Mixed Electrical-Thermal and Electrical-Mechanical Simulation of Electromechatronic Systems Using PSPICE.

K.O. Petrosjanc, P.P. Maltcev

Moscow University of Electronics and Mathematics,
3/12, Bolshoy Vuzovsky per., 109028, Moscow, RUSSIA

Abstract *- The design methodology and technique is presented to expand the power of commercial SPICE to simulate mixed electrical-thermal-mechanical microsystems, consisting of motors being driven by smart power ICs. New electro-thermal and electro-mechanical models of mixed system elements are developed and included into SPICE model library as subcircuits. Practical examples are discussed illustrating the possibilities of developed techniques and software tools.*

Introduction

Electromechatronic systems are the basic elements of robotics, automotive, avia, naval and space navigation, computer peripherals, telecommunication products. In this paper we consider the electromechatronic system as motor-drive system which consist of electronic and electromechanical parts (see fig. 1). From this point of view the system design is divided in two parts: for the smart power IC driver and for the motor being controlled.

Mixed electrical-thermal IC driver simulation.

The smart power IC is the complex microelectronic one chip system which consists of the high power output bipolar or DMOS transistors or circuit driving the motor windings and middle/low power circuits or elements which carry out the functions of sensoring, control, protection, testing, A/D or D/A convertion, memorizing (see fig.1). For this IC which is realized by monolith or hybrid technologies occur the following specificities:

- arrangement of high power, middle/low power, sensor elements on the chip;
- use of the circuit elements with different physical operating mode: BJTs, FETs, MOSFETs, DMOSs, CMOSs, sensors, specific multifunctional elements;
- operating with high current, voltage, temperature conditions for power elements and high linearity and stability for sensor and analog elements;
- strong electrical and thermal intercommunion between the elements on the chip.

The thermal effects is the main limiting factor in stationary and transient smart power IC operation. Accurate thermal IC driver modeling is necessary to: 1)obtain the realistic evaluation of the reliability of circuit from electro-thermal standpoint; 2) stabilization of electrical characteristics by reducing temperature gradients; 3) reduction of thermal/mechanical chip deformation and stress.

As a result the smart power IC driver must be considered as a mixed electronic-thermal microsystem. The CAD of such system can't be made by the traditional way with commercial software tools PSPICE, PCAD at el., which give good choice to simulate the system consisting especially of electronic components.

The high power dissipation and the strong electro-thermal intercommunions induce high local temperatures of the elements and change there electrical regimes. This situation is not forced in standard commercial PSPICE where the temperature is used as an external parameter identical for all elements on the chip.

To overcome this problem we developed the compact electro-thermal models of middle/low power BJT, high power BJT, MOSFET, CMOS, high power DMOSFET and passive elements: resistors and capacitors. These models were included into PSPICE model library as subcircuits.

The electro-thermal BJT and MOSFET (DMOSFET) models consist of electrical and thermal parts are shown in fig. 2. They were developed for a real IC construction fig. 3 and taken into account a thermal interaction between the neighboring elements on the chip. We have added one more local temperature node TQ_i to the traditional electrical voltage nodes (e, b, c) of BJT and (g, d, s, sub) of MOSFET.

In the electrical subcircuit fig.2 the internal transistor is described by standard BJT or MOSFET PSPICE model and the current source GTQ_i is governed by local temperature TQ_i. In the thermal subcircuit the voltage source VT0 represents the ambient temperature T_0; the current source GPQ_i is governed by the total dissipated electrical power of the transistor PQ_i; RTQ_i and CTQ_i - thermal resistance and capacitance of the transistor Q_i from the ambient to the bottom of the chip; RTQ_iQ_j - mutual thermal resistance between transistors Q_i and Q_j on the chip. The thermal subcircuit is equal to the discrete representation of classic heat flow equation which describe the 3D-temperature distribution T(x,y,z) inside the real IC structure

$$\nabla\left[\lambda(T)\nabla T(x,y,z,t)\right]=\rho\cdot C\frac{\partial T(x,y,z,t)}{\partial t} \tag{1}$$

with the boundary condition on the top of the chip

$$\lambda(T)\frac{\partial T}{\partial z}=\begin{cases}P & \text{(inside element area)}\\ 0 & \text{(outside element area)}\end{cases} \tag{2}$$

and at the bottom of the chip

$$T_b(t)=P_T R_T+T_0-R_T C_T\frac{dT_b(t)}{dt}, \tag{3}$$

where λ, ρ, C - thermal conductivity, density and specific heat of the chip material respectively; P, P_T - power density and total dissipated power of the device; R_T, C_T - thermal resistance and capacitance from the ambient to the bottom of the chip; T_b - temperature at the bottom of the chip; T_0 - ambient temperature.

