Electro-Thermal Simulation of Silicon Carbide Power Modules

A. Akturk, N. Goldsman, S. Potbhare
CoolCAD Electronics LLC
5000 College Ave. Ste. 2103, College Park, MD, 20740, USA
akin.akturk@coolcadelectronics.com

***Abstract*— We report on our development of a Silicon Carbide Power System Computer Aided Design Tool to address the need for improved methodologies for developing next generation high efficiency power electronics using Silicon Carbide power devices. The first major achievement is to develop compact models for SiC power MOSFETs and to input these models into CoolCAD's SPICE-type simulator, CoolSPICE (the student version that is for simulating standard SPICE circuits is available online: http://coolcadelectronics.com/coolspice/), to facilitate SiC power circuits design. A second aspect of the work is to extract thermal models for SiC MOSFETs, dies, packages, printed circuit boards, and modules, and then input these thermal models into a thermal simulator for calculating temperature of electronic devices, circuits and modules.**

Keywords*— *Silicon carbide, power electronics, SPICE for power devices, thermal simulations, thermal management

I. Introduction

Power semiconductor devices are ubiquitously used in applications ranging from computer power supplies, consumer electronics, DC-DC and DC-AC converters, electric motor drives, to power converters in satellites and space-stations. There is a constant need for developing new devices and semiconductor materials for lowering switching losses, improving ON state resistance, and increasing the ability to switch higher voltage levels in such applications. Material properties of SiC **[1]** such as large bandgap (3.2eV), high thermal conductivity (4.9 W/K/cm), and high breakdown electric field (3.0 MV/cm), make it uniquely suitable for developing power converters for high-voltage and high-current applications such as electric vehicle (EV), hybrid electric vehicle (HEV) and fuel cell vehicle (FCV) motor drives. CAD tools for device and system design enable manufacturers and engineers to lower system costs by providing a thorough understanding of potential roadblocks in system implementation. Such capabilities will significantly improve reliability, performance and lifetime of power electronics for hybrid electric vehicles.

Characteristics of SiC power MOSFETs such as the presence of interface traps, extremely low intrinsic carrier concentration, incomplete ionization, etc. make it extremely difficult to accurately model and predict the device performance using currently available tools that are focused on Silicon devices. This presents us with a unique opportunity of

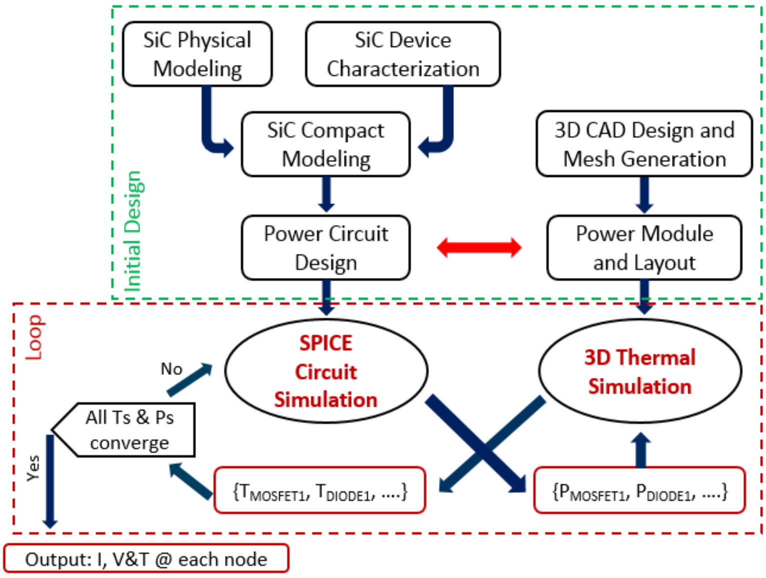

Fig. 1: Proposed system design methodology for high power high temperature SiC power systems.

developing CAD tools that incorporate details of the physics of SiC devices, identify and predict performance degradation mechanisms, and enable engineers to design more reliable and higher performance devices and circuits. Current circuit simulators and SPICE type models do not model the behavior of SiC power devices very well due to lack of common knowledge of the physics of transport in SiC devices. Using the understanding we obtain from our device level physical modeling **[1]** and calibration with experiment, and with mixed-mode (device and circuit) modeling of the power converter systems, we extract meaningful and accurate SPICE-type models for SiC devices, and use them to design robust, reliable and error-free power converter circuits.

