

International IR Rectifier DESIGN TIPS

DT 92-5

INTERNATIONAL RECTIFIER APPLICATIONS • 233 KANSAS ST. • EL SEGUNDO, CA 90245 • TEL: (310) 322-3331 • FAX: (310) 322-3332

SPICE MODELS FOR MOS-GATED POWER DEVICES

By Donald A. Dapkus II

As PCs become more powerful and design cycle times shrink, more and more designers are looking to complement their breadboarding with PC simulation tools based on SPICE. Unfortunately, SPICE was developed with low power, integrated circuits in mind - the MOSFET in SPICE is the lateral IC type. As such, SPICE offers no built-in features to simulate the subtleties of vertical double diffused power MOSFETs (DMOS) and Insulated Gate Bipolar Transistors (IGBT).

Simplicity is the key to successful SPICE modeling. The end goal is to get a feel for the operation of a circuit, not to develop models that match actual operation 100%. Simulation results that match benchtop results in power electronic circuits can only be obtained by characterizing all the parasitics that occur in the circuit, which is a formidable task.

Most of the limitations of PC-based simulation are removed with the use of more powerful engineering workstations, and the software associated with them. One such software package is Saber provided by:

Analogy, Inc.
9205 S.W. Gemini Dr., Suite D
P.O. Box 1669
Beaverton, OR 97075-1669
(503) 626-9700

THE PROBLEM

The greatest limitation in using SPICE to model power MOSFETs and IGBTs is the effects of the gate-drain capacitance, C_{RSS} or Miller capacitance, which is a highly non-linear function of V_{DS} , especially at low V_{DS} values. Power MOSFET data-sheets have a graph depicting this effect. Unfortunately, in SPICE, this is modeled by a fixed capacitance. Thus, matching modeled switching characteristics with actual device behavior is the fundamental problem in simulating switching circuits containing power MOSFETs.

Much effort has been expended to remedy this situation. Semiconductor companies have provided complex sub-circuit models (see International Rectifier AN-975 which covers several early International Rectifier HEXFET-brand power MOSFETs). Various simulation software companies have developed models for the power MOSFET for use

with their software. Three companies have developed software programs to develop models using only information present on the datasheet. International Rectifier no longer develops SPICE models for its products, however, using one of these programs, it is possible to develop a model for virtually any HEXFET. Some of the programs support power IGBTs (Insulated Gate Bipolar Transistor) in addition to power MOSFETs.

Simulation of IGBTs additionally suffers in that SPICE has no built-in IGBT model, so it is necessary to use an equivalent circuit which consists of a MOSFET input, and bipolar output stage.

TOOLS AVAILABLE

This Design Tip focuses on simulation programs currently being offered by three companies (a list of addresses and phone numbers is at the end of this Design Tip). The three software packages have certain basic similarities - they all use a graphical interface for schematic capture, as well as for analysis of the simulation results. The simulation section of all three packages is based on SPICE. All three packages provide the option of developing custom models for power MOSFETs using information from the datasheet. The main difference among them is the basic philosophy on how to overcome SPICE's limitation of not having built-in vertical power MOSFET (DMOS) models. There are two methods used to compensate for this limitation: 1) simply modifying the .MODEL parameters of the built-in lateral MOSFET model, or 2) using additional circuit elements in a .SUBCKT section.

IMPLEMENTATION

A .MODEL statement is used to input physical characteristics into the simulation of the MOSFET, diode, and other semiconductors. The parameters entered modify the equations used by SPICE to calculate the device's current and voltage. An example for the MOSFET is the gate threshold voltage, V_{TO} which determines the minimum gate to source voltage at which current conduction begins.

On the other hand, a .SUBCKT can be compared to a subroutine in a programming language. From the outside, the power MOSFET .SUBCKT looks like a three-terminal device, yet upon closer inspection, the user sees that the .SUBCKT contains the MOSFET, resistors, switches, diodes, dependent sources, etc.

The tradeoffs involved in selecting which method (.MODEL or .SUBCKT) to use include: simulation time, accuracy (especially important is the switching performance), and convergence. The .MODEL method reduces the simulation time (there are fewer nodes), and decreases the chance of non-convergence, at the expense of accuracy, primarily in the switching region. On the other hand, the .SUBCKT method's goal is to compensate for the changing input capacitance (due to changes in V_{DS} and the Miller effect) to improve the model's accuracy. This goal is achieved at the expense of simulation time (due to the increase in the number of nodes), and increases the possibility of non-convergence.

for the entire circuit (100°C being some arbitrarily determined operating temperature).

