Multi-level Electro-Thermal Simulation of Power PCB Electronic Modules for Motor Driving

KONSTANTIN O. PETROSYANTS, IGOR A. KHARITONOV, MIKHAIL S. TEGIN
National Research University "Higher School of Economics",
Moscow Institute of Electronics and Mathematics,
Tallinskaya ul., 34, Moscow, 123458,
RUSSIA

Abstract: -A scheme of automated multi-level electro-thermal modeling of power PCB modules using software tools Comsol at the device construction level, SPICE tool at the circuit level, and Asonika-TM tool at board level was proposed to improve the conventional design approach. The effectiveness of the proposed methodology is demonstrated in the example of electro-thermal analysis of real power MOSFET driver circuit realized on PCB.

Key-Words: - multi-level electro-thermal simulation, PCB power electronic module, SPICE simulation, COMSOL, ASONIKA-TM, junction temperature jumps, thermal network.

Received: January 15, 2023. Revised: Octobert 12, 2023. Accepted: November 11, 2023. Published: December 12, 2023.

1 Introduction

To provide effective thermal management of power electronic systems from the reliability viewpoint, the electro-thermal simulation of the device/circuit system in package placed on PCB with cooling conditions is required, [1], [2]. To realize the electro-thermal simulation flow, two approaches are used:

1) circuit analysis tool (SPICE like) is coupled with 3D thermal numerical simulation tool (AnSYS, [3], Comsol, [4], [5], [6]), for modeling of electro-thermal processes in the 3D construction of the electronic module under test;

2) only the circuit-based tool (SPICE like) is used for electro-thermal simulation of the power PCB modules. The electronic devices are described by the electro-thermal SPICE models and packages/heatsinks are described by the RC thermal networks according to the thermo-electric analogy, [3], [7], [8], [9], [9], [10].

The first approach provides the complete electro-thermal solution for electronic modules. However, its realization is difficult because of detailed description of the 3D PCB module construction and much time spent on numerical simulation of the 3D construction using the finite-element (FEM) method.

The second approach is more understandable and convenient for circuit designers because the approach requires only the circuit simulator, [11]. Its advantage is the possibility of fast analysis of electrical characteristics, power, and temperature.

The acceleration of the motor rotor can cause high current or voltage jumps in output circuit load that increases the power loss and device junction temperature. Therefore, the problem for designers is to predict the junction temperature in the dynamic operation. The main task is to minimize the CPU time of transient response simulation in PCB modules.

Several studies have been done in the direction of realizing the fast electro-thermal technique for dynamic analysis. In, [4], the joint ELDO-COMSOL tools for electro-thermal dynamic simulation of power MOSFET in the package were used. Unfortunately, the electrical circuit of the test module was not considered and the passive device with equivalent power loss instead of a real MOSFET was used for IR thermal measurement.

The thermal Cauer networks of IGBTs, [12], and MOSFETs, [13], were used to minimize the CPU time for the dynamic analysis. Additional work needs to be done to further verify the models with junction temperature measurements.

In, [14], the fast ET simulation model for long real-time thermal simulation of a three-phase IGBT inverter power module was presented. The design technique is original and does not use the standard commercial tools like SPICE, COMSOL, ANSYS, etc. So it is not easy for the engineers to implement the new technique in the real design system for practical use.

The authors of, [15], [16] have analyzed different approaches to electro-thermal design and proposed simple ET SPICE models of power semiconductor transistors in packages to minimize simulation time.

Summarizing the critical analysis of the publications we can conclude that the problem of fast and easy for implementation electro-thermal simulation technique is as usual under consideration. Especially the works with the real practical solutions that were proposed are required.

In this connection, the presented paper describes the effective ET solution scheme for the dynamic behavior of motor driving power module. The complete chain is discussed: 1) compact SPICE ET model of power MOSFET in package; 2) package R_T determination using Comsol solution; 3) coupled SPICE-ASONIKA-TM technique for ET transient analysis of PCB electronic module; 4) thermal models verification by comparison with the results of IR measurement of the module surface temperature; 5) MOSFET junction temperature analysis and probable failure detection; 6) improvement of the module construction to guarantee the reliability of the driver circuit and motor winding of coils.

2 Methodology

To provide an effective electro-thermal analysis of the PCB electronic modules we used multi-level electro-thermal simulation and both of the mentioned approaches with addition of infra-red thermal analysis, [17]. Furthermore, to automize and simplify this procedure we have developed the special software tools ST1, ST2, ST3, [18], which were integrated into the electro-thermal iteration loop in Figure 1.

Our important innovation in improving the electro-thermal design process is the automation of the next processes:

- circuit component powers calculation from the SPICE analysis,

-transfer the component powers from the SPICE tool to the thermal simulation tool,

- temperature values transfer from the thermal simulation stage to the circuit SPICE simulation stage,

- power electronic devices and ICs case thermal subcircuit generation for SPICE electro-thermal simulation.