This SiC power system design platform, partially depicted in **Fig. 1**, provides a very crucial and economically indispensable link between SiC power device manufacturers (CREE, Infineon, etc.) and commercial (GM, Ford, etc.) and military (General Dynamics, Raytheon, etc) EV, HEV and FCV manufacturers. In the automobile industry, where savings of a few cents per vehicle has a huge impact on profit, extensive use of CAD tools in design of new power electronics is an obvious necessity.

II. METHODOLOGY

With the SiC circuit compact models and simulator, as well as the thermal simulator developed, we work on improving algorithms to integrate electrical and thermal simulators into a self-consistent, electro-thermal SiC power module computer aided design tool that accounts for the effects of temperature induced variations in electrical performance during operation, as shown in **Fig. 1**. Test circuits are being fabricated and used to calibrate the models with experimental data. This new CAD tool will shorten development time for SiC power system components, and significantly reduce design cost by creating virtual design capabilities from the device to the system level that is not achievable for currently available design tools.

A. Silicon Carbide Electrical / SPICE Models

We are developing SPICE models for commercial and development stage silicon carbide diodes, MOSFETs and JFETs, with our main focus on silicon carbide power MOSFETs. We then incorporate these models into our proprietary SPICE simulation suite CoolSPICE for circuit simulation and electro-thermal characterization. Accurate temperature-dependent SPICE modeling of SiC MOSFETs is of utmost importance for the development of an inclusive SiC design platform. To this end, we have performed the initial groundwork for accurate modeling of commercially available SiC MOSFETs from CREE. As an example, we show in **Fig. 2** accurate fits of SiC simulations to measured current-voltage curves from room temperature to 200C for the commercially available CREE MOSFET CMF10120D.

To achieve the fits between simulations and empirical data shown in **Fig. 2**, we have modified the standard BSIM4 model used for CMOS device simulations. These modifications were necessary to resolve the unique silicon carbide electrical characteristics such as relatively high interface trap densities

