Review of Various Available Spice Simulators

Rajendra Pratap
Research Student, EED,
Motilal Nehru National Institute of Technology, Allahabad, India
rajendra_pratap@ymail.com

Vineeta Agarwal
Professor, EED
Motilal Nehru National Institute of Technology, Allahabad, India
vineeta@mnnit.ac.in

R K Singh
Professor, EED,
Motilal Nehru National Institute of Technology, Allahabad, India
rksingh@mnnit.ac.in

***Abstract* – In this review paper we have tried to understand various existing transistor models defined, used and understood by various industry simulators viz., Spectre, LTSpice, PSPICE and HSpice. Comparative studies of industry standard spice simulators have also been done viz PSPICE, Spectre and HSpice. Towards the end numerical analysis method has been described to give an overview of the numerical methods used by various simulators during circuit analysis. The paper is organized into three sections. In Section I we discuss about various models understood by Spectre. We have given detailed discussion on each of the effects which are simulator specific and required in Spectre to achieve better and more accurate simulation results.**

Keywords – Simulators, Spectre, PSPICE, HSpice

I. SECTION I

In this section we give syntax level details of device models used by various simulators.

Model Statement

PSPICE: .model *mname* NMOS|PMOS Level=3 [param= *value*]*

Spectre: model *mname* mos3 type= [n|p] [param= *value*]*

Level refers to the MOSFET model that describes the terminal I-V characteristics of the transistors. Level 1 is the simplest MOSFET model and is in general sufficient for describing discrete transistors. Level2 includes l and uses extensive second-order parameters, while Level 3 is a semi-empirical model that is better suited for short-channel transistors. Another model that is especially suited to model short-channel effects is called the BSIM model.

PSPICE MODEL

This simulator provides six MOSFET device models, which differ in the formulation of the I-V characteristics.

The Level parameter selects among different models during simulation runtime.

Level=1:Shichman-Hodges model
Level=2:Geometry-based, analytic model
Level=3:Semi-empirical, short-channel model
Level=4:BSIM model
Level=5:EKV model version 2.6
Level=6:BSIM3 model version 2.0
Level=7:BSIM3 model version 3.1

The DC characteristics of the first three model levels are defined by the parameters VTO, KP, LAMBDA, PHI, and GAMMA. Unlike the other models in PSPICE, the BSIM model is designed for use with a process characterization system that provides all parameters. Therefore, there are no defaults specified for the parameters, and leaving one out can cause simulation problems. The Level 4 (BSIM) and Level 6 (BSIM3 version 2) models have their own capacitance model, which conserves charge and remains unchanged.

Spectre MODELs

There are various additional model versions of MOSFETs available in Spectre simulator - BSIM3, BSIM3V3, BSIM4, level 0, 1, distortion model 1000.1100, 11010, 11011, level 2 and level 3

The MOS0 model is a simplified MOS level-1 model. The MOS0 DC drain current model is different from the Shichman and Hodges model because body effects are not modeled. The intrinsic MOS gate capacitances are replaced by the following linear overlap capacitances:

Gate to source/drain (capmod = overlap)
Gate to bulk (capmod = bulk)
Gate, source, and drain to ground (capmod = gnd)

MOS0 is usually used as a MOS switch. This model recognizes all the MOS and BSIM instance parameters but only uses l and w, ignoring all other parameters.

The MOS1 model is derived from the FET model of Shichman and Hodges. The velocity saturation and the mobility variation effects can also be incorporated into MOS1. Three charge models are available.

The MOS2 model is the level-2 model from Berkeley SPICE. The MOS2 model is an analytical, one-dimensional model that incorporates most of the second-order small-size effects. Above three charge models are also available.

The MOS3 model is the level-3 model from Berkeley SPICE, and is a semi-empirical model. The three charge models are available.

Model Comparison between PSPICE and Spectre Level1 MODELS

Drain current model parameters are same for both. Both include Variable depletion layer model which include parameters like gamma, lambda, phi, vt0, theta, vmax, Kp.