For the first three tasks the special software tool ST2 was developed. For the last task a special Table with thermal parameters of standard component cases was formed (Figure 1). The table contained standard component case names and corresponding parameters of the case thermal subcircuit. The data in the table were taken from the components datasheets or the universal 3D physical software tool Comsol was used to generate R_{ti}, and C_{ti} elements when the required thermal parameters were not presented in component data sheets. The required thermal parameters were selected from the Table using developed ST1 tool.

As a result the multi-level design methodology, using Comsol at device construction level, SPICE at the circuit level, and ASONIKA-TM at board level was developed.

The tools Comsol, [19], and SPICE, [20], are well known.

ASONIKA-TM software tool, [21], developed at Moscow Institute of Electronics and Mathematics was used for PCB thermal simulation. The tool has the following advantage: very quick calculation of PCB thermal mode due to using analytical thermal models for components thermal and cooling parameters and numerical simulation for analysis of temperature distribution along PCB surface.

2.1 Special Software Tools Developed for Processing Simulators Data and Transferring Information between Simulators

The details of the developed electro-thermal simulation scheme are the next.

The tool ST2 (at the right in Figure 1) provides, [18]:

- automated, user-controlled calculation of the circuit element powers in SPICE tool;

- component power values transfer from SPICE to thermal analysis packages (Comsol, ASONIKA-TM etc.);

- component temperature values transfer back from the thermal analysis packages to the SPICE analysis tool.

The software tool ST1 (at the left in Figure 1) provides automated generation of thermal subcircuits for the power component cases and packages, [18], as was described upper.

The tool ST3 (at the right in Figure 1) provides automation of power- temperature transfer between electrical and thermal subcircuits during SPICE electro-thermal simulation (it will be illustrated later).

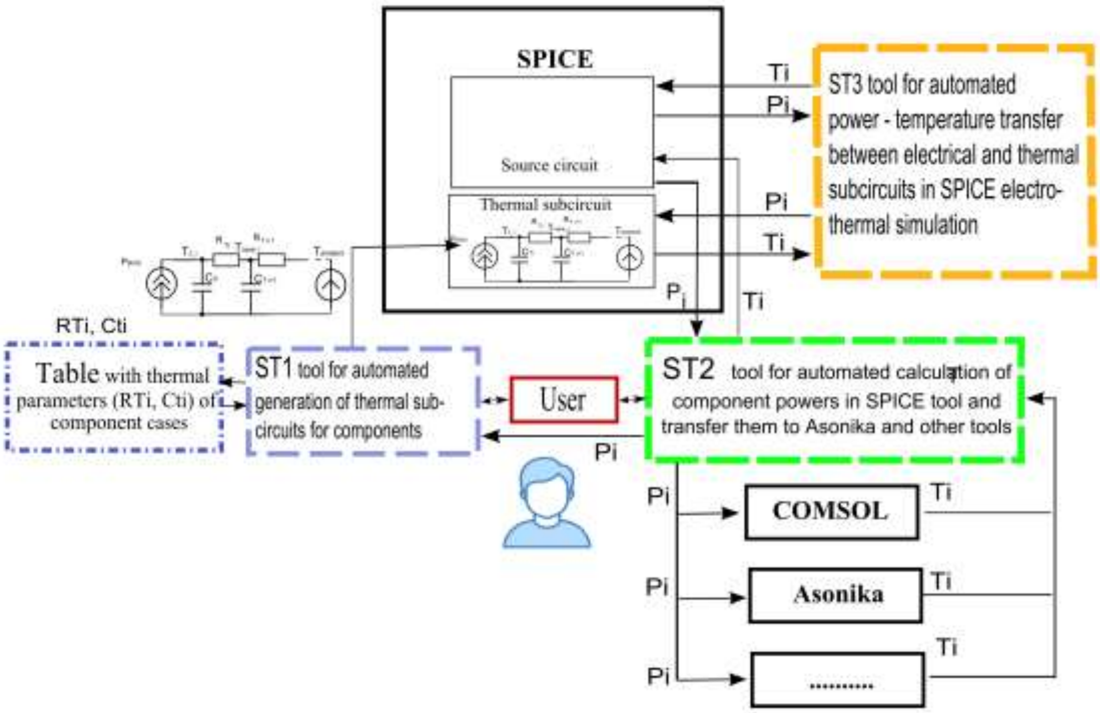


Fig. 1: Scheme of software tools interactions for electro-thermal modeling and simulation of power circuits on PCBs used in the present paper

3 Joint Spice, Comsol, Asonika-TM Simulation and Analysis of Electro-Thermal Modes of Stepper Motor Control Circuit

The described scheme of interactions of software tools in Figure 1 was used by us for electro-thermal modeling and simulation of power PCB modules and analysis of power MOSFETs junction temperature jumps for a power 4-phase stepper motor control circuit realized on PCB (Figure 2). The circuit is a bridge with 4 IRFB4615 DMOS transistors for one phase control, [22]. The four output DMOS transistors were placed on OMNI-UNI-30-50-D heat sink, [23], with thermal resistance R_θ=4.06 °C/W under natural convection conditions.

