

NAME

`nspice`, `nspice_pp`, `nspice_bs` - encapsulated spice simulator

SYNOPSIS

nspice cell commandfile

nspice_pp cell commandfile

nspice_bs cell

DESCRIPTION

Nspice is the encapsulated spice simulator that performs a *spice* (1ICD) analysis for a circuit cell that is present in the database, using a commandfile with *spice* and/or *sls* (1ICD) commands. *Nspice* consists of a unix shell-script that extracts a circuit cell from the database using *xspice* (1ICD), converts a commandfile with *spice* and/or *sls* commands to a spice commandfile using *nspice_pp*, runs *spice*, and finally executes *nspice_bs* to back-substitute node numbers in the output file by their corresponding names as specified in the database. The output of *nspice* is found in a file called "*cell.ana*".

nspice_pp: The program *nspice_pp* converts, for a particular cell (first program argument), a commandfile with *spice* and/or *sls* commands (second program argument) to a spice commandfile. In the inputfile nodes may be referred to by their names as specified in the database. The output of the program is sent to *stdout*. The conversion is executed by using information from a spice network description that is present in the file "*cell.spc*". This file is typically extracted from the database using *xspice* with the options **-hilmf**. This will generate a file "*cell.spc*" with a full (hierarchical) description of the cell, including the appropriate transistor models, terminals for n-bulk and/or p-bulk connections, and a name translation table that translates node numbers into node names as used in the database. To reduce the number of nodes in the spice network description the use of the options **-xy** [**-z name**] is further recommended (see manual page of *xspice*).

All *sls* commands that are allowed with the *sls* simulator (see "SLS: Switch-Level Simulator User's Manual") may be used in the commandfile of *nspice_pp*. However the ones that are translated into spice commands are the following: The **"set"** commands in the commandfile are translated into piecewise linear voltage sources that are connected to the relevant input nodes of the network. If **"option level=3"** is specified, *sls* **"plot"** commands are translated into **".print tran node1 node2 ..."** cards, and a transient analysis card will be placed in the outputfile that specifies a simulation from $t=0$ till $t=t_{end}$, where t_{end} is given by "simperiod * sigunit" or otherwise - if simperiod is not specified - by the time of the last input signal transition. The value of *tstep* in the **".tran"** card should explicitly be defined with the keyword **tstep** in the commandfile (see below).

The other information that is allowed in the commandfile should be specified in the following way:

```
*%  
  
... (1) translation information ...  
  
*%  
  
... (2) direct spice commands ...  
  
*%
```

The separators **'*%'** should be placed at the beginning of each line. The first part consists of translation information that is used to translate *sls* commands (if any) into spice commands. It may consist of zero or more of the following lines:

```
tstep    <real>  
tstart   <real>  
uic
```

```

trise    <real>
tfall    <real>
vhigh    <real>
vlow     <real>

```

The field <real> denotes a specification of a real value. It may optionally have a scale factor 'G' (1e9), 'M' (1e6), 'k' (1e3), 'm' (1e-3), 'u' (1e-6), 'n' (1e-9), 'p' (1e-12) or 'f' (1e-15). The variable **tstep** specifies the value of tstep as used for transient analysis. This value should be specified when “**option level=3**” is used with the sls commands (see above). The variable **tstart** optionally specifies the start time for the transient analysis card. If the keyword **uic** is used, this keyword will be added to the “.tran” card. The variables **trise** and **tfall** respectively specify the rise and fall time of the input signals (default: trise = tfall = tstep). The variables **vhigh** and **vlow** respectively specify the voltages that correspond to the high-state and the low-state of the input signals (default: vhigh = 5 and vlow = 0). For the x-state a voltage of “(vhigh + vlow) / 2” is used.

In the second part of the spice information, direct spice commands may be specified. These commands are directly copied to the output, however with node names that are specified between quotes translated to their corresponding node numbers. E.g. “.ic v("out")=5v” might be translated into “.ic v(12)=5v”. If no sls commands are used, this part of the commandfile - beside the information that is extracted with *xspice* - will contain all the information that is necessary for the spice simulation.

An example of a commandfile with only *spice* commands is given below:

```

*%
*%
vphil1 "phil1" 0 pulse (5v 0v 29.4ns 0.6ns 0.6ns 29.4ns 60ns)
vphi2 "phi2" 0 pulse (0v 5v 29.4ns 0.6ns 0.6ns 29.4ns 60ns)
vvdd "vdd" 0 5v
vin "in" 0 pulse (5v 0v 29.4ns 0.6ns)
.tran 0.2n 120n
.print tran v("phil1") v("phi2") v("1") v("2") v("3")
.print tran v("4") v("in") v("out") v("out_inv")
.options nomod
.options limpts=601
.options cptime=100
.ic v("3")=0 v("out")=5
*%

```

An example of a commandfile with mixed *sls* and *spice* commands is as follows:

```

set phil = (h*30 l*30)*~
set phi2 = (l*30 h*30)*~
set vdd = h
set vss = l
set in = h*30 l
option level = 3
option simperiod = 120
option sigunit = 1n
option outacc = 10p
plot phil phi2 1 2 3 4 in out out_inv

/*

*%

```

```

tstep 0.2n
trise 0.6n
tfall 0.6n
*%
.options nomod
.options limpts=601
.options cptime=100
.ic v("3")=0 v("out")=5
*%

*/

```

(Note that the spice information is placed between sls comment signs ("/*" and "*/") so as to allow to use the same inputfile for an sls simulation.)

nspice_bs: The program *nspice_bs* back-substitutes node numbers by node names in a spice output file for a cell (specified as argument). This is done according to a name translation table that should be present in the "cell.spc" file. The name of the output file must be "cell.ana". *Nspice_bs* will only back-substitute node numbers that are in the header of a transient analysis table.

The *nspice* unix-shell script has the following form:

```

# xspice: always use options -hilf;
#      the use of options -xy [ -z name ] is recommended
xspice -hilf -xy $1
nspice_pp $1 $2 >> $1.spc
spice $1.spc > $1.ana
if nspice_bs $1
then
    exit 0
else
    exit 1
fi
# eof

```

EXAMPLE

```
$ nspice latch latch.cmd
```

AUTHOR

A.J. van Genderen, S. de Graaf

FILES

commandfile	input file
cell.spc	generated input file for spice
cell.axa	temporary file
cell.ana	output file

SEE ALSO

sls(1ICD), xspice(1ICD), spice(1ICD), simeye(1ICD),
 User's Guide for SPICE, Univ. of California at Berkeley,
 SLS: Switch-Level Simulator User's Manual.

DIAGNOSTICS

Nspice_pp and *nspice_bs* send their diagnostics to *stderr*. Spice diagnostics are found in the output file "cell.ana".