In additional to (1)-(3) it was reasonably assumed a thin chip, good heat sinking and negligible heat flow from the sides.

The equivalent circuits in fig.2 illustrate the main idea of the modeling and don't include the details of the transistor parameter temperature dependence [1, 2].

Using the electro-thermal models of elements we can simulate the whole IC with commercial PSPICE and obtain I-V characteristics and local temperatures of all elements on the chip.

The results of electro-thermal design with PSPICE are illustrated for 4.5 W Op Amp LM12 [7]. The layout of the OA without details is shown in fig. 4. The high power multi-section output transistors Q46 and Q53 are symmetrical and occupy more than 30% of chip area. Because the squares of transistors Q46 and Q53 are very large they are layout on the chip as three subsquares (see fig. 4). Accordingly in SPICE transistor Q46 (Q53) is modeled by three transistors electrically connected in parallel, by thermally they are connected through thermal resistances. Thus each of this transistors Q46 and Q53 is described by three local temperatures. The multi-section representation of large square power elements improves the accuracy of the modeling. Some results of electro-thermal simulation of OA fig. 4 are given in Table 1. It is clear that the maximal temperature of transistors Q46 and Q53 is high (about 70°C) and the neighboring middle power elements are being under a strong thermal influence which varies their electrical regimes.

We have analyzed the convergence of SPICE simulation process of electro-thermal modeling for different types of power ICs and didn't discover insurmountable problems.

To increase the possibilities of power IC electro-thermal design we have connected the modified PSPICE with the 3D temperature simulators of monolith and hybrid ICs [3] which are based on numerical solution of heat flow equations (1)-(3) with conventional boundary conditions taking into account the real IC packaging construction. These simulators provide two very important functions: 1) design of mutual thermal resistance matrix $\|RTQ_iQ_j\|$, thermal resistance and capacitance columns $|RTQ_i|$, $|CTQ_i|$ which are input data for electro-thermal circuit simulation with PSPICE; 2) optimization of IC layout and package construction reducing the rise in temperature caused by heat generation. For example, the final 2D-temperature distribution on OA chip layout is shown in fig. 4.

Mixed electrical/mechanical modeling.

For mixed systems where IC's drive the electrical-mechanical loads, for example, automotive motors, small hybrid stepping motors for floppy disks head positioners, canstack motors for small printers and other computer peripherals, special motors for robots the mixed electrical/mechanical modeling is necessary. The problem is rather complex because the electrical behavior of the transistors driving the windings of the motor is very closely interconnected

	Standard Electrical PSpice Design $R_{load} = 85\ \Omega$; $T_{ambient} = 300$ K			**Electro-Thermal PSpice Design** $R_{load} = 85\ \Omega$; $T_{ambient} = 300$ K		
No of transistor (fig. 4)	Collector current, A	Dissipated power, W	Local temperature K	Collector current, A	Dissipated power, W	Local temperature K
Q52	$6.5 \cdot 10^{-4}$	$1.24 \cdot 10^{-2}$	300	$1.62 \cdot 10^{-3}$	$4.21 \cdot 10^{-2}$	345
Q45	$6.4 \cdot 10^{-4}$	$1.24 \cdot 10^{-2}$	300	$1.66 \cdot 10^{-3}$	$4.32 \cdot 10^{-2}$	347
Q53	$1.37 \cdot 10^{-2}$	0.274	300	0.145	3.75	369.3
Q46	$1.27 \cdot 10^{-2}$	0.254	300	0.148	3.8	370
CPU time*, s	36			40		

*PC-AT 286 20 MHz.

Table 1.

with the mechanical behavior of the driving shaft (torque, rotor velocity, shaft friction) through the current and voltage induced in these windings.