The values of IRFB4615 channel resistance $R_{ds\ on}$= 35 Ohm and supply voltage 24 V were used.

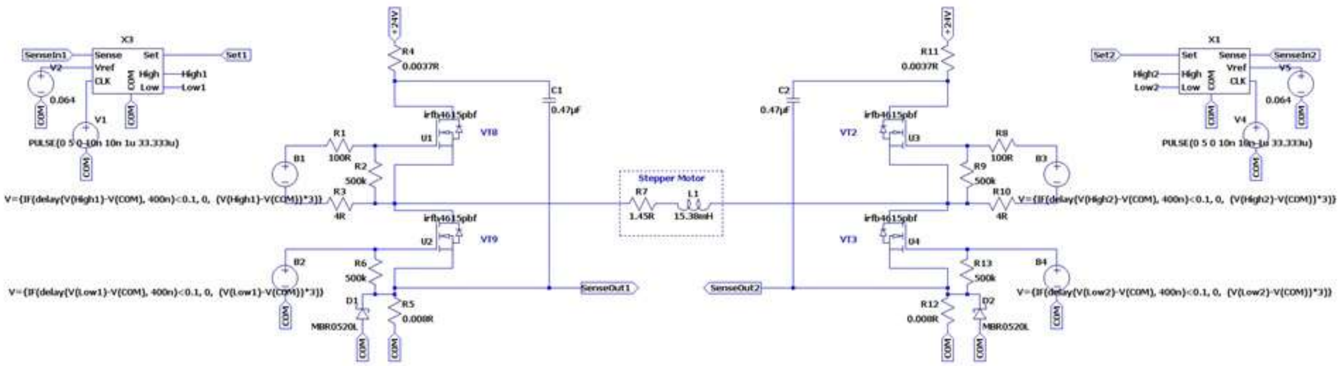
3.1 Electro-thermal Simulation of the Step Motor Driver Circuit using the SPICE Analysis Package

For the electro-thermal simulation of DMOSFET temperature jumps of the mentioned driver circuit using the SPICE simulator, the electro-thermal SPICE model for IRFB4615 DMOSFET has been built. The model consisted of two interconnected parts:

- the electrical part for the IRFB4615 DMOSFET with temperature dependent parameters for the threshold voltage, mobility, and MOSFET channel R_{on} and

- the thermal subcircuit for the IRFB4615 DMOSFET case with the current source describing the dissipated power of the DMOSFET .

Parameters of the electro-thermal model were defined using measured static and dynamic characteristics of IRFB4615 DMOS FET with account for the measured MOSFET temperature.



(a)

(b)

Fig. 2: Simplified schematic (a) and PCB realization (b) of stepper motor driver circuit

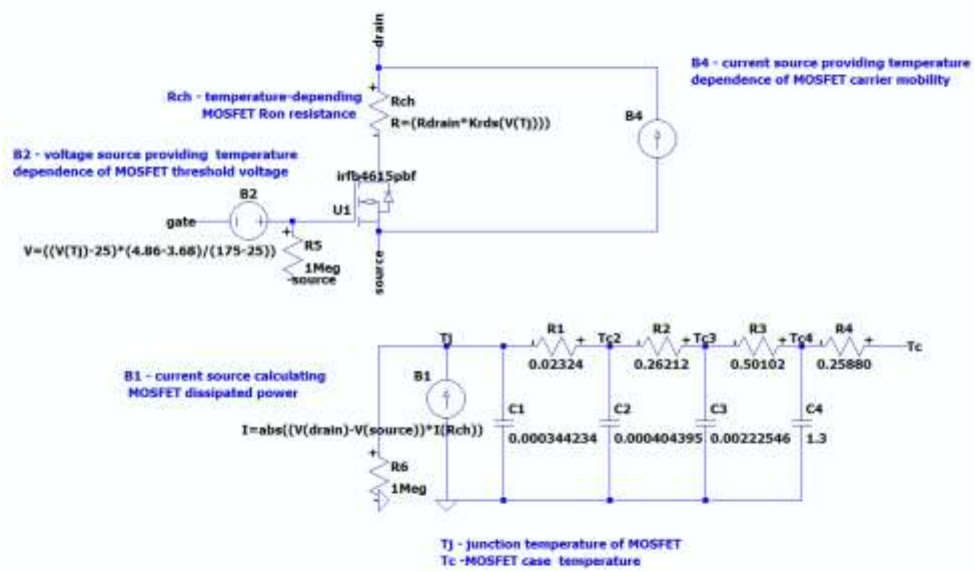


Fig. 3: Electro-thermal model of IRFB4615 DMOSFET

The hardest (from the electro-thermal point of view) mode of the motor driver circuit is the motor holding position mode – the maximum constant DC current is consumed by the motor in this mode. Furthermore, this maximal current is provided by the same MOSFET in this mode. So the motor driver circuit was simulated and analyzed by us in this mode of work.

