Material characterization and modeling for power electronics

PhD Candidate: **Alessandro Sitta**
Tutor: **Giuseppe Mirone**
Department of Electric, Electronic and Computer Engineering, Università degli Studi di Catania
alessandro.sitta@unict.it

Analysis of warpage in substrates for power electronics

Introduction

The metallized insulating substrate serves as the supporting structure for the circuitry of Power Module. It works as a base for mechanical support of all the active and passive components, as well as resin framework and heatsink. Due to their specific functions, substrates for power electronics are made by different materials: metal conductive layers that work as thermal path towards heat spreaders, allowing electrical conduction with low resistivity, and ceramic layer that provide excellent electrical insulation. These features play an essential role in the operation of power modules, which are often operated at high voltage and high current density. Since the substrates are composed by different material and considering the typical temperature variation induced in the package during its duty life, warpage is generated also in the substrate, representing an issue for reliability. The scope of the presented work is the characterization of the out-of-plane warpage of active metal brazed substrates (AMB), using a numerical approach, validated by experiments. **The elastoplastic properties of the metal layer have been measured** with a tensile tester, determining yield strength and true plastic flow curve. The temperature softening effect has been also evaluated. These characteristics are needed to calculate **AMB warpage through finite element models (FEM)**, simulating the warpage induced by a passive temperature cycling. Warpage computed from numerical model have been benchmarked and validated with optical warpage measurements. A validated numerical model has been developed to optimize the substrate warpage variation during cycling.

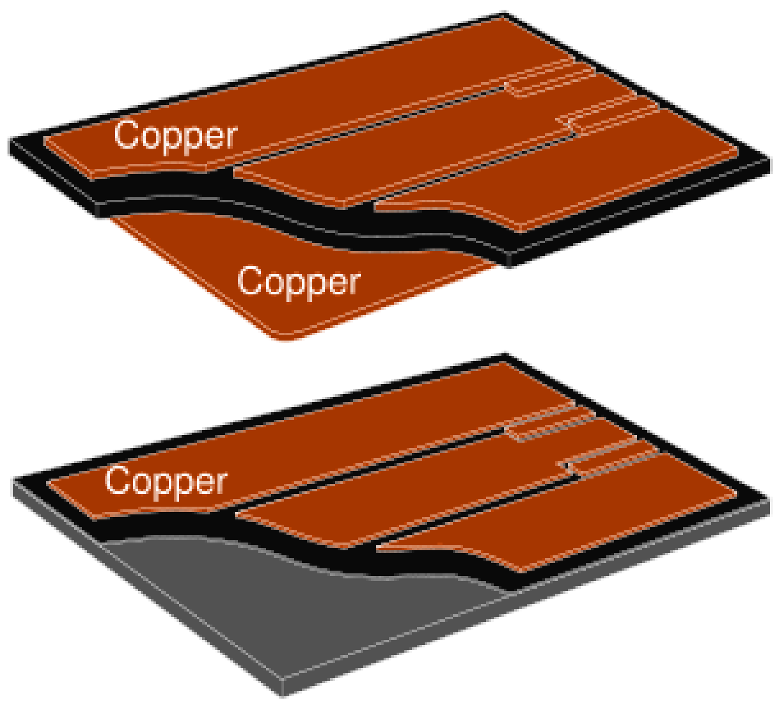

Figure 1: Active Metal Brazed substrate for power electronic packages

Main results

1. The elastoplastic properties of substrate metal layer have been characterized by uniaxial tensile test (10 kN "Instron 5985"), performed at two different operative temperature (RT and 150°C) shows that temperature softening occurs.