For mixed analysis of electrical-mechanical motor-drive system the mathematical model of the motor is necessary, which consists of electrical and mechanical parts. This models have been developed earlier [4]. The circuit model representation of two-phase asynchronous motor is shown in fig. 5. The electrical part has the special differentiators which define the EMF induced by the neighboring winding currents.

The mechanical subcircuit is represented by the current source T, which is equal to the electro-magnetic torque; capacitance J, which is equal to the rotor momentum and resistance R, which is equal to the bearing friction. These three elements are connected to the node *w* in which the voltage is equal to angular velocity of the rotor. The mechanical part has the special integrator which define the potential F equal to turning angle of thc rotor. The turning angle F is the argument of the current sources in electrical subcircuits.

The circuit model fig. 5 can be simulated as an electrical network by SPICE. As an example fig. 6 represents the result of two-phase asynchronous motor electrical-mechanical modeling with stator and rotor parameters: L^s=0.037 H, L^r=0.0255 H, M=0.8 H, R^s=16.6 Ω, R^r=22 Ω, J=2.18 g·m^2, R=10^{-5} kg·m^2/s.

For this motor the electro-magnetic torque T is described by the following equation

$$T = M \begin{bmatrix} \left(I_1^r \cdot I_1^s + I_2^r \cdot I_2^s\right) \cdot \mathit{Sin}(F) \\ + \left(I_2^r \cdot I_1^s + I_1^r \cdot I_2^s\right) \cdot \mathit{Cos}(F) \end{bmatrix} \tag{4}$$

where R^s,R^r and L^s,L^r - active resistances and scattered inductances of stator and rotor windings;

M - mutual inductance between the stator and rotor windings.

The results of mixed electrical-mechanical modeling (see fig. 6) can help a designer to make a more intelligent match between an IC drive and a given motor or for a given drive chip to provide the current necessary to deliver rated stall torque and also operate the motor at as high speed as possible [6].

The developed approach is general and applicable to the different electromechanotronic systems. It completes the other works where SPICE was used for functional modeling of mixed electro-mechanical systems [5].

Conclusions

Effective design methodologies and techniques of mixed systems using commercial SPICE in the combination with self-made simulators have been developed.

1. New electro-thermal circuit models of ASIC components were presented to use PSPICE for circuit simulation. The temperature node was added in the device equivalent circuit and the thermal depended elements were introduced

which took into account self-heating effects and electro-thermal intercommunions between the elements on the chip .

The effective software system connected PSPICE with 3D temperature simulators providing the additional power for ASIC-designer was developed.

2. The effective approach to use PSPICE for mixed motor-drive system and others mechanotronic systems was suggested. The electro-mechanical equivalent circuit model for different types of motors were developed. The approach can be expended by adding the thermal subcircuits taking into account the IC-driver and motor winding heating effects.
3. The practical examples were discussed illustrating the possibilities of developed design techniques and software tools.

References

1. P.Antognetti,G.Massobrio. Semiconductor Device Modeling with SPICE. McGraw-Hill Book Comp., 1988.
2. R.Fox, S.-G.Lee, D.Zweidinger. The Effects of BJT Self-Heating on Circuit Behavior. IEEE Journ. of Solid-State Circuits, 1993, v. 28, №6, p.p. 678-685.
3. K.O.Petrosjanc, P.P.Maltsev. Thermal designing and simulation of monolith and hybrid ICs. - Proc. of 2-nd Int. Design Automation Workshop (Russian Workshop - 92), Moscow, July 1992.
4. Techn. report "Mathematical models of mechanotronic elements". Applied Problems Section of Russian Academy of Sciences, H-59, 1992.
5. K.L.Paap, B.Klaasen. Functional modeling of mixed systems using SPICE 3. - Proc. of 3-rd Int. Design Automation Workshop (Russian Workshop - 93), July 1993, Moscow, p.p. 171-176.
6. C.Taft, S.Prina. Stepping Motor Drive Chip Selection Considerations. Smart Power Economics, Technology and Applications. Edited by M.W.Smith. Adree Techn. Services Inc., 1988, p.p. 310-338.
7. R.Widlar, M.Yamatake.A Monolith Power Op Amp.- IEEE Journ. of Solid-State Circuits, 1988, v. 23, №2, p.527-535.