Our measurements showed that the driver circuit reaches a stationary thermal mode for about 50 minutes. Electro-thermal simulation of the driver circuit with SPICE tool using the original electronic circuit and with additional thermal sub-circuit (Figure 4) did not allow to reach a stationary thermal mode, because the time step in SPICE simulation was about 1 µs (due to the operating frequency of the circuit 30 kHz) , but the time constant for the thermal part was many orders larger (due to large values of thermal capacitances of transistor case and heat sink). This problem with very large differences of time constants for electronic and thermal parts is well known, [24]). We applied the following simple approach, accelerating electro-thermal calculation.

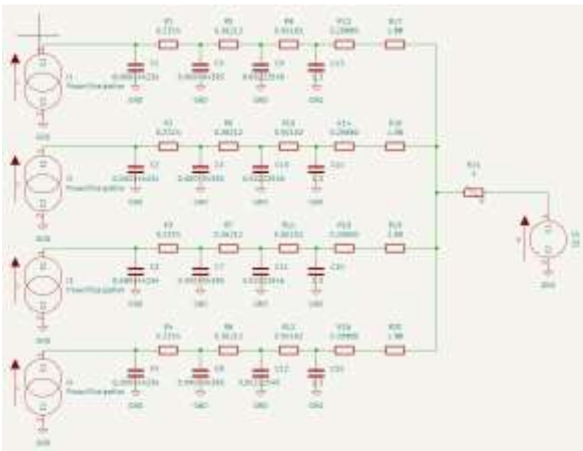


Fig. 4: Thermal subcircuit for 4 output power transistors mounted on heatsink

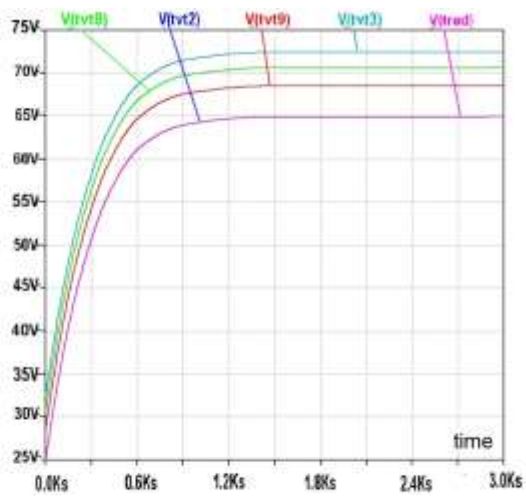


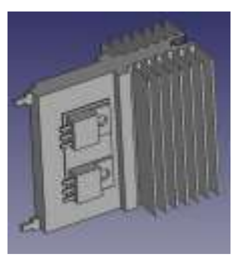
Fig. 5: SPICE simulated temperature-time dependences for power transistor cases in the circuit Figure 2 after the 4th iteration (the motor is in the holding position mode)

The process of temperature and power transfer was repeated automatically several times (using ST3 tool). This iterative process was finished when small changes of the power transistor temperature values (0.5 ºC) were reached from one iteration to the next iteration. In our case it took 4 iterations. Figure 5 shows SPICE simulated temperature dependences of output power transistors cases on time after the 4-th iteration. The y-axis displays power transistor (tvt8, tvt2, tvt9, tvt3) cases and heatsink (trad) temperatures in degrees Celsius as the voltage's values. The x-axis displays time of the driver working in ksec after its switched on.

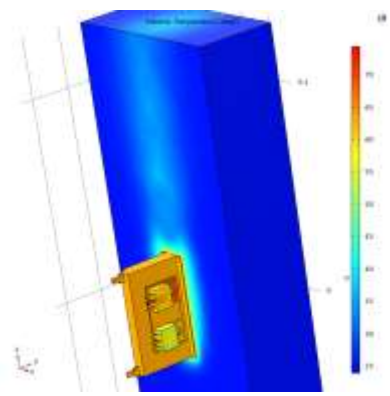
3.2 Electro-thermal Simulation of the Motor Driver Circuit on PCB using SPICE and Comsol Packages

To check the correctness of the thermal subcircuit (Figure 3 and Figure 4) electro-thermal simulation of the motor driver circuit on PCB was performed using SPICE and Comsol packages/ The power values of the output transistors (VT2, VT3, VT8, VT9) of the circuit Figure 2 were automatically determined from a SPICE analysis of the circuit when a stationary thermal regime was reached (see previous paragraph).