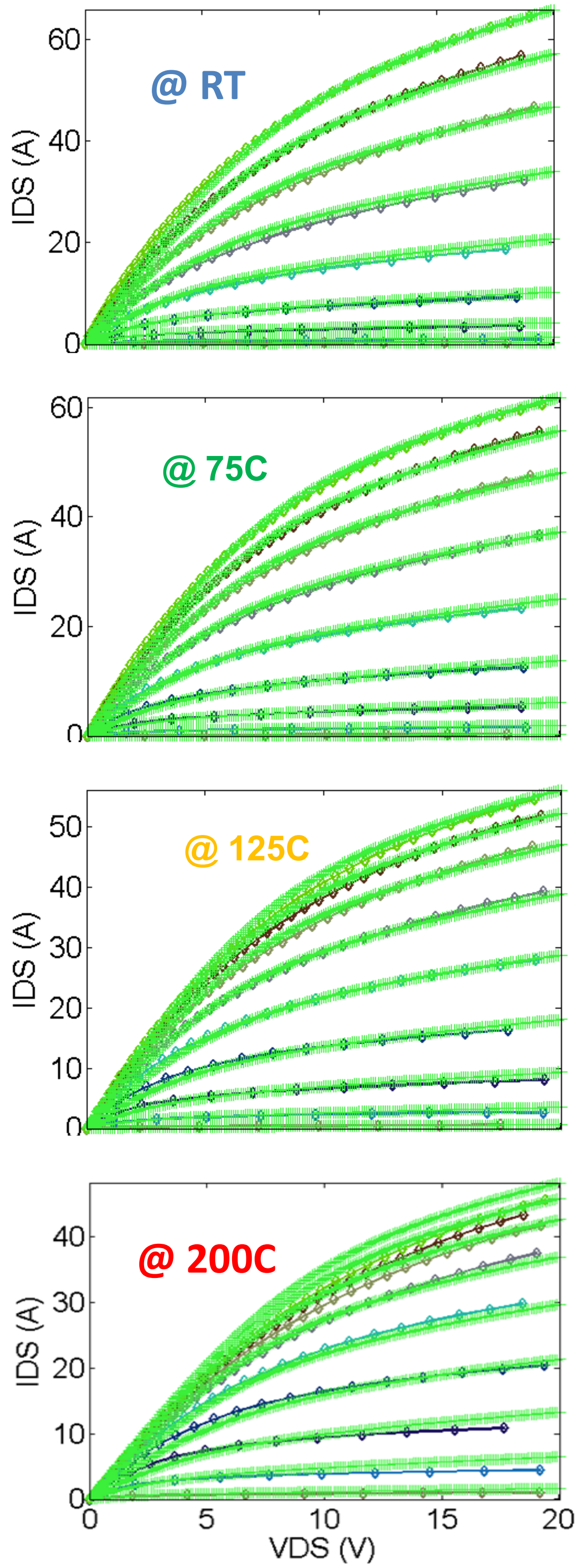

Fig. 2: Example comparisons of simulations obtained using CoolSPICE (green +) and measurements (symbols).

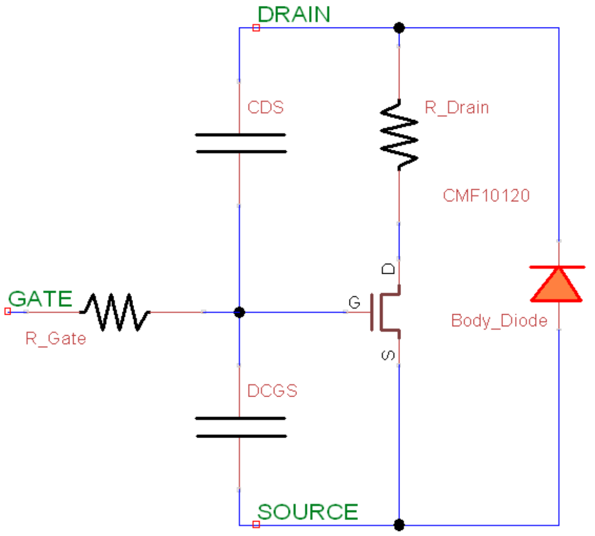

Fig. 3: CoolSPICE SiC Power MOSFET SPICE Model.

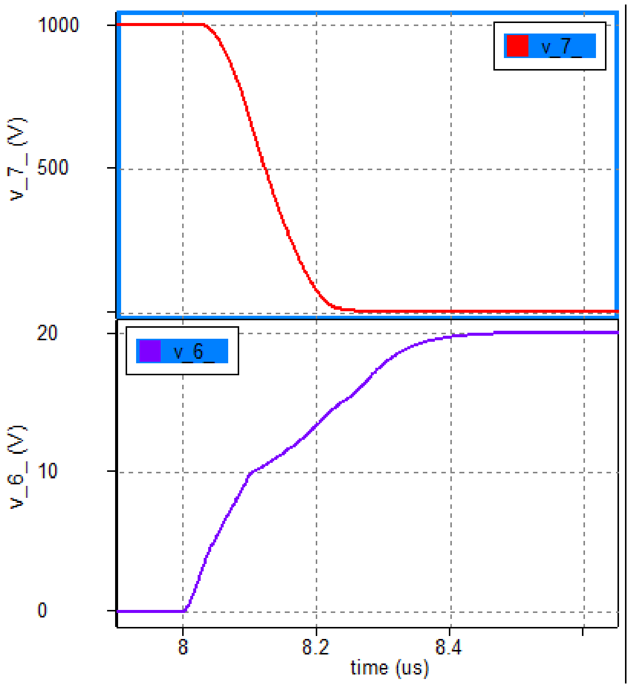

Fig. 4: CoolSPICE simulation example. v_7: SiC power MOSFET output voltage decrease due to gate turn on and a subsequent current flow over a 50 Ohm resistor. v_6: Charging of the SiC MOSFET gate due to a current pulse with 0.1 µs rise time.

and possible oxide traps and charges. Additionally, modifications were incorporated to resolve the temperature-dependent changes in drive currents, i.e. saturation current decreases at VGS=20V, but increases for VGS=16V, as temperature rises. Furthermore, here the experiments were performed using a pulsed high power current/voltage setup in conjunction with a mounting stage whose temperature is set using a temperature controller.