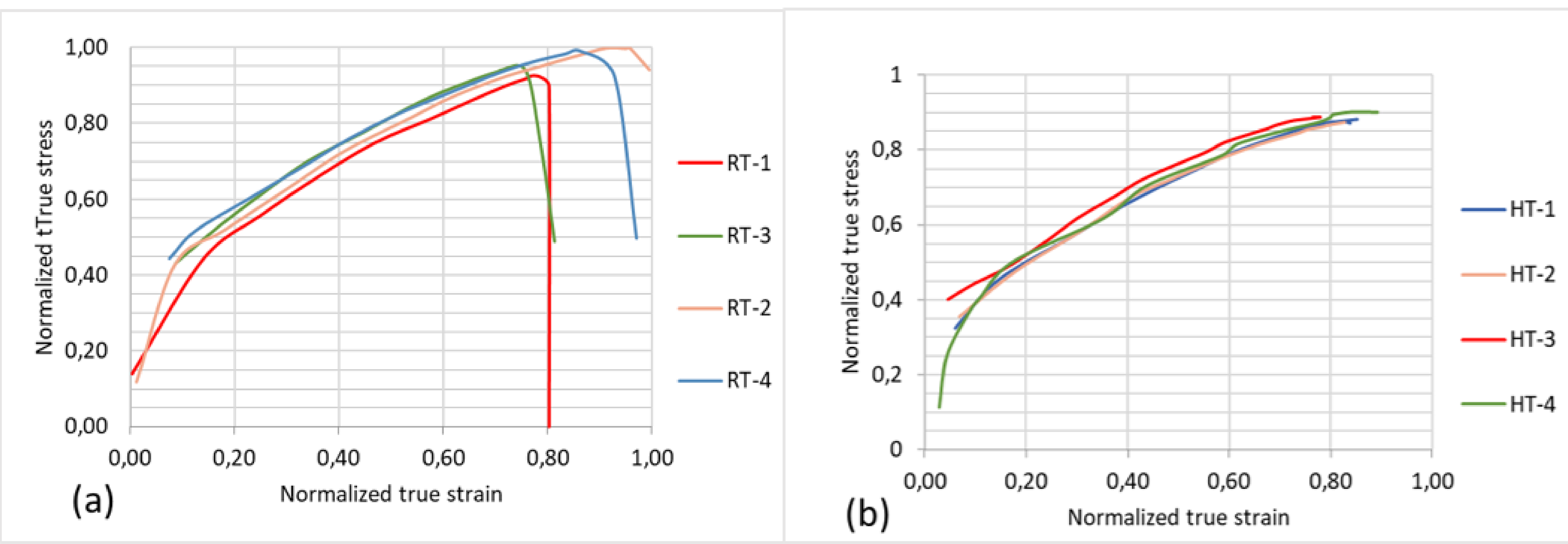

Figure 2: True stress/strain curves for (a) room temperature and (b) high temperature samples.

2. Finite Element Model for warpage calculation has been experimentally validated measuring (with Phase Shift Moiré method) the relative warpage during a *thermal cycle*, performed between RT and 150°C at low temperature rate (0.2 °C/s).