Figure 6 (a) shows a 3D image of the power transistors placed on the heatsink package in Comsol tool, and Figure 6 (b) shows Comsol simulated temperature distribution in the power transistors and heatsink. The temperature values of power transistors were close to the values obtained from SPICE analysis of the circuit. A detailed comparison of thermal analysis results obtained using different simulation tools will be presented below.



(a)

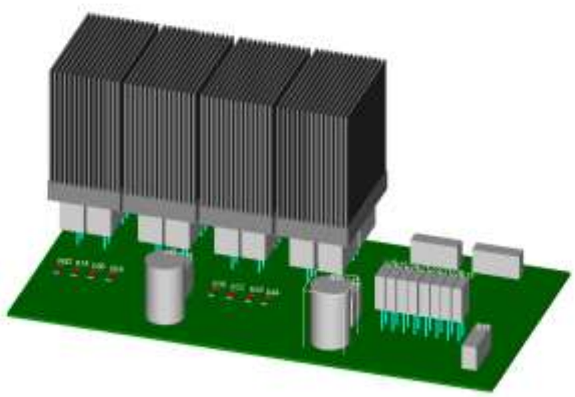


(b)

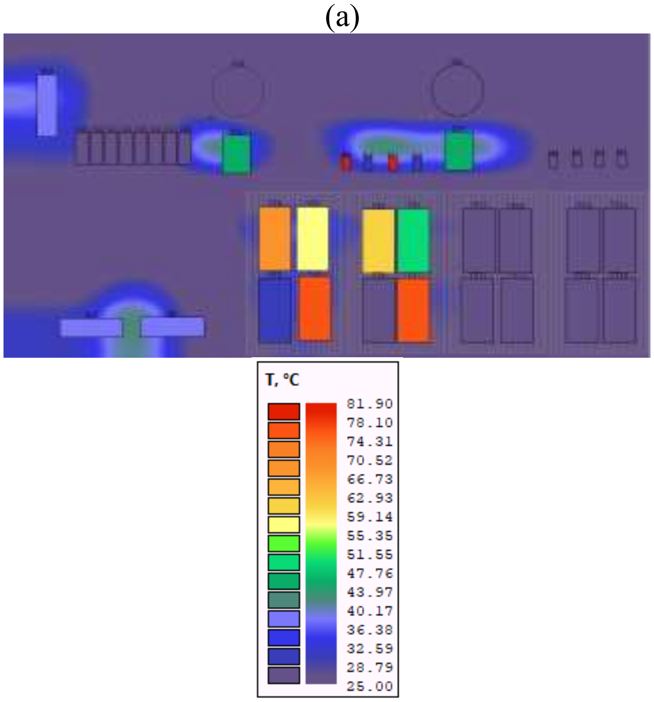
Fig. 6: 3D view (a) and thermal simulation results of stepper motor driver power MOSFETs (1 phase) (b) using Comsol software tool

3.3 Electro-thermal Simulation of the Motor Driver Circuit on PCB using SPICE and ASONIKA-TM Software Packages

ASONIKA-TM tool allows quick (few seconds) thermal analysis of circuits realized on PCB with account for real PCB construction features: layer thicknesses, PCB and electronic component sizes, convection conditions and more details . The power values of the transistors (VT2, VT3, VT8, VT9) were determined from the SPICE analysis of the operation of the circuit when a stationary thermal mode was reached (as described in the previous section) and were transferred from SPICE to Asonika-TM tool, (using ST2 tool) and the temperature values of power elements were transferred back to SPICE tool using the same software tool, at the right in Figure 1).



(a)



(b)

Fig. 7: 3D view (a) and thermal simulation results (b) of all 4 phases of stepper motor driver using Asonika TM software tool

Figure 7 (a) presents a 3D image of power transistors placed on heatsink (for all 4 phases) in Asonika-TM software package. Powers were supplied to 2 bridges of 4. Figure 7 (b) presents the simulated temperature distribution in power transistors and PCB.

3.4 Infrared Thermal Analysis of the Thermal Mode of the Stepper Motor Driver Circuit

Flir A-40 thermal imaging camera was used for study of the thermal model of the stepper motor control circuit and for verification of the simulation results. The obtained temperature distribution on power transistors (of one phase) in a steady state thermal mode and the motor in the holding position mode is presented in Figure 8.

