

The following list HSPICE specific parameters which have been used in BSIM3v3 models extracted for HSPICE. For convenience, they are listed in alphabetical order, by category.

1. General parameters

BINFLAG - This is not used for PSpice.

SCALM, SCALE In HSPICE these are the "model scaling factor" and "element scaling factor". There are no PSpice equivalents.

TREF - Temperature at which parameters are extracted. Use TNOM for PSpice.

VERSION - This is not used for PSpice.

2. Length and Width

LREF, WREF - In HSPICE these are "channel length reference" and "channel width reference". There are no PSpice equivalents.

XW, XL - In HSPICE these account for masking and etching effects. Use WINT and LINT in PSpice. $WINT_{new} = WINT_{old} - XW/2$. $LINT_{new} = LINT_{old} - XL/2$.

3. MOS Diode Model Parameters

ACM - In HSPICE this selects the area calculation method. PSpice uses only one model for the bulk-to-source and bulk-to-drain diodes. It corresponds to the HSPICE equivalent of ACM=0. Do not use this parameter for PSpice. ACM is not needed if AS, AD, PS, and PD are specified explicitly.

CJGATE - In HSPICE this is the zero-bias gate-edge sidewall bulk junction capacitance used with ACM=3 only. There is no PSpice equivalent.

HDIF, LDIF - In HSPICE this is the "length of heavily doped diffusion" and "length of lightly doped diffusion". They are used with the HSPICE ACM=2 MOS diode models, and there are no PSpice equivalents. HDIF and LDIF are not needed if AS, AD, PS, and PD are specified explicitly.

RDC - In HSPICE this is additional drain resistance due to contact resistance. If RD is specified, use $RD_{new} = RD_{old} + RDC$. If RSH is specified, then RDC should be added to $RD = NRD * RSH$. Since NRD is a device parameter and not a model parameter, a typical NRD value must be used.

RSC - In HSPICE this is additional source resistance due to contact resistance. If RS is specified, use $RS_{new} = RS_{old} + RSC$. If RSH is specified, then RSC should be added to $RS = NRS * RSH$. Since NRS is a device parameter and not a model parameter, a typical NRS value must be used.

WMLT, LMLT - In HSPICE these are "length of heavily doped diffusion" and "length of lightly doped diffusion" used in the ACM=1-3 models. There are no PSpice equivalents. WMLT and LMLT are not needed if AS, AD, PS, and PD are specified explicitly.

4. Capacitance Parameters

CAPOP - Do not use CAPOP for PSpice. CAPMOD is included in the BSIM3v3 model, and also in PSpice. CAPOP is HSPICE specific, and not included in the BSIM3v3 parameter set (nor in PSpice). The default BSIM3v3 capacitance model is CAPMOD=2, which is recommended. CAPMOD=3 will cause non-convergence! <JET, 06/12/02>

CJM - This is an HSPICE alias for CJ.

MJ0 - This is an HSPICE alias for MJ.

PJ - This is an HSPICE alias for PB.

CTA, CTP - In HSPICE these are the "junction capacitance CJ temp. coeff." and "junction sidewall capacitance CJSW temp. coeff.", used with TLEVC=1. There are no PSpice equivalents.

PTA, PTP - In HSPICE these are the "junction potential PB temp. coeff." and "fermi potential PHI temp. coeff", used with TLEVC=1 or 2. There are no PSpice equivalents.

PHP - This is an HSPICE alias for PBSW. Use PBSW for PSpice.

TLEV, TLEVC - In HSPICE this is the "temperature equation level selector" and "temperature equation level selector for junction capacitances and potentials". Do not use these parameters for PSpice.

5. Impact Ionization

ALPHA, LALPHA, WALPHA, VCR, LVCR, WVCR, and IIRAT - are impact ionization parameters in HSPICE. There are no PSpice equivalents. BSIM3v3 has its own impact ionization model which is instead used in most cases.

6. V3.2 parameters

PSpice 9.2 implements BSIM3v3.1. The following parameters are new for v3.2:

ALPHA1, ACDE, MOIN, NOFF, VOFFCV. All except ALPHA1 are used in a new capacitance model (CAPMOD=3). ALPHA1 modifies the substrate current equation as follows:

$$I_{sub} \sim (\text{ALPHA0} + \text{ALPHA1} * \text{Leff}) / \text{Leff}$$

LIBRARY FORMATTING CONSIDERATIONS.....

Anything of the Form

Replace With

='MixedMathAndAlphaExpression'

={MixedMathAndAlphaExpression}

=ParamName

={ParamName}

LEVEL=<some number>

LEVEL=7

No LEVEL Statement

LEVEL=7

.LIB TT (or whatever)

*.LIB TT (or whatever)

|

|

.ENDL

*.ENDL

[Then copy this block of text to a new file "TT.LIB"]

MODEL NAME ISSUES.....

Model names of the form "NMOS.1" cannot be used in PSpice.

This is called "binning" and is an HSpice-specific method which assigns different models for different device dimensions.

I am working this issue, and have a complex solution that is not yet available to other PSpice users.

For now, replace NMOS.1 with NMOS_1, and designate a specific model name for each symbol placement.