SIMULATION OF THE PLASTIC DEFORMATION OF CONTACT PADS DURING MICROELECTRONIC TESTING

D. DICKINSON[1], J. SÄGEBARTH[1], H. SANDMAIER[1], M. SCHNAITHMANN[2], J. SEEKAMP[2], G. BÖHM[2], W. SCHÄFER[2]

[1] *Chair in Microsystem Technology of the University Stuttgart*, *dave.dickinson@mst.uni-stuttgart.de*

[2]*Feinmetall GmbH, matthias.schnaithmann@feinmetall.de*

Keywords: *Microelectronic testing, Plastic deformation, Structural mechanics, Integrated circuit*

During microelectronic testing of ICs (**I**ntegrated **C**ircuits), contact elements of a testing device get in contact with e.g. aluminium-pads on the IC. The pads are penetrated and deformed plastically (see figure 1). The worst case scenario of testing is to damage the pad such that microstructures located beneath the pad are damaged too [1]. This could reduce the effectiveness of the tested microchips or even cause a total failure. To prevent this issue, a numerical simulation of those plastic deformed pads is essential. The objective of the simulation is to predict the size of the deformation and the mechanical stress in the pad and the IC.

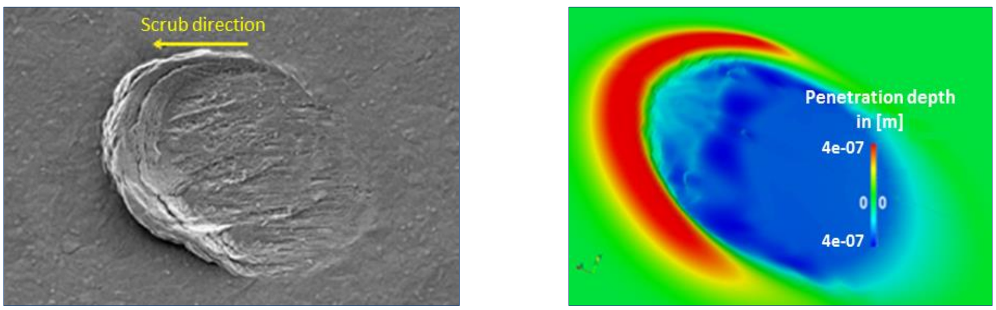

Figure 1: Illustration of the plastically deformed aluminum-pad from experiment (left) und simulation (right)

At the MST (Chair in **M**icro**s**ystem **T**echnology of the University of Stuttgart) OpenFOAM [2] is used as simulation software for the development of novel contact elements for the above described testing devices. The base of this work is the solid mechanics library [3] of foam-extend-3.1. By substituting constitutive equation (material relation) in the momentum equation and by the use of Lamé parameters and the basic approach of the ideal plasticity ε_p [4], the governing equation in the mechanic solver is as follow:

$$\nabla \cdot \{\mu \nabla(du) + \mu[\nabla(du)]^T + \lambda I tr[\nabla(du)] - [2\mu(d\varepsilon_p) + \lambda I tr(d\varepsilon_p)]\} = 0 \quad (1)$$

The implementation of the geometry and mesh generation has been done with the help of SALOME [5]. The parameterization of the material model was based on the literature study and the experimental design properties. In addition, the contact model parameters of the solid mechanic solver were optimized using a parameter study with DoE method [6] (**D**esign **o**f **E**xperiment) to ensure a real contact design and terminated computing time.

To obtain a realistic simulation result, the simulation has been validated with experimental data. To simplify the validation, a test was performed using nanoidentation [7]. The nanoindenter [8] penetrates the pad surface with a predefined depth and length and deforms it plastically (see figure 2, left). For the validation different penetrations depths and lengths were generated with two different nanoindenters, a Berkovich- and a spherical-indenter. The topological analyses of the plastic deformation are done with AFM (**A**tomic **F**orce **M**icroscopy, figure 2 right) and SEM (**S**canning **E**lectron **M**icroscope, figure 1 left). During the experiment, the required force has been measured.