Process parameters are also same which include doping concentration, charge density, diffusion, width and length variation caused due to masking or etching effects.

Impact ionization parameters are not specified in PSPICE. These parameters come into effect due to short channel lengths, the

978-1-4799-5912-9/14/$31.00 ©2014 IEEE

high velocity of electrons in presence of high longitudinal fields that can generate electron-hole (e-h) pairs by impact ionization, that is, by impacting on silicon atoms and ionizing them. Overlap Capacitance are also present in both i.e the lateral diffusion is same for both and thus generated parasitic capacitance called overlap capacitance.

Parasitic resistance parameters are also not included in PSPICE. These parameters contribute for drain/source diffusion length contact resistance, lateral diffusion beyond gate, spacing between contacts etc.

Junction diode parameters defer with the presence of explosion current parameter in Spectre. If I_{melt} is specified (in the model or in Options) junction explosion current = I_{melt}; otherwise, if I_{max} is specified (in the model or in Options) junction explosion current = I_{max}; otherwise, junction explosion current = model I_{melt} default value (which is the same as the model I_{max} default value). Junction capacitance parameters are also specified in both. Operating region warning parameters which specify warning for over current, junction reverse breakdown, oxide breakdow i.e. parameters crucial for device protection Temperature effects are also not included in level 1 model of PSPICE, such as temperature dependence of some constants like lambda, kappa, drain/source resistance, junction capacitance, bandgap temperature offset, mobility temperature coefficient etc.

Some parameters of Noise model are missing in PSPICE such as flicker noise frequency exponent etc. In the expression for flicker i.e. power spectral density of noise current, measured in squared amperes per hertz include those parameters, where frequency exponent is the exponent over frequency in the expression. Flicker noise expression works in entire region of operation of MOSFET while thermal noise is valid only for saturation region. Flicker noise model is derived from carrier fluctuation along with mobility fluctuation model. Flicker noise is dominant at lower frequencies.

Auto model parameters which help in deciding which model is to be used depending on the perimeter size are not present there in PSPICE. These are max, min channel length, width etc.

Degradation parameters: These are included in Spectre while not in PSPICE. These parameters take care of the fact that threshold voltages have the tendency to drift over time. This is mainly due to hot carrier effect. With the decrease in device length i.e. for short channel devices, electrical field is increased for the given supply voltage. This results in the penetration of electrons in the oxide layer beneath the gate. This leads to the increase in threshold voltages for NMOS and decrease of threshold voltage for PMOS.

Also trans-conductance degrades due to surface mobility variation series resistance and threshold voltage and device scaling. Inversion layer thickness becomes comparable to thin oxide thickness in small devices and gate channel capacitance approaches slowly the oxide capacitance. Thus peak trans-conductance is limited

So, these degradation parameters gives the limit for the amount of degradation of the MOSFET parameters which in-turn affects MOSFET characteristics such as trans-conductance and threshold voltage and thus drain current.

Some parameters are the limits for max trans-conductance and threshold voltage and some are constants which help in calculating those limiting voltages. Following stress parameters are also not there in PSPICE. Stress introduces enhancement or suppression of dopant diffusion during the processing. The threshold voltage and other second order effects are affected. Thus critical field for calculating v_{dsat} is affected and gate voltage, substrate current also gets modified. BERT- Berkeley Reliability Tool – parameters include critical electric field coefficient and substrate current coefficient and dependence of these parameters on device dimensions. BSIM3v3 is the version-3 of BSIM3 model. The versions supported are 3.1, 3.2, 3.21, 3.22, 3.23 and 3.24. It uses single-piece equations for all regions to improve the smoothness of the model characteristics. BSIM3v3 also allows the binning option. This option is provided for people who want to achieve the highest accuracy of the model. The binning equation is given by

$$P = P_0 + P_l / L_{eff} + P_w / W_{eff} + P_p / (L_{eff} * W_{eff})$$

Only the P_0 parameters are listed. P_l, P_w, and P_p are not shown but can be recognized. This device is supported within alter groups. It has also been noticed that there are some extra parameters in BSIM3v3 than in Level 1 model such as – defaults for instance parameters, some parameters are noise model DC mismatch dependent, cross term dependent mos table and lod model, gate leakage current parameter, sub-threshold parameters, o/p resistance, substrate current, etc.

