3D Electro-Thermal Study for Reliability of Automotive Power Vertical MOSFET Using COMSOL Multiphysics

T. Azoui*[1], S. Verde[1,2], J. B. Sauveplane[1] and P. Tounsi[1]
[1]LAAS/CNRS, 7 avenue du Colonel Roche - 31077 Toulouse, France.
[2]Department of Electronics and Telecommunications Engineering, University of Naples "Federico II"; Naples, Italy
*Corresponding author: tazoui@laas.fr

Introduction

In this paper we present 3D electro-thermal FE Model using COMSOL Multiphysics software of power vertical MOSFET used in the automotive industry. This model is used to study the influence of metallization thickness and position and number of bonding wires on the electrical and thermal behavior of the power device. The maximum temperature allows evaluating the reliability of the power component. Such modeling is useful for optimization of structure design to guarantee a longer lifespan. It is impossible to simulate microscopic electrical effect of each MOS transistor cell with a F.E. model. In spite of this limitation it is possible to simulate the ‘‘fully ON" behavior of the transistor which is the dominating heat loses if the frequency is about the KHz range. The power component consists of power chip that dissipates heat during its operation, two lead frames are used to evacuate the heat and eight parallel aluminum bonding wires connect the aluminum metallization layer to the cooper lead frame. The component is geometrically and loads symmetric, so we considered that it's sufficient to model half of the structure to describe the total behavior of the component.

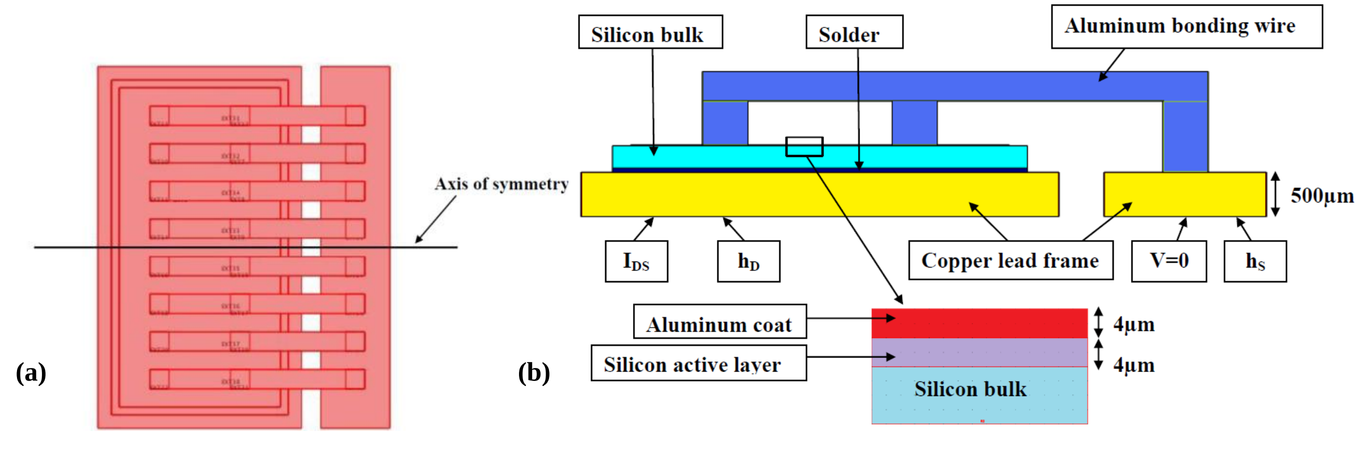

Figure 1: Power component studied
(a) Top view of the power device (b) FEM description

Use of COMSOL Multiphysics

Power device model is achieved with COMSOL Multiphysics software using a 3D electro-thermal element type that has two dependent variables, voltage and temperature. The issue raised when attempting to get 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 has to be minimized. To solve this issue, a smart meshing is made. Boundary conditions on the FEM are applied to the bottom surface of both copper lead frames. Zero volt potential is applied to the source lead frame (the one connected to the aluminum wire) and a current is imposed on the bottom of the drain lead frame (under the silicon die). The thermal boundary conditions are forced convection, the convection coefficients (h) equal to 2000 W m^{-2} K^{-1} is applied to the bottom surface of both copper lead frames. Transient simulations are made. The number of elements for the model is around 36560 and computational time is about 7 minutes.