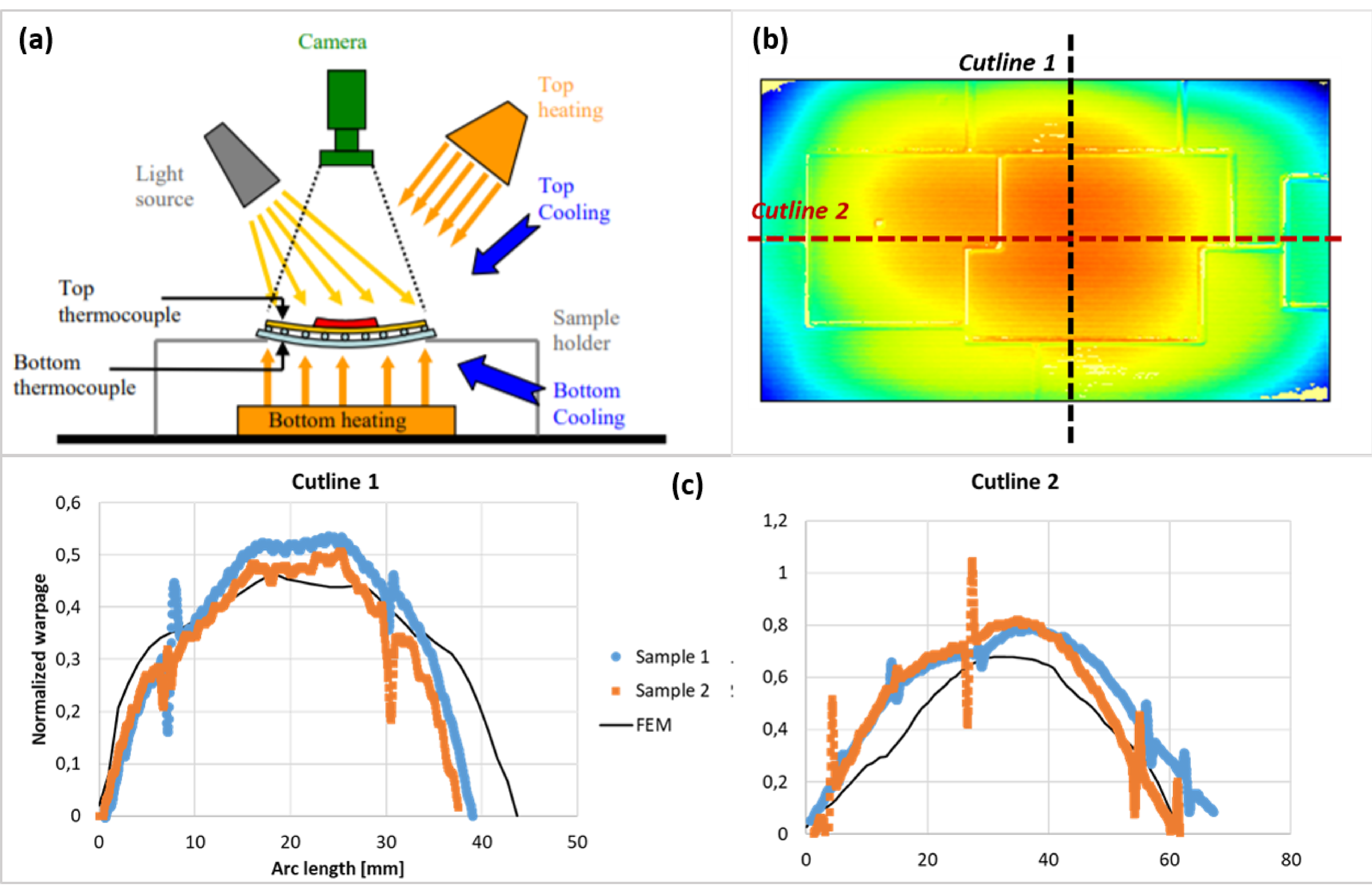

Figure 3: Experimental and simulation activities for substrate warpage calculation. In (a), the schematic of experimental setup for warpage measurements was show, while in (b) a 2D warpage map is depicted. Figure (c) benchmarks simulated values with experimental data coming from thermal cycle measurements.

Wire Bonding Modeling

Introduction

Wire bonding is a important stage in an electronic package assembly process: it serves to provide the electric connection between the semiconductor chip and the external world. Due to the huge number of wires used in semiconductor devices and considering that mechanical stresses are induces during process, the reliability of wire bonding is needed for a product robust design. Modeling is a good approach to understanding the stress impact to both the wire bonding process and semiconductor device, and it may further help to improve the design of the whole bonding process for avoiding die failure such as crack/debonding.

Currently, wire bonding is modeled considering pure mechanical bonding loads with static methods to simulate the free air ball (FAB) under compressive bonding process. Only few simulations include both the dynamic nonlinear wire bonding process and semiconductor device stresses in the same model due to difficulties getting convergence. Furthermore, these authors never consider 3D model, which is instead necessary for accurate calculation due to the typical process boundary conditions.

The aim of the activity is to develop a **3D Finite Element Model** that is able to calculate both the global variable (e.g., bonding force) material stress map **induced by wire bonding stage**. In order to get the optimal calculation accuracy, both dynamic effects and non-linear material properties must be considered and numerical results have to be benchmarked with customized experimental trials and characterization.