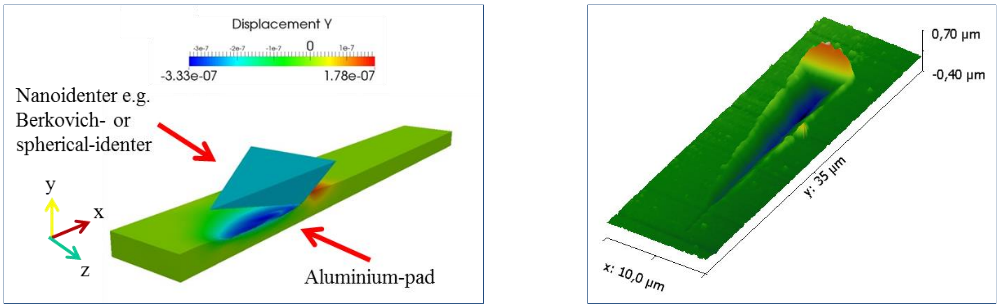

Figure 2: Results of the plastic deformation of an aluminium-pad penetrated with a Berkovich-indenter (Left) simulation of the result, (right) AFM measurement of the experimental result

Finally the measurement results were compared with the simulation results. To ensure a proper validation, the motion of the indenter within the experiment and simulation has to be matched (see figure 3 left). And then the force and the penetration depth were compared (see figure 3 right). The force is registered on the y-axis and takes progress depending on the penetration depth on the x-axis. The red curve was drawn from the experimental result, while the blue curve comes from the simulation result. Hereby is clearly to see that the simulation reflects the reality very well.

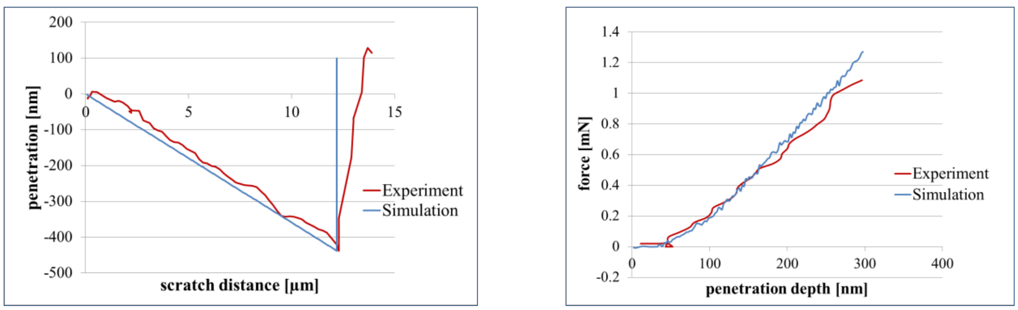

Figure 3: (left) Comparison of the motion, (right) comparison of the force depending on the penetration depth

The occurred deviation here could be the measurement failure and also imprecise parametrisation of the simulation, which should be optimized in the future work. In addition, the computing time for the calculation should be further reduced, so that it would be flexible to handle for industrial practice. A greater challenge and optimization is still in the removal of the material surface, which was carried by the contact element during the penetration. For such virtual mathematical illustration, the use of crack solver [9] within OpenFOAM is conceivable.

References

[1] ASM International, Microelectronics Failure Analysis, 5th ed. Ohio: EDFAS, October 2004.

[2] The OpenFOAM® Extend Project: OpenFOAM User Guide, CFD Direct Ltd., Reading UK, 2015.

[3] P. Cardiff, A. Karac, Z. Tukovic, H. Jasak, D. Carolan, M. Leonard, V. Kanyanta, D. McAuliffe, D. McNamara, T. Tang, Solid mechanic solvers to numerically approximate the displacements and stresses in solid bodies undergoing deformation, Dublin: University College Dublin, Alojz Ivankovic's research group, 2014.

[4] T. Tang, Implementation of solid body stress analysis in OpenFOAM, Denmark: Technical University of Denmark, 2013.

[5] SALOME: The Open Source Integration Platform for Numerical Simulation. SALOME 7.7.1 documentation, OPEN CASCADE, 2016.

[6] B. Giger, Einführung in die statistische Versuchsplanung (DoE), TQU Group, June 2014.

[7] W. C. Oliver, G. M. Pharr, An improved technique for determining hardness and elastic modulus using load and displacement sensing indentation experiments, 7th Vol. Tennessee : Materials Research Society, June 1992.

[8] MICRO STAR TECHNOLOGIES, Nano Indenters from Micro Star Technologies, 2.3 Rev. Huntsville: Micro Star Technologies, 2015.

[9] D. Carolan, Z. Tuković, N. Murphy, A. Ivankovic, Arbitrary crack propagation in multi-phase materials using the finite volume method, Dublin: University College Dublin, Computational Materials Science, 2013.