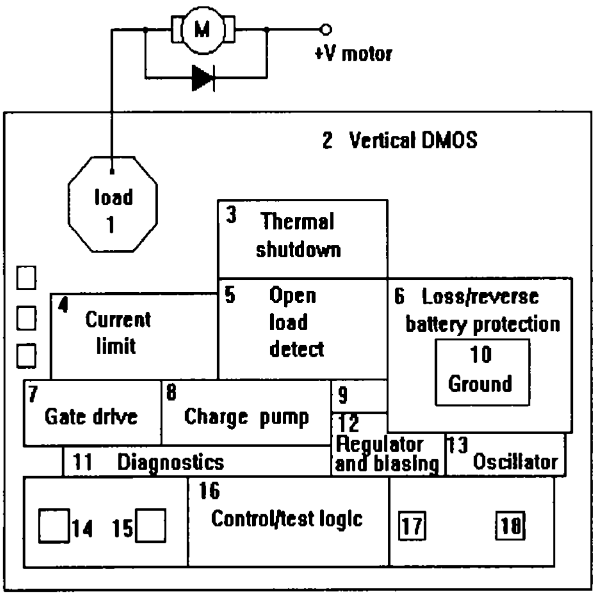


Fig.1. Mixed electro-mechatronic system "the motor being driven by smart power IC".

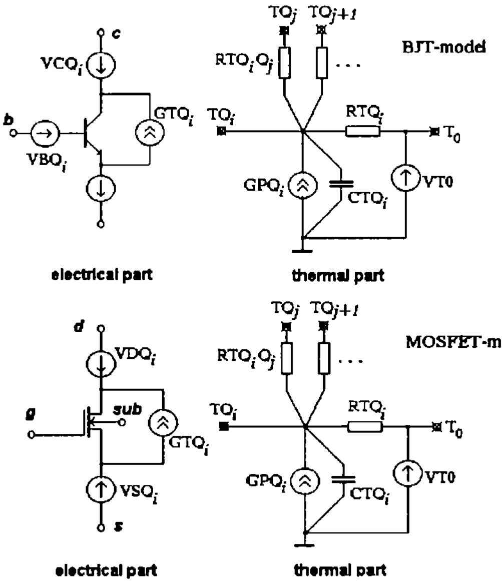


Fig. 2. Electro-thermal SPICE BJT- and MOSFET models.