Comparison between PSPICE vs Spectre MOSFET MODELs except Level1 models

Leaving all the parameters that are already included in level 1 model, rest are discussed here. Threshold voltage parameters non-uniform doping and short channel length affects threshold voltage. The whole device operation is divided into three regions depending on gate terminal voltage compared to threshold voltage. Vertical non-uniform doping, lateral non-uniform doping, narrow channel effect and short channel effects are some effects which influence threshold voltage. So, a general equation is given which covers all these regions. Thus corresponding parameters are listed in this list such as lateral non uniform doping coefficient, short channel effect coefficient, narrow width coefficient, no saturation coefficient etc.

Below described parameters are available in both PSPICE and Spectre –

Mobility parameters – Scattering mechanism responsible for surface mobility includes coulombic scattering, surface roughness and phonons. Mobility as such depends on various process parameters and bias conditions such as gate oxide thickness doping concentration, threshold voltage, gate voltage and substrate voltage, bulk charge effect. Bulk charge comes into picture when either drain voltage is very high or very long channel length is there in device. Due to this the depletion thickness does not remain constant and thus affects the threshold voltage.

Output resistance parameters – The output resistance (which is the reciprocal of the first order derivative of the I-V curve) curve can be clearly divided into four regions in which have distinct R_{out} *vs.* V_{ds} dependences. The first region is the triode (or linear) region in which carrier velocity is not saturated. The other three regions belong to the saturation region. There are three physical mechanisms which affect the output resistance in the saturation region: channel length modulation (*CLM*), drain-induced barrier lowering (*DIBL*) and the substrate current induced body effect (*SCBE*). Channel length modulation (*CLM*) dominates in the second region, *DIBL* in the third region, and *SCBE* in the fourth region. So parameters corresponding to these effects are included here. Although these parameters are also present in both cases.

Sub-threshold parameters – The sub-threshold drain current is the current that flows between the source and drain of a MOSFET when the transistor is in sub-threshold region, or weak-inversion region, that is, for gate-to-source voltages below the threshold voltage. The sub-threshold region is often referred to as the weak inversion region. It is also included in both simulators. Defaults for instance parameters are not in PSPICE while Spectre has these defaults.

Length and width parameters – are there to remove bias dependencies of the device from channel length and width. Some of the terms are also for the user to modify the dependencies.

DC mismatch – A transistor is described by an analytical model consisting of several electrical parameters such as the threshold voltage *V*T, the gain factor κ, the mobility reduction θ or the drift resistance *RD*. To describe the mismatch of a high-voltage transistor, an appropriate linear variance model is introduced based on an analytical model of the MOS transistor. This variance models relate the relative mismatch of the measured current $\sigma(\_ID/ID)$ to the mismatch of the transistor parameters *V*T (threshold voltage), κ (gain factor), θ (mobility reduction) and *RD* (drift resistance). These parameters are not included in PSPICE.

LOD (level of detail) model parameters – These parameters are also not included in PSPICE. These are the parameters which define extreme details of the device. These are in general not needed and are selectable too. If user wants to incorporate them then they can use them or they can be skipped. The parameters included in this are stress effect parameters for mobility, threshold, etc. It is also not present in PSPICE simulator.

Different MOSFET MODELs available in HSPICE

Here we consider different MOS models, namely LEVEL-2, 3, 13(BSIM1), 29 (Synopsis proprietary model based on BSIM1) and 39. Each model has its own strength and weaknesses.

Level2:

- The Level2 model is an enhanced *Grove equation*. It is the most common MOS model in all simulator.

  I_{DS} eqn:

  For $V_{gs}>V_{th}$,

  $I_{ds}=\beta\{(Vgs-Vbi-\eta.Vde/2)\ Vde-2/3\ \square[(PHI+Vde+Vsb)^{3/2}-(PHI+Vsb)^{3/2}]\}$

  Where Vde=min(Vds, Vdsat) The basic current equation with the 3/2-power terms.
- Later channel length modulation and vertical field reduction effects were added.