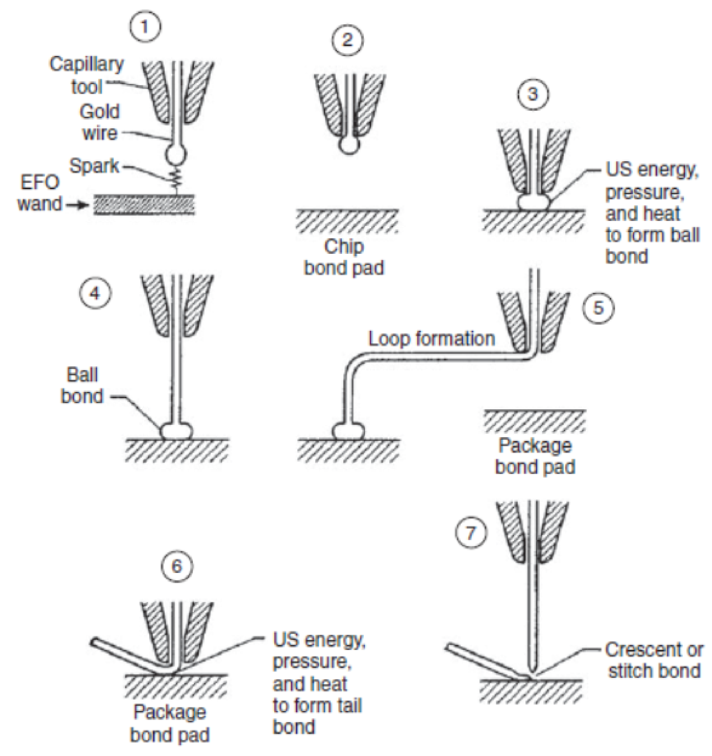

Figure 4: Typical process flow for wire bonding

Interaction between wire and pad during

The tool capillary (in red) presses the FAB (in green) on the pad structure. After the contact between pad and FAB, force increases up to maximum value ("impact force"). Then force is released up to "bond force", when ultrasound is applied on capillary. Considering that ultrasound is applied along one direction, the axisymmetry of the first phase is broken. Due to the short time of wire bonding process, dynamic effect affects the welding phenomenology. Ultrasound helps to soften the material, so the force needed for welding is reduced, with less damage on pad.