Figure 3 shows the basic subcircuit model we use to simulate silicon carbide power MOSFETs. The curves shown in **Fig. 2** are determined by the MOSFET and the series drain resistance shown above. Drain resistor is a standard temperature dependent SPICE resistor with a room temperature value set to approximately half of the drain-to-source on-resistance. Here the MOSFET, which models the lateral MOSFET / channel of the power MOSFET, is based on a slightly modified version of the standard BSIM 4.5 model. Even though BSIM is developed for silicon MOSFETs, and cannot discern silicon carbide device subtleties, our access to BSIM model equations enable us to incorporate material and temperature dependent silicon carbide details into the BSIM code. Besides obviating the need to develop a complete SPICE MOS model, slight modification of existing BSIM equations also inherits the well behaving numerical convergence characteristics of the BSIM equation set.

In addition to the DC current-voltage characteristics, we also fit simulated capacitance-voltage, as well as breakdown characteristics to those measured. To this end, we disable the internal drain-to-gate capacitances incorporated into BSIM while conserving charge between terminals, and instead use an external behavioral capacitance. This models the neck region of a power MOSFET, which is similar to a MOSCAP structure in the off state. Additionally, we model gate-to-source capacitance using the internal BSIM capacitances, as well as an external behavioral capacitance to compensate for small differences between the simulated and measured curves. The drain-to-source capacitance is modeled by the junction capacitance of the body diode, which is also used to model the device breakdown, as well as reverse recovery time and drain-to-source shortening under negative VDS bias. Lastly, we use a standard SPICE resistor to model the gate resistance, which affects charging time of the gate insulator. **Figure 4** shows the gate charging and subsequent drain voltage decrease, when a current source in parallel with a resistor is connected to the gate and pulsed with a 0.1 µs rise time. Here, there is a 50 Ohm resistor between a 1000V source and the device drain.

B. Actual Layout Driven Thermal-Electrical Power Module Simulations

One of the most important features of our CoolSPICE simulator is that it incorporates new algorithms to couple thermal and electrical simulations through the use of a Finite Element Method (FEM) software called Elmer **[2]**. The user starts with a schematic and 3D model for their circuit. The CoolSPICE software couples with the FEM software to characterize the heat influence of each desired device. This characterization allows the SPICE software to quickly converge on a temperature power relationship for each transistor without running a FEM simulation each time. Using this device characterization simulation, the CoolSPICE software can then compute accurate thermal simulations even with parameter and schematic changes, so long as the physical layout position of each transistor is not fundamentally changed.

1) The first step is the schematic entry and 3D model generation. The 3D model can be drawn in a variety of standard 3D graphics software, where each part is drawn and then brought together as an assembly.

2) After drawing the model, it is exported into a file to be turned into a mesh for simulation. To demonstrate the software abilities with a practical example, we started with an already fabricated power conversion module to create the accompanying 3D model, as shown in **Fig. 5**.

3) After CAD drawings and thermal mesh generation, we switch our focus to SPICE. We start with a netlist, in which we tell SPICE what devices we want tracked, and associate it with the accompanying 3D CAD drawing, described above.

4) For these devices, we next determine the SPICE load lines that specify change in power for each device as a function of temperature, as shown below:

$$P = As \times \Delta T + Bs \quad (1)$$

Here, P is power consumed by a device. ΔT is its temperature rise above the ambient. Bs is the power consumed at the ambient temperature, and As shows coupling of temperature effects between circuit elements.