Level 3

- It is computationally more efficient, replacing the 3/2-power terms with a first order Taylor expansion.
- It includes the drain-induced barrier lowering effect (ETA parameter).
- It is impressively physical, modeling all two dimensional effects based on junction depth and depletion depths.

Level 13(BSIM1)

- The approach is empirical rather than physical.
- It frequently uses polynomials. This makes it easier to write a parameter extraction program, but the polynomials often behave poorly.

The inclusion of polynomial approach doesn't prove beneficial as it produces a quadratic that is non-monotonic, causing $G_{DS} < 0$ problem.

The Synopsys implementation of BSIM1 as LEVEL13 MOSFET device model removed discontinuities in the current function, added temperature parameters and diode & capacitance models consistent with other models.

Level 28: It is Synopsys device model for submicron devices. It basically targets the problems associated with BSIM1, which are as follows

- Negative G_{DS}
- Poor behavior of some polynomial expressions.
- A kink in GM at threshold

It also includes some additional parameters

- Model is designed for optimization
- It doesn't include a simple extraction program.

- It routinely optimizes I_{DS}, G_{DS} and GM data.

Level 39:

- It is designed to model deep submicron devices.
- It uses a cubic spline to produce a smooth weak inversion transition.
- It has many additional parameters for improved accuracy.
- The G_{DS} transition at V_{DSAT} is markedly smoother than in BSIM1.

SUMMARY:

The list of models shows a trend towards empirical rather physical models, and an increasing number of parameters. Short-channel devices are much more sensitive to the detail of the process. I-V curves from different manufactures show qualitative differences in the shape of the curves. Therefore, the models need to be very flexible, requiring a large number of empirical parameters.

II. SECTION II

In this section we discuss simulator algorithm level details of each simulator and also do a comparative study among them

PSPICE vs Spectre as Simulators

1. The capabilities of Spectre circuit simulator are similar in function and application to SPICE, but Spectre is not descended from SPICE.
2. Spectre and SPICE use the same basic algorithms eg. Newton Raphson, direct matrix solution, but every algorithm is newly implemented.
3. Manual claims Spectre algorithms are the best currently available and is faster, accurate, more reliable and more flexible than previous SPICE like simulators
4. The Spectre circuit simulator can simulate larger circuits than other simulators because its convergence algorithms are effective with large circuits, because it is fast, and because it is frugal with memory and uses dynamic memory allocation. For large circuits, the Spectre circuit simulator typically uses less than half as much memory as SPICE
5. The capacitance-based nonlinear MOS capacitor models used in many SPICE derivatives can create or destroy small amounts of charge on every time step. The Spectre circuit simulator avoids this problem because all Spectre models are charge-conserving.
6. Spectre diagnosis mode, available as an options statement parameter, gives user information to help diagnose convergence problems. While such an option is not available in PSPICE.
7. PSPICE does not support Level0 model while Spectre does.
8. Spectre CMI option lets to include new devices using C language interface. While PSPICE does not support this.
9. The periodic small-signal analyses–periodic AC (PAC) analysis, periodic transfer function (PXF) analysis, and periodic noise (Pnoise) analysis in PSPICE are similar to Spectre's AC, XF, and Noise analyses, but the PSPICE's small-signal analyses are limited to circuits with DC operating points. While in Spectre small signal analysis can be applied to circuits with periodic operating points, such as mixers, choppers, oscillators, frequency dividers etc.
10. If portability of design across different environment is desired a SPICE based simulator is recommended.
11. If we talk for models that are same in both, then it is noticed that Spectre has more parameters than PSPICE. It can be stated that PSPICE is more general than Spectre, thus PSPICE is more portable than Spectre.

Parameter ranges: (for spectre)

Parameter ranges have following two limits:

(1)Hard limit – Hards limit is enforced by the Spectre simulator. If it is violated, the simulator issues an error and terminates.