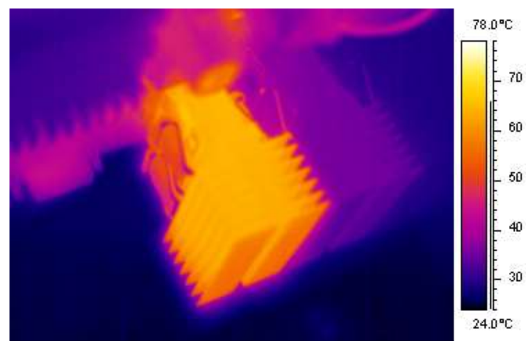


Fig. 8: Temperature distribution in 1 phase of stepper motor driver (Figure 2) measured with Flir A40 thermal imaging camera (in a steady state thermal mode when the motor is in the holding position mode)

3.5 Analysis of the Obtained Results

Table 1 presents the comparison of the power transistors temperature values (circuit Figure 2) obtained by the IR measurement method and by the simulation methods with the different tools (SPICE, Comsol и Asonika-TM).

It is seen that the results obtained using the different thermal analysis packages are quite close to each other and are confirmed by the results of thermal IR camera imaging that confirms the correctness of our electro-thermal simulation process.

3.6 Analysis of the Power MOSFETs Temperature Jumps

The most important parameter of PCB driver module directly related to thermal condition is the reliability. From this point of view, the motor driver module was analyzed in the most hard electro-thermal mode: ambient temperature 40°C, maximal value of IRFB4615 channel resistance $R_{ds\ on}$= 39 mOhm, supply voltage 28 V and the stepper motor has to be in the holding position mode.

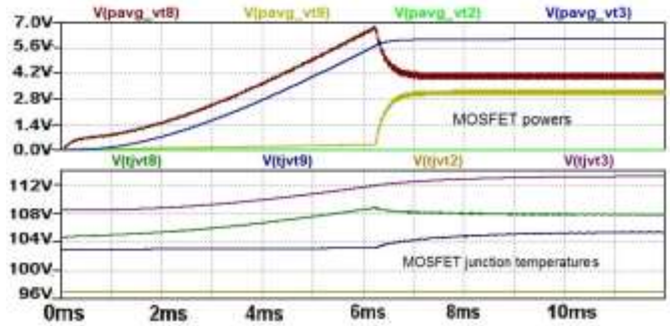
Table 1. comparison of the power transistor temperature values IR measured and simulated with SPICE, Comsol and Asonika-TM tools

Transistor / heat-sink	Power, W	T, °C (IR measured)	T, °C (Spice)	T, °C (Asonika)	T, °C (Comsol)
VT8	3,04	72,1	70,7	66,76	69,2
VT9	1,91	72,7	68,6	55,35	67,2
VT2	0	67,7	64,9	52,37	63,4
VT3	4	74,5	72,5	77,84	70,9
Heat-sink	0	67,3	64,9	-	63,9

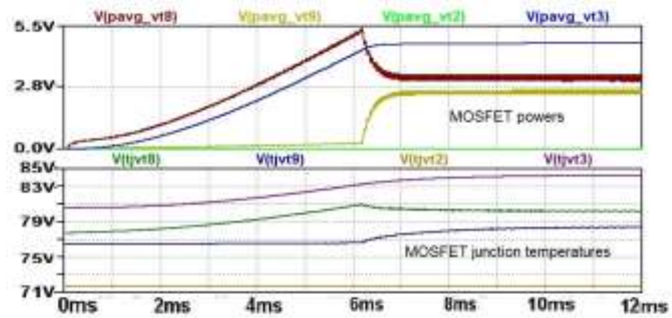
For comparison, the conventional operational mode was simulated with the following PCB module parameters: Tamb=25°C, $R_{ds\ on}$= 35 mOhm, supply voltage 24 V and the motor is in the holding position mode too.

Since the MOSFET parameters and power depend on junction temperature value it is necessary to use the developed and verified the electro-thermal SPICE models for power MOS FETs to get the maximal temperature estimation.

SPICE simulated temperature values of IRFB4615 MOSFET for the most hard electro-thermal mode are presented in Figure 9 (a). At the upper picture, the y-axis displays DMOS FETs powers (pavg_tvt8, pavg_tvt2, pavg_tvt9, pavg_tvt3) as the voltage values. At the bottom picture, the y-axis displays power transistor (tvt8, tvt2, tvt9, tvt3) cases and heatsink (trad) temperatures in digress Celsius as the voltage values. The x-axis displays the time of the driver's work in ksec after it was switched on. The circuit entered the holding mode at 6 msec. It is seen from the Figure 9 (a) (V(tjvt8) curve) that VT3 MOSFET junction maximal temperatures were nearly 115°C – it could result to the MOSFET failure.



(a)



(b)

Fig. 9: SPICE simulated DMOSFET powers and DMOSFET junction temperature jumps for the hardest thermal mode (a) and for enhanced cooling conditions - the larger heat sink with R_θ=2.5 °C/W (b).