To overcome the 'typical' problem, it is sometimes possible to get the ratio from the Electrical Characteristics section of the datasheet of the typical and maximum values of the item of interest. This ratio is then used to scale the graph from typical to maximum.

There may be some confusion as to what to do with the fourth node of the SPICE MOSFET statement - the body/substrate node. Since all modern-day power MOSFETs are of the double-diffused, VERTICAL type structure, the body/substrate node is always tied to the source.

All International Rectifier MOS-gated power transistors are either n- or p-channel *enhancement-mode* devices, that is, the devices need positive voltage between gate and source to form a channel for current conduction. When biased at zero volts gate to source, only the leakage current exists between drain and source (I_{DSS}). This is opposite of the depletion-mode MOSFET.

All the model parameters are die-based, regardless of the HEXFET's packaging. Thus, if trying to model a standard HEXFET not listed in the library, it is often possible to find an existing device model that uses the same die. By tweaking the listed model (possibly package inductance and other parameters), it is possible to develop a model for the part number in question. The simplest method of determining which two packaged devices use the same

die is to compare the $R_{DS(on)}$'s, $V_{(BR)DSS}$'s (breakdown voltage, also known as BV_{DSS}), and $V_{GS(th)}$'s.

It is possible to develop a model for a logic-level HEXFET (IRL...) from a standard HEXFET (IRF...) by dividing TOX by two, changing the V_{TO} to a lower value, and modifying the value of E_{OX} . These three modifications should generate reasonably accurate I_D versus V_{GS} characteristics for the logic-level HEXFET.

CONCLUSION

SPICE was originally devised to simulate low-power integrated circuits, however, its popularity has extended its use well beyond this original application. Power electronic circuits can successfully be simulated if the limitations stated here are kept in mind. For the latest in models, we suggest you contact the simulation companies directly.

SOURCES

IntuSoft
222 W. Sixth St., Suite 1070
San Pedro, CA 90731
(310) 833-0710

MicroSim Corp
20 Fairbanks
Irvine, CA 92718
(714) 770-3022

Spectrum
1021 S. Wolfe Road
Sunnyvale, CA 94086
(408) 738-4387.

Since both methods are standard SPICE syntax, either may be used in *any* SPICE-based simulation tool.

PROGRAMS

IsSPICE is the simulation section of IntuSoft Corporation's ICAP CAE software. As of January 1992, their library contained 160 power MOSFET models of the .SUBCKT variety. They also have a program called SPICEMOD which generates SPICE models for power MOSFETs not in the library from information on the datasheet. They are willing to send models to interested parties, contact them for more information.

PSPICE is the simulation section of MicroSim Corporation's Design Center CAE software. As of January 1992, their power MOSFET library contained 553 device models of the .MODEL variety. Their device characterization routines allow the user to develop models for power MOSFETs not in the library using nothing more than the information present on the datasheet. The user picks values off the datasheet graphs, inputs them into the program which then generates the model. In past revisions of their software, this was called the Parts program.

MicroCap IV is an integrated software program by Spectrum Software that includes schematic capture, simulation, and analysis. As of June 1992, their library contained 315 power MOSFET device models of the .MODEL variety. Also included in the package is a model generation program for power MOSFETs not in the library. Their model

generation program is similar to above in that the user picks values off the datasheet graphs, and inputs them into the program which generates a model.

TIPS ON USING SPICE FOR POWER MOSFET CIRCUITS

Two major design considerations are not addressed using these types of programs: 1) the power dissipation causes an increase in junction temperature, which in turn, affects operating parameters, and 2) parameter distribution is not contained in datasheet graphs which usually represent typical data.

Power devices are usually called to dissipate significant power levels. Many times, the choice of which MOS-gated power device to use is based on the thermal characteristics of the design. Many of the primary parameters change significantly with an increase in junction temperature. For example, the $R_{DS(on)}$ of a power MOSFET typically increases by approximately a factor of two when the junction temperature is at its rated maximum, compared to 25°C.

Possible solutions for this shortcoming is as follows - do a hand calculation to determine the approximate junction operating temperature, and scale $R_{DS(on)}$ by the appropriate value from the $R_{DS(on)}$ versus case temperature graph, using the calculated junction temperature as the case temperature. The threshold voltage also changes with temperature, approximately $-6mV/°C$. This is an iterative process which requires additional simulation effort. This will give better results than simply setting $.TEMP=100$