Electro-Thermo-Mechanical Finite Element Modeling to Investigate the Reliability of Automotive MOSFET Transistor

T. Azoui*[1,2], P. Tounsi[1,2] and J.M. Dorkel[1,2]
[1]CNRS ; LAAS ; 7 avenue du colonel Roche, F-31077 Toulouse, France
[2]Université de Toulouse ; UPS, INSA, INP, ISAE ; LAAS ; F-31077 Toulouse, France
*Corresponding author: tazoui@laas.fr

Abstract: Even under normal operating conditions, power devices are subject to temperature changes that generate high level of stress repeatedly causes mechanical breakdown. In this context thermomechanical simulations are necessary to predict life time and guarantee a longer lifespan. This paper aims to modeling electro-thermo-mechanical behavior of a vertical power transistor for predicting the influence of various geometric parameters such as the thickness of the metallization source on the electrical, thermal and mechanical behavior, and the reliability of the component.

Keywords: Reliability, power devices, finite element modeling (F.E.M), metallization.

1. Introduction

The transportation industry requires a high level of microelectronics component reliability. This implies that the power components should provide a good operation of various vehicle systems, whatever the weather conditions, and for a lifetime of around ten years. Components thermomechanical fatigue is being the major cause of failure of power systems, its study and its evaluation are therefore crucial when designing a component or system. For this purpose, numerical modeling and finite element is particularly become the tool of research and development leading to behavioral analysis electro-thermo-mechanical components and microelectronic power modules. However, the accuracy of simulation results depends strongly on the knowledge of the thermo-mechanical parameters of materials used in the model.

This paper presents a qualitative study of the reliability of automotive MOSFET transistor results from electro-thermo-mechanical simulations. Maximum temperature results obtained from an electro-thermal simulation are mandatory to evaluate the reliability of a power device [1]. This study particularly interested in the stresses generated at the interface between the bonding wire and the source metallization to assess the influence of the thickness of aluminum on stress distribution.

2. Power device F.E model

This section deals with the electro-thermo-mechanical finite element modeling of ultra-low on-state resistance power device. The vertical MOSFET power switch used in the automotive industry is considered to sustain current up to 150 A on a 2 mΩ on-state resistance device.

Power device model is achieved with COMSOL Multiphysics software using a 3D electro-thermo-mechanical element type. The power component consists of power chip that dissipates heat during its operation, two lead frames that are used to evacuate the heat and eight parallel aluminum bonding wires which connect the aluminum metallization layer to the copper lead frame. The component is geometrically and loads symmetric. It considered that it is sufficient to model half of the structure to describe the total behavior of the component. The power device is shown in Figure 1.

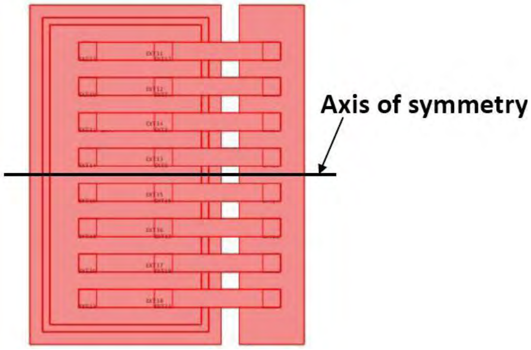


Figure 1. Power device top view.

2.1 Material properties

The temperature dependence of the electric resistivity of the active layer is considered in this model. It is the dominant resistance in the model and has the largest variation with temperature

[2]. The variation of the electrical resistivity with the temperature is given by:

$$\rho(T) = \rho_{T_0}[1 + \alpha(T - T_0]\qquad(2)$$

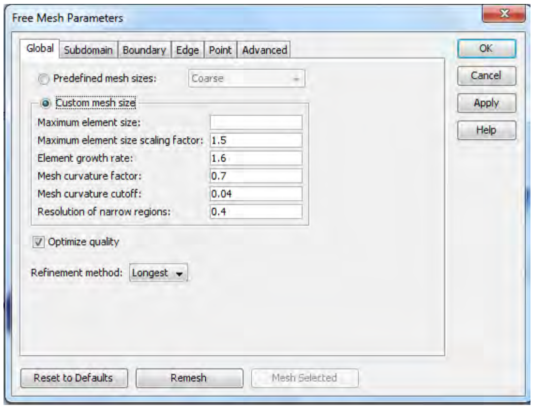
where ρ_{T_0} is the electrical resistivity at reference temperature T_0 and α is proportionality constant [k^{-1}] for the temperature dependence. All the other material resistivities are considered to be constant with temperature because their influences and their variations are limited over the temperature range of interest in electronics (T<200 °C). All thermal properties of materials are considered to be constant with temperature for the same reasons as aforementioned. Mechanical parameters of materials were chosen from the conventional data in literature [3] and [4]. The table below summarizes mechanical properties used in finite element model of the power device.