To guarantee more reliability of the driver circuit and to enhance MOSFET cooling conditions, a larger heat sink with less thermal resistance R_θ=2.5 °C/W (instead of 4.06 °C/W) under natural convection conditions was used Figure 9 (b) presents SPICE simulated results for this case. It is seen that the DMOSFET junction maximal temperature (top graphs on Figure 9 (b)) decreased to 84°C, which enhances reliability.

4 Conclusion

1. The scheme for multilevel joint electro-thermal design of power electronic circuits on PCBs using software tools Comsol, SPICE, ASONIKA-TM and IR thermal analysis was realized and verified.

2. The special software tools and data table with the power component thermal properties including the names of the component cases and the parameters of their thermal subcircuit (R_{Ti}, C_{Ti}) were developed to atomize and simplify the forming thermal models for power components.

3. The significant acceleration of electro-thermal SPICE-ASONIKA-TM simulation process was achieved (more than 10 times in comparison with the conventional electro-thermal simulation technique using coupled SPICE-Comsol tools).

4. The effectiveness of the proposed methodology was demonstrated at the example of real PCB construction for motor driver with power MOSFETs.

5. The results obtained using the different thermal analysis packages were quite close to each other and were confirmed by the results of thermal IR camera imaging.

6. The probable thermal failures were detected and the ways to reduce the MOSFET junction temperature were proposed.

References:

[1] Rencz, M.; Szekely, V.; Poppe, A.; Courtois, B., "Cosimulation of dynamic compact models of packages with the detailed models of printed circuit boards," *27th Annual IEEE/SEMI International*, San Jose, CA USA, vol., no., pp.285,290, 2002 doi: 10.1109/IEMT.2002.1032768.

[2] T. Shinoda, "Three Thermal Simulation & Test Innovations for Electronics Equipment Design. Electronics Cooling", 2019, [Online]. https://www.electronics-cooling.com/2019/06/three-thermal-simulation-test-innovations-for-electronics-equipment-design/ (Accessed Date: April 15, 2023).

[3] T. Hauck, W. Teulings, W. Teulings, E. Rudnyi, "Electro-Thermal Simulation of Multi-channel Power Devices on PCB with SPICE", *15-th Thermal Investigations of ICs and Systems International Workshop, (THERMINIC 2009)*, Leuven, Belgium, pp.124-129, 2009.

[4] G. De Falco, M. Riccio, G. Romano, L. Maresca, A. Irace and G. Breglio, "ELDO-COMSOL based 3D electro-thermal simulations of power semiconductor devices", *2014 Semiconductor Thermal Measurement and Management Symposium (SEMI-THERM)*, San Jose, CA, USA, pp.35-40, 2014.

[5] C. T. Kao, A. -Y. Kuo and Y. Dai, "Electrical/thermal co-design and co-simulation, from chip, package, board to system,", *2016 International Symposium on VLSI Design, Automation and Test (VLSI-DAT)*, Hsinchu, Taiwan, pp.1-4, 2016.

[6] Yi. Jia , F. Xiao, Ya. Duan , Yi Luo , Bi. Liu , Y. Huang. "PSpice-COMSOL-Based 3-D Electrothermal–Mechanical Modeling of IGBT Power Module, *IEEE Journal of Emerging and Selected Topics in Power Electronics*, vol. 8, No. 4, pp.4173-4185, 2020.

[7] K. Mohamad Ali, R. Sommet, S. Mons, E. Ngoya. "Behavioral Electro-thermal Modeling of Power Amplifier for System-Level Design", *Intern. Workshop on Integrated Nonlinear Microwave and Millimeterwave Circuits (INMMIC-2018)*, Brive La Gaillarde, France, pp.1-3, 2018.

[8] L. Codecasa, V. d'Alessandro, A. Magnani, A Irace. "Circuit-Based Electrothermal Simulation of Power Devices by an Ultrafast Nonlinear MOR Approach", *IEEE Transactions on Power Electronics*, Vol.31, No. 8, pp.5906-5916, 2016.

[9] Dan Eddleman, "LTspice: SOAtherm Support for PCB and Heat Sink Thermal Models" , [Online]. https://www.analog.com/en/technical-articles/ltspice-soatherm-support-for-pcb-and-heat-sink-thermal-models.html (Accessed Date: April 1, 2023). Tutorial: ngspice electro-thermal simulation, [Online]. http://ngspice.sourceforge.net/ngspice-electrothermal-tutorial.html (Accessed Date: March 3, 2023).

[10] Byron Blackmore, "Electrothermal circuit simulation enabled by VHDL-AMS thermal netlists", [Online]. https://blogs.sw.siemens.com/simcenter/electrothermal-simulation-thermalnetlist-vhdl-ams-flotherm/ (Accessed Date: December 8, 2023).