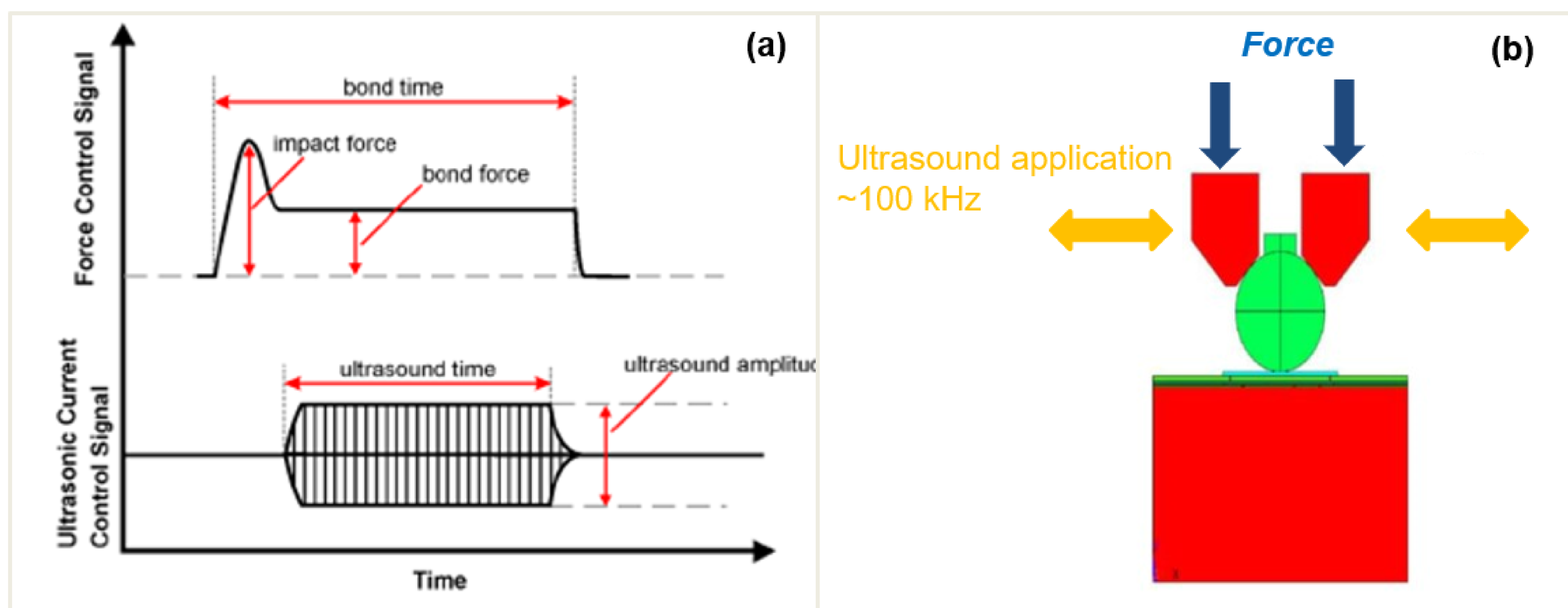

Figure 5: Detailed figuration of wire/device pad interaction in wire bonding process. In (a), the dynamic force and ultrasound profiles are depicted, while in (b) the force application on ball is sketched.

Preliminary results (impact force)

Deformation and stress in deformed ball have been calculated after the impact phase, considering a half-symmetry model. The vertical reaction force matches the impact force applied during wire bonding process.

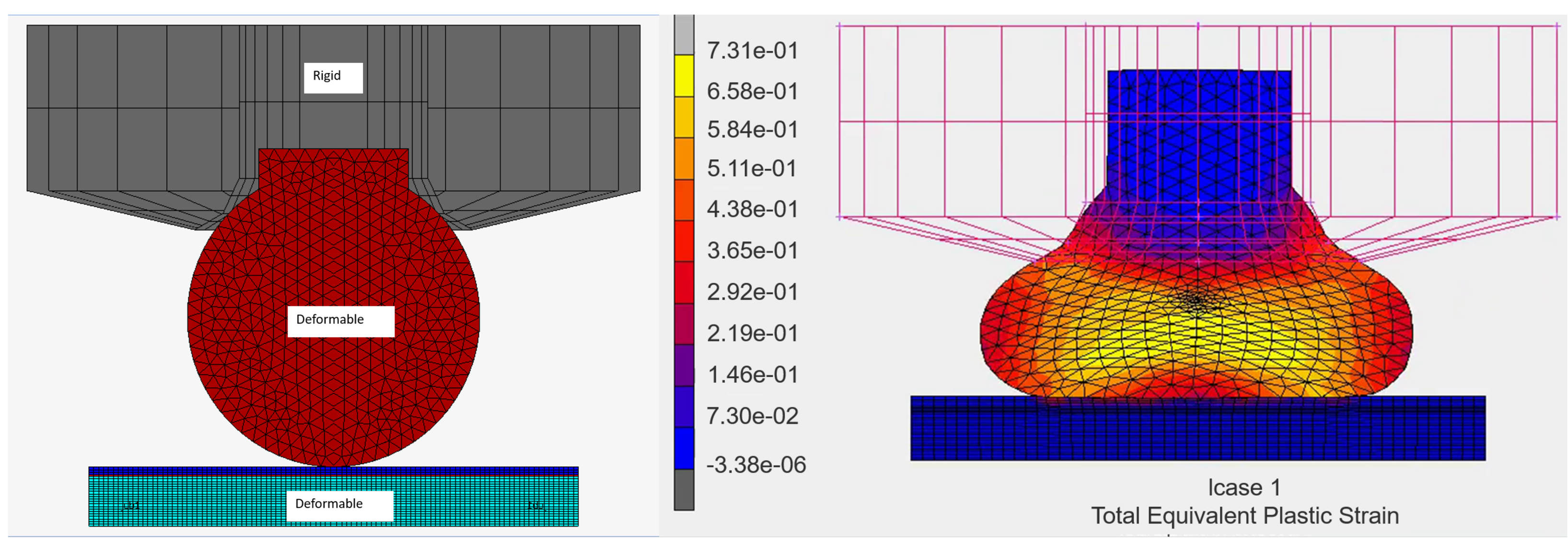

Figure 6: Geometry mesh and plastic strain in wire ball. Due to the strong plasticity level, slight differences in terms of stress distribution are obtained in the surface contact area between Cu ball and capillary (rigid element).