(2)Soft limit – If it is violated, the simulator issues a warning and terminates. Soft limits are used to define reasonable ranges for parameter values and can help find "unreasonable" values that are likely errors. One can change soft limits, which are defined in one or more files.

HSPICE vs Spectre

The Spectre mode accepts only the following ANSI standard (SI) scale factors:

$T=10^{12}$, $G=10^{9}$, $M=10^{6}$, $K=10^{3}$, $k=10^{3}$, $\_=10^{-1}$, $\%=10^{-2}$, $c=10^{-2}$, $m=10^{-3}$, $u=10^{-6}$, $n=10^{-9}$, $p=10^{-12}$, $f=10^{-15}$, $a=10^{-18}$

SI scale factors are case sensitive. SPICE mode accepts only the following SPICE scale factors:

$t=10^{12}$, $g=10^{9}$, $meg=10^{6}$, $k=10^{3}$, $p=10^{-12}$, $m=10^{-3}$

$mil=25.4*10^{-6}$ $u=10^{-6}$ $n=10^{-9}$

HSPICE scale factors are not case sensitive.

Available Types of Analysis:

1. DC analysis
2. AC/small signal analysis
3. Noise analysis
4. Transfer function analysis - Linearizes the circuit about the DC operating point and performs a small signal analysis. It calculates the transfer function from every source in the circuit to a specified output.
5. S-parameter analysis(sp)
6. Time –domain reflectometer analysis (tdr).
7. Envelope –following analysis (envlp).
8. Periodic AC analysis (pac).
9. Periodic distortion analysis (pdisto).
10. Periodic steady state analysis (pss).
11. Periodic noise analysis (pnoise).
12. Periodic transfer function analysis (pxf).
13. Sensitivity analysis (sens).
14. Fourier analysis (fourier).
15. Sweep analysis (sweep).
16. Monte carlo analysis (montecarlo).

Trading off speed and accuracy of simulation is possible in Spectre. In Spectre user has three ways to achieve this tradeoff:

1) Liberal mode

2) Moderate mode

3) Conservative

Model equation evaluation criteria:

Following are some general features of the HSPICE MOSFET models that are same for all levels:

- Potential for good fitting to data.
- Ease of fitting to data.
- Behavior follows actual devices in all circuit condition.
- Ability to simulate process variation
- Gate capacitance modeling.

Generally, the model with largest number of parameters has the best potential fit for the purpose of comparing the models. Simulation counts the number of parameters in the two ways – Simulation counts only the drain current parameters, not the diode or series resistance, nor gate capacitance and impact ionization because these are almost same for all levels.

(1)Number of parameters like:

Level2: VTO, PHI, GAMMA, XJ, DELTA, UO, ECRIT, UTRA, UEXP, NSUB, LAMBDA, NFS

LEVEL3: VTO, PHI, ETA, DELTA, XJ U0, THETA.

LEVEL13: VFBO, PHIO, K1, K2KETAO, X2E, etc.

(2)Minimal number of parameters – the minimal number of parameters is a subset of the above parameters, which user uses to fit a specific W/L device.

Each model has certain number of physical parameter and empirical parameters like:

- For Level 2: PHI, XJ, U0, ECRIT, NSUB and NFS are physical. And VTO, GAMMA, UCRIT, UTRA, UEXP and LAMBDA are empirical.
- For Level 3: PHI, XJ, UO, VMAX NSUB and NFS are physical, which is 55% of the overall model parameters.
- For Level 13: 12% physical
- For Level 28: 11% physical
- For Level 39:7% physical.