Table 1: Mechanical properties of materials

Materials	Young's modulus (GPa)	CTE (10^{-6} 1/K)	Poisson's ratio
Aluminum (4µm)	90	24.85	0.33
Aluminum (>10µm)	80	24.85	0.33
Bonding wire (Al)	70	24.85	0.33
Silicon	150	2.6	0.22
Solder (SnPb)	40	2.6	0.4
Lead frame (Cu)	160	2.6	0.33

2.2 Mesh

The issue found when attempting to obtain the FEM of the power device is the scale difference between the thickness of layers making up the chip (micrometer) and its dimensions (millimeter). Layer thickness varies from 4 to 500 µm. Model length is approximately 9000 µm and width 4000 µm. Model meshing is the main difficulty given the micrometer and millimeter scale dimensions of the layers. Thin layers need to be fine-meshed but for reasonable simulation time, the model size must be minimized. To resolve this issue, a smart meshing is undertaken. The free mesh parameters box in the Mesh menu was used and the parameters have been varied until a compromise between the number of elements and precision is obtained. The total number of elements for the model is around 45690.



○ Predefined mesh sizes: Coarse
● Custom mesh size
Maximum element size:
Maximum element size scaling factor: 1.5
Element growth rate: 1.6
Mesh curvature factor: 0.7
Mesh curvature cutoff: 0.04
Resolution of narrow regions: 0.4
☑ Optimize quality
Refinement method: Longest

Figure 2. Free mesh parameter box.

2.3 Boundary conditions

Boundary conditions on the FEM are applied to the bottom surface of both copper lead frames. A zero volt potential is applied to the source lead frame (the one connected to the aluminum wire) and current (I_{DS}) equal to 150 A is imposed on the bottom of the drain lead frame (under the silicon die). The thermal boundary conditions are forced convection. Convection coefficients (h) equal to 2000 W/m².K are applied to the bottom surface of both copper lead frames and external temperature (T_D,T_S) are equal to 20 °C. The mechanical boundary conditions set for the model forcing lower surfaces of the two lead frames to zero displacement in all directions in space and free displacements for the other parts of the device. Transient simulations are made during 50 ms with a step of 10 ms.

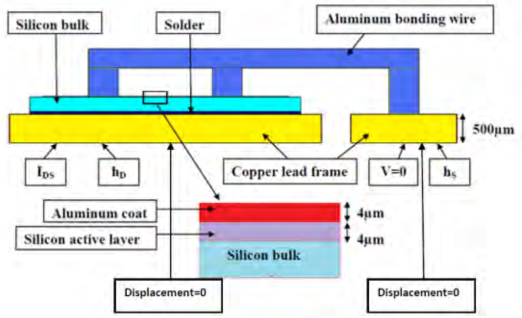


Figure 3. F.E model description.

3. Analysis of simulation results

Electro-thermo-mechanical results presented here were performed with a model with a thickness of metallization of 4 µm and double contact wire connection on the metallization.

Electrothermal results have already been the subject of an article [5] and will not be detailed again here. Figure 4 shows the absolute displacements undergone by the materials.

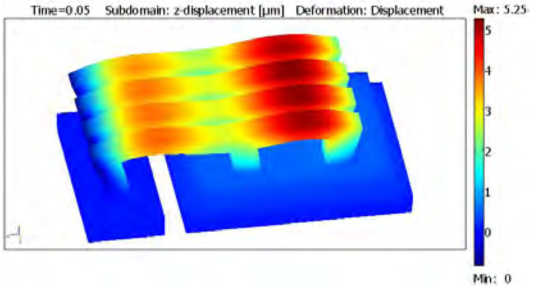


Figure 4. Absolute displacement (µm) of the device.

Mechanical loading, which authorized the movements of all external surfaces of the model, with the exception of lead frames, causes that bonding wires and metallization which have high coefficients of thermal expansion, are being the most deformations under the effect of heat. The values of the higher constraints are around the second wire bonding contact with the power chip. This is illustrated in the figure 5 which shows the Von Mises stress simulated on the metallization.

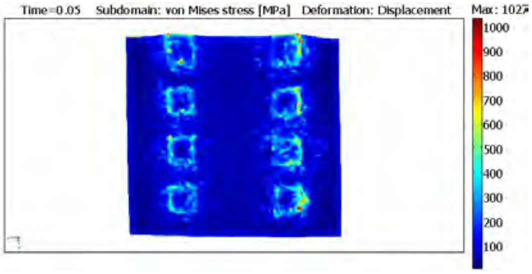


Figure 5. Von Mises stress (MPa) on metallization (e_{Al}=4 µm).