5) We next obtain thermal load lines for the same devices using 3D thermal simulations, as shown below:

$$\Delta T = At \times P \quad (2)$$

Using the thermal lines represented by At, we determine the thermal resistance from a device to ambient as well as the thermal coupling between critical components of the circuit.

6) We then guess temperatures to be used in SPICE for power calculations. The algorithm then enters a loop that continues until it converges onto a temperature and power solution for that power module. Specifically, the body of each transistor acts as a heat source, getting its value in watts from the power dissipated according to CoolSPICE. And CoolSPICE gets the local temperature values from the thermal simulator. After an array of heat source and temperature configurations is examined, the board is fully characterized and these temperatures are fed back to CoolSPICE. This results in a more accurate, device coupled, thermal and power performance simulation. Furthermore, for transient simulations, we keep track of the average power consumed over a time that is thermally long enough for incremental heating. At the same time, we keep marching in time for electrical simulations.

In summary, CoolSPICE provides a unique method to keep track of electrical and thermal characteristics of SiC and silicon power circuits at the device and 3D module levels, which provide feedback to each other to determine transient and steady state heating and electrical characteristics. Such detailed electrical and thermal characterization is important to take full advantage of the nascent silicon carbide devices, and power devices in general.

ACKNOWLEDGMENT

This project is supported by US ARL cooperative agreement W911NF-12-2-0001 and OSD SBIR W911QX-13-C-0189. We thank OSD and ARL, and specifically Drs. Ronald Green and Aivars Lelis at ARL for their support.

[1] S. Potbhare et al., IEEE Trans. on Electron Devices 55(8), 2061-2070 (2008).

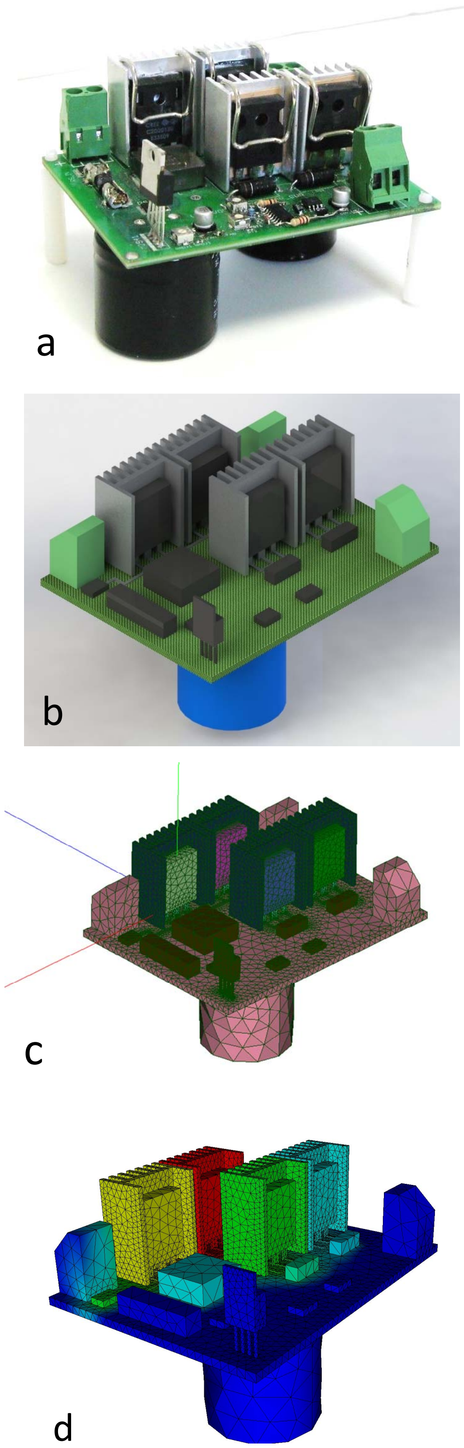

Fig. 5: a) Photograph of a fabricated test dc-to-dc conversion module. b) CAD rendering of its 3D model. c) Mesh built using netgen. d) An example simulation of thermal coupling between the circuit elements.

[2] http://www.csc.fi/english/pages/elmer