CONVERGENCE PROPERTIES OF THE MODELS:

- Discontinuities in the GM, G_{DS} and GMBS derivatives can cause convergence problems. Also, because real devices are continuous derivatives, a discontinuity leads to a large inaccuracy in the derivatives near the discontinuous region. The most common discontinuities are G_{DS} at $v_{ds}=v_{dsat}$ and GM at $v_{gs}=v_{th}$. The Level 2 and 3 models contain these continuities, but the Level13, 28 and 39 models don't.
- The Level13 model (BSIM1) often produces a negative G_{DS}, which is obviously inaccurate, and causes oscillation, which can lead to convergence failure or a timestep too small error. A Level13 Model can avoid negative G_{DS}, but it depends on complex relationships between the MUZ, X2M, MUS, X2MS, X3MS, U1, X2U1 and X3U1 parameters. X3MS=0 is set to remove negative GDS.
- The Level39 (BSIM2) model can also produce negative G_{DS}, unless parameters are selected carefully.
- The Level28 model does not create negative G_{DS}.
- The Level39 model internally protects against conditions in the Level13 model that cause convergence problems due to negative output conductance.

Comparison of parameters for MOSFET model LEVELS:

Level	2	3	13	28	39
Continuous derivatives	fail	fail	Pass	pass	Pass
+ve GDS	Pass	pass	Fail	pass	fail
Monotonic GM/IDS in weak inversion	fail	fail	Pass	pass	fail
Number of parameters	13	12	60	63	99
Minimal number of parameters	12	11	17	18	28
Physical parameters	50%	55%	12%	11%	7%

Following are several new BSIM-type MOSFET models supported by HSPICE:

- Level 47 BSIM3 V-2
- Level 49 and 53 BSIM3v3 MOS model
- Level 54 BSIM4
- Level 57 UC Berkeley BSIM3-SOI model
- Level 59 UC Berkeley BSIM3-SOI FD model
- Level 60 UC Berkeley BSIM3-SOI DD model
- Level 65 SSIMSOI model

Level 47:

This model is based on BSIM 3 v-2 from UC Berkeley.

Following are some features of Level 47:

- The Level 47 model supports the TNOM model parameters as an alias for TREF.
- Here user can select one of three ways to calculate Vth:

  1) Using K1, K2 values that we specify.

  2) Using GAMMA1, GAMMA2, VBM and VBX values that we enter in .MODEL statement.

  3) Using NPEAK, NSUB, XT and VBM values.
- We can enter the NPEAK & UO model parameters in meter or centimeter.
- Level 47 uses the common Synopsys MOS parasitic models, which ACM specifies.
- Level 47 uses the MOS noise models, which NLEV specifies.

Level49: Level49 is an HSPICE –enhanced version of BSIM3v3. It maintains compliance with UC Berkeley release of BSIM3v3 with the following three exceptions.

- Default parameter values – To eliminate differences in default parameters values, Level49 explicitly assigns the CAPMOD and XPART parameters and set ACM=10
- Parameter range limits — Differences occur only in the severity of the warning for five parameters. Level49 issues warning that model exceeded the parameter range, but continue with the simulation however Berkeley release issues a fatal error and aborts the simulation.
- Improvement in numerical stability.

Level53: Maintains full compliance with the Berkeley release, including numerically –identical model equation.

Level49 and 53 both support the following instance parameters, along with the DELVTO instance parameter for local mismatch and NBTI (negative bias temp instability). Both levels support a superset of model parameters that include HSPICE – specific parameters.

For Level53, in all cases, HSPICE specific parameters default to OFF. The single exception in Level49 is that ACM defaults to 0.

III. SECTION III

In this section we discuss the generic numerical analysis method commonly referred and used by various spice simulators. Newton Rapson method forms the simulator core engine. Various industry simulators use modified and optimized version of this method. Below we discuss details of this method as used by spice simulators.

Newton Rapson (N-R) Method: It is a mathematical tool to determine the roots of non linear equations. The main advantage of using N-R method is its fast convergence rate. The mathematical equation used to determine the next approximate root (after making initial guess) is as follows:

$X_a = X_a - f(X_0)/f`(X_a)$ where X_0 = initial approx root.

Guess criteria:

The method requires an initial guess, not of the interval [x1,x2] but of single value of X which must be near the real root. Such value of X can be found by graphical sketch of given function against X. The initial guess value of Xo can be either side of the intersection X value.