It could be seen that the most important constraints are on the edge of the contact interface between the bonding wire and the metallization. Figure 6 shows the Von Mises stress at the connection interface between the bonding wire and the power chip. The stress values greater than 450 MPa were not represented in this figure.

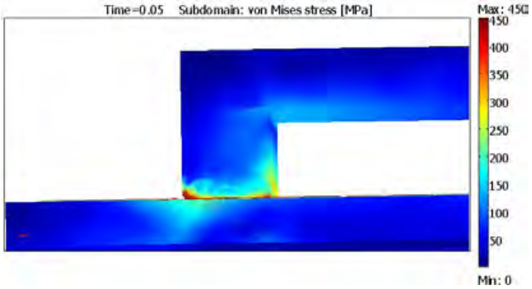


Figure 6. Von Mises stress (MPa) at the interface bonding wire-metallization (e_{Al}=4 µm).

The Von Mises stress cannot allow the visualization of the sign of constraints in materials. For this, the following figure shows the shear stress in the plane of the section of the device.

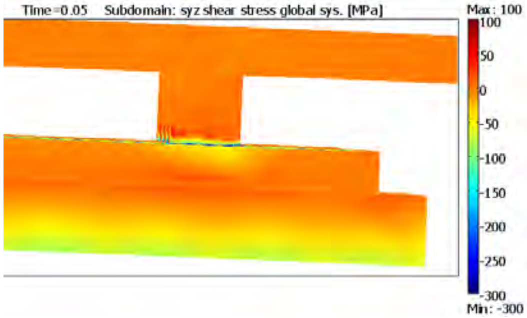


Figure 7. Shear stress (MPa) in the plan section of the chip and the bonding wire (e_{Al}=4 µm).

For shear stresses, we can notice that are essentially negative throughout the contact wire-chip. Analysis of these results allows identify the failure mechanism of the device at this contact, which will result physically by breaking it, following the crack propagation from the areas where concentrations of tensile stresses are greatest.

4. Thermomechanical study of metallization thickness influence

In this section the influence of metallization thickness on the stress distribution at the interface of the bonding wire with the chip power is simulated. A second set of simulations is performed on a device having a 10 µm and 30 µm metallization thickness. Boundary conditions are the same as for previous simulations. The Von Mises stress is presented in the following figure.

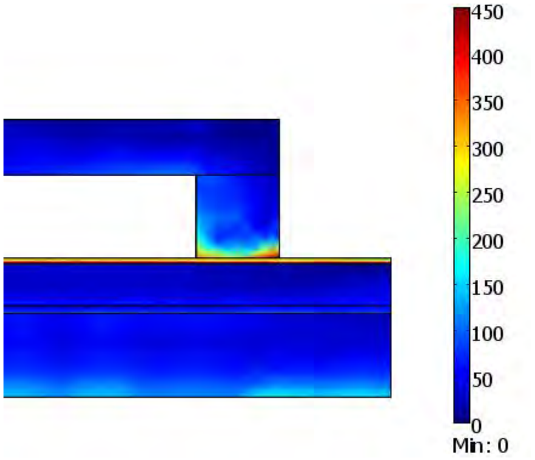


Figure 8. Von Mises stress (MPa) at the interface bonding wire-metallization (e_{Al}=30 µm).

A thick metallization reduces the stress gradient in the metallization. These thermomechanical simulations allow us to conclude that thick metallization improves the reliability of contact between lead wire and chip power.

5. Conclusions

Finite element simulation is an interesting tool to predict the influence electro-thermo-mechanical elements inside and outside the power device. In this paper finite element model of automotive MOSFET transistor is presented. This model is used to study the electro-thermo-mechanical behavior during a short circuit mode of the device in order to investigate its reliability. We have shown that the thickness of the metallization reduces the heating of the device which reduces of the thermomechanical stress sustained by the device.

6. References

1. Am Amro R, Lutz J, Rudzki J, Sittig R, and Thoben M, Power cycling at high temperature swings of modules with low temperature joining technique, *in Proc. ISPSD* (2006)
2. Chauffleur X, Tounsi P, Dorkel J.M , Dupuy P, Alves S, and Fradin J.-P, Nonlinear 3D electrothermal investigation on power MOS chip, *in Proc. BCTM* (2004)
3. P.Aloisi, Les semiconducteurs de puissance - De la physique du solide aux applications, Ellipses, 2001
4. P.Roux, E.Woirgard, C.Zardini, “Influence de la prise en compte d’un état mécanique résiduel sur la modélisation d’un assemblage de puissance”, 2ème journée du RTP Fiabilité des composants et packaging, 2004
5. T.Azoui , S.Verde , J.B.Sauveplane , P.Tounsi, 3D Electro-Thermal Study for Reliability of Automotive Power Vertical MOSFET Using COMSOL Multiphysics, COMSOL Conference 2009