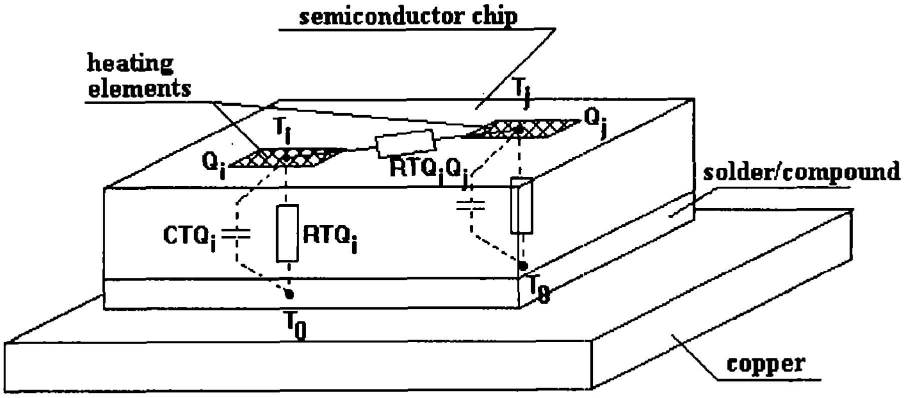
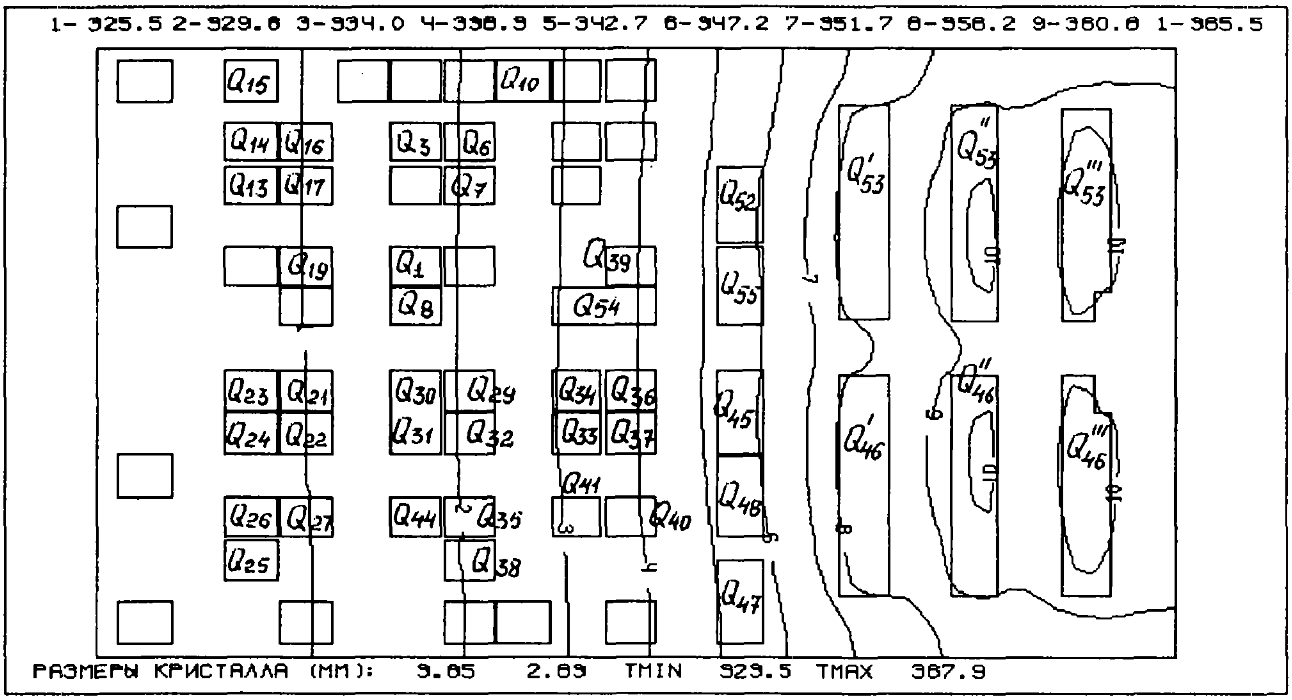
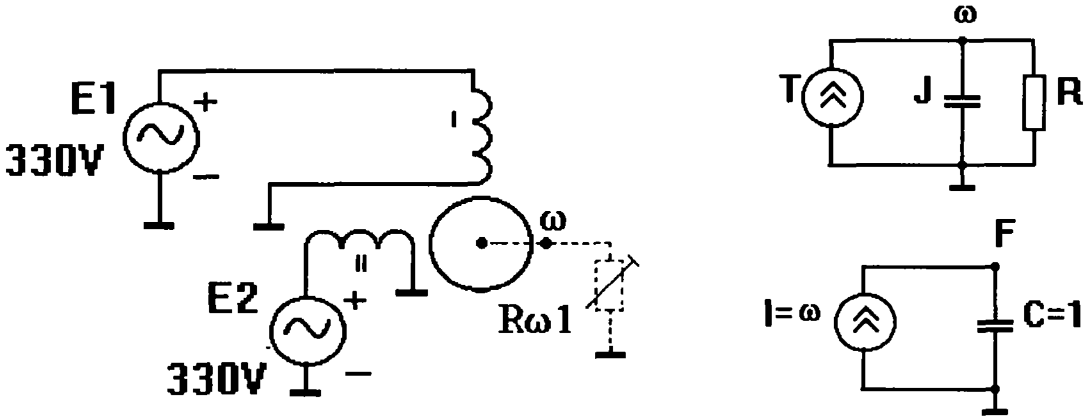


Fig. 3. Power IC representation taken into account thermal interactions between the elements.



Crystal sizes (mm) : 3.85 x 2.83

Fig. 4 2D-temperature distribution on the layout of the power OA.



