION INTERP=1

***For Selecting Integration Method:
*.OPTION RUNLVL=5
*.OPTION METHOD=TRAP PURETP
*.option method=gear maxord=1
*.option method=gear
*.option method=bdf
```