[11] Byron Blackmore, "The future of thermal design – earlier electrothermal analysis", 2020, [Online]. https://blogs.sw.siemens.com/simcenter/future-of-thermal-design-electrothermal-modeling-bci-rom-vhdl-ams/ (Accessed Date: March 1, 2023).

[12] Kojima, T., Nishibe, Y., Yamada, Y., Ueta, T., Torii, K., Sasaki, S., & Hamada, K. (2006, June). Novel electro-thermal coupling simulation technique for dynamic analysis of HV (hybrid vehicle) inverter. In *37th IEEE Power Electronics Specialists Conference*, Jeju,

Korea (South), 2006, pp.1-5, doi: 10.1109/pesc.2006.1712076.

[13] Van Cappellen, L., Deckers, M., Alavi, O., Daenen, M., & Driesen, J. (2022, September). A Real-time Physics Based Digital Twin for Online MOSFET Condition Monitoring in PV Converter Applications. In *28th International Workshop on Thermal Investigations of ICs and Systems (THERMINIC)*, Dublin, Ireland, pp. 1-4, 2022. doi: 10.1109/THERMINIC57263.2022.9950636.

[14] Zhongfu Zhou, M.S. Kanniche, S. G. J. Batcup, Petar Igic, "High-speed electro-thermal simulation model of inverter power modules for hybrid vehicles", *IET Electric Power Applications*, vol.5(8), pp.636-643, 2011.

[15] Ma K.and Blaabjerg F., "Multi-timescale modeling for the loading behaviors of power electronics converter", presented at the *IEEE Energy Conversion Congress and Exposition (ECCE 2015)*, Montreal, Quebec, Canada, pp.5749–5756, 2015.

[16] J. Urkizu, M. Mazuela, A. Alacano, *et al*, Electric Vehicle Inverter Electro-Thermal Models Oriented to Simulation Speed and Accuracy Multi-Objective Targets, *Energies*, MDPI, vol.12(19), pp.1-29, 2019.

[17] Petrosyants K.O., Kharitonov I.A., Ryabov N.I., "Electro-Thermal Design of Smart Power Devices and Integrated Circuits", *Advanced Materials Research*, vol.918, pp.190-194. 2014.

[18] Igor Kharitonov, Gleb Klopotov, Valentin Kobyakov, Michael Tegin, Evelina Silchenko, Konstantin Ivlev, Dmytry Loktionov, "Extension of the capabilities of SPICE analysis tools for electro-thermal simulation of power electronic circuits", *2022 Moscow Workshop on Electronic and Networking Technol. (MWENT)*. Moscow, Russia. pp.1-5, 2022.

[19] Simulate real-world designs, devices, and processes with multiphysics software from COMSOL, [Online]. https://www.comsol.com (Accessed Date: December 4, 2022).

[20] The Spice Page (by Jan M. Rabaey), [Online].http://bwrcs.eecs.berkeley.edu/Classes/IcBook/SPICE/ (Accessed Date: June 20, 2021).

[21] Asonika Tool, [Online]. https://asonika-online.ru (Accessed Date: October 18, 2022).

[22] IRFB4615PbF datasheet, International Rectifier, 2008, [Online]. https://www.alldatasheet.com/view.jsp?Searchword=IRFB4615PBF&sField=4 (Accessed Date: December 8, 2023).

[23] Wakefield-Vette OMNI-UNI-30-50-D, [Online]. https://eu.mouser.com/ProductDetail/Wakefield-Vette/OMNI-UNI-30-50-D (Accessed Date: February 3, 2023).

[24] Ceccarelli L., Kotecha R. M., Bahman A. S., Iannuzzo F., Alan H.. "Mission-Profile-Based Lifetime Prediction for a SiC MOSFET Power Module Using a Multi-Step Condition-Mapping Simulation Strategy", *IEEE Transactions on Power Electronics*. vol.34, pp.9698-9708, 2019.

Contribution of Individual Authors to the Creation of a Scientific Article (Ghostwriting Policy)

- Konstantin Petrosyants - the methodology of electro-thermal analysis and simulation of power circuits on PCB development and convergence analysis of Section I, Section II.
- Igor. Kharitonov was responsible for the scheme of software tools interactions for electro-thermal modeling and simulation and executed the IR experiments of Section III.
- Mikhail S. Tegin was responsible for the SPICE, COMSOL simulation and simulation results analysis of Section III.

Sources of Funding for Research Presented in a Scientific Article or Scientific Article Itself

The study was carried out within the strategic project "Digital Transformation: Technologies, Effects and Performance", part of the HSE University "Priority 2030" Development Programme".

Conflict of Interest

The authors have no conflicts of interest to declare.