Electrical part **Mechanical part**

Fig.5. The circuit representation of electrical-mechanical model of two-phase asynchronous motor.

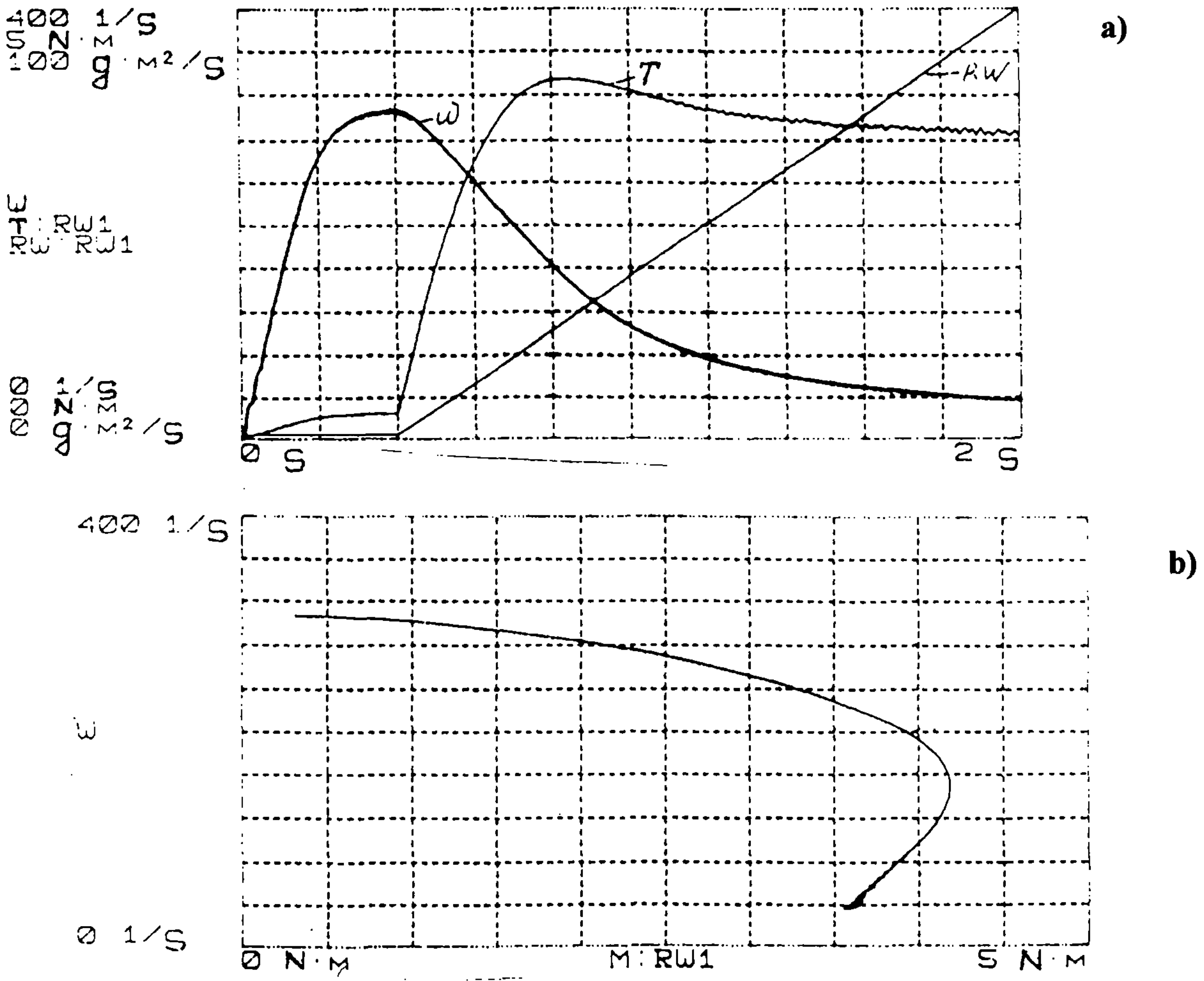


Fig.6. a)Rotor speed ω, electro-magnetic torque T and friction function Rω vs. time; b)Rotor speed ω, vs. torque T.