Termination criteria:

One can continue the process of finding new guess of root till the new guess of root satisfies f(x)=0 exactly. Many times to arrive at this situation hundreds of the steps will be required or the situation may not be arrived at all due to limitation of precision in number representation on the computer. Hence the iterative process of finding new guess is stopped when $|f(x)| \leq$ some specified limit □, or error between X_a and X_0 is less than some specified limit which can be tested by checking if relative error in root is less than specified limit **e**.

i.e. $|(X_a - X_0)/X_a| \leq \mathbf{e}$

The iterative process will be stopped by either

$|f(x)| \leq$ □ or $|(X_a - X_0)/X_a| \leq e$

Whichever is satisfied earlier, an additional situation that has to be taken into consideration is the one when f `(x) is very small, almost nearly zero. In such case next new guess becomes infinite. Thus iterative process must be stopped if magnitude of f `(x) is less than or equal to some small value.

Limitations of this method:

- If the value of gradient at guess point is very small then even a very small change in the gradient value of function has very large effect on the next value of guess and the next guess may be taken away from the exact root.
- Iteration may be divergent if initial guess is not proper.
- If initial guess is not appropriate, Newton Rapson method provides no guarantee of convergence.

IV. References

[1] HSpice Simulation and Analysis Users Guide https://webspace.utexas.edu/rd8658/www/hspice_sa.pdf

[2] HSpice MOSFET Model Manual www.ece.rochester.edu/courses/ECE222/hspice/hspice_MOSFET.pdf

[3] Virtuoso Simulator Circuit Components and Device Models Manualhttp://emf.mit.edu/afs/cs.pitt.edu/usr0/don/spectre/doc/spectremod/spectremod.pdf

[4] Virtuoso Spectre Circuit Simulator http://petrus.upc.es/rfcs/Material/Cadence%202004/spectreuser.pdf

[5] SPICE : User Guide and Reference http://www.sel.eesc.usp.br/informatica/graduacao/material/sel301/private/SPICE-User's%20Guide%20and%20Reference.pdf

[6] OrCAD PSPICE User's Guide http://www.electronics-lab.com/downloads/schematic/013/tutorial/PSPICE.pdf

[7] PSPICE Advanced Analysis User Guide http://ecee.colorado.edu/~ecen4827/spice/PSPICE/pspaugca.pdf

[8] Spice 3 User's Manual http://newton.ex.ac.uk/teaching/cdhw/Electronics2/userguide/

[9] Power Spice Model User Manual www.fairchildsemi.com/models_static/spicemanual.html

[10] A. Agarwal, R. Callanan, M. Das, B. Hull, J. Richmond, Sei-Hyung Ryu and J. Palmour "Advanced HF SiC MOS Devices" *IEEE Trans. on Power Electronics and Applications*, 2009.

[11] Application Note on "a new approach to parameter extraction for the spice power mosfet model" by M. Melito, F. Portuese from ST Microelectronics

[12] R. J. Callanan, A. Agarwal, A Burk, M. Das, B. Hull, F. Husna, A. Powell, J. Richmond, Sei-Hyung Ryu, and Q. Zhang, "Recent Progress in SiC MOSFETs and JBS Diodes at Cree", IEEE Industrial Electronics 34th Annual Conference – IECON, pp 2885 – 2890, 10 – 13 Nov. 2008

[13] Infineon IPW90R120C3 CoolMOS Datasheet, Rev 1.0, 2008-07-30.

[14] Infineon IGW25T120 IGBT Datasheet, Rev 2.3, Feb. 2008.

[15] Infineon SGW25N120 IGBT Datasheet, Rev 2.4, Feb. 2008.

[16] F. Bjoerk, J. Handcock, and G. Deboy, "CoolMOSTM CP – How to make most beneficial use of the latest generation of super junction technology devices", Infineon Application Note AN-CoolMOS-CP-01, Version 1.1, Feb 2007.

[17] Numerical Analysis Methods http://en.wikipedia.org/wiki/Numerical_analysis

[18] New Rapson Method http://en.wikipedia.org/wiki/Newton's_method