Design Support and Development of Miniaturised Si-Microphone Packages

Johann-Peter Sommer

Fraunhofer Institut for Reliability and Micro Integration (IZM), Berlin,
Dept. Mechanical Reliability and Micro Materials, Email: sommer@izm.fhg.de

Introduction

Numerical sensitivity analysis by means of finite elements (FE) is useful to reduce cost and time to market by minimising real tests and expensive redesign. Within the research project "InFON", funded by the German government (BMBF), the FE methodology was improved and applied to miniaturised Si-microphones, to be starting already at an early stage of the design process. Several material combinations for board, adhesives, and case were investigated in order to evaluate the pre-stress stability of the chip membrane, which is influenced by the sequence of manufacturing steps and their temperatures due to the unavoidable thermal mismatch between different materials. Besides them, the membrane deflection under electrostatic loading had to be characterised regarding the pre-stress state in the membrane and the real topology of the backplate electrode.

Modelling of the manufacturing process

One of the main targets of the FE analysis was to understand the stress/strain-behaviour of the microphone during manufacturing before any real device was available. Starting with an axisymmetric FE model the following fabrication steps were modelled using the so-called sequential build-up principle:

- Apply defined pre-stress into the membrane,
- Heat up to the die bond curing temperature,
- Cool down the microphone-carrier-connection to room temperature (RT),
- Heat up the part to the case bond curing temperature,
- Add case and cool down the complete microphone to RT.

Each new part was assumed to be stress-free when added. This is an appropriate assumption due to the relaxation property of epoxy adhesives at an elevated thermal level.

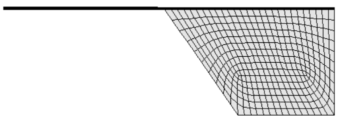

Figure 1: Active part of the FE model during computational steps 1 and 2

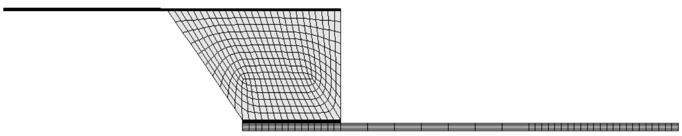

Figure 2: Active part of the FE model during computational steps 3 and 4

Fig. 1 and 2 give an imagination of the sequential build-up FE modelling.

Reliable numerical results can only be expected based on reliable material data for all components. First, pure elastic but temperature-dependent material parameters were used for the adhesive, which decreased the effort for material data measurement. Bond materials from several suppliers were compared, and the influence of a varied glue layer thickness and of different carrier types on the membrane stress constancy was evaluated. This way, recommendations for the microphone design could be derived already very early in the InFON project.

High strain concentrations are expected in the adhesive layers at the free interface edges. But the membrane stress is affected relatively slightly due to the low glue stiffness compared to that of the die and the substrate. After the decision for the best-suited adhesives, the visco-elastic behaviour of the most important adhesives was characterised by rheometer measurements, and the relaxation properties of this glues could be taken into account in the further FE analyses. This way, a more precise understanding of the manufacturing process was achieved. As an example, Fig. 3 shows the membrane stress variation during assembling for 3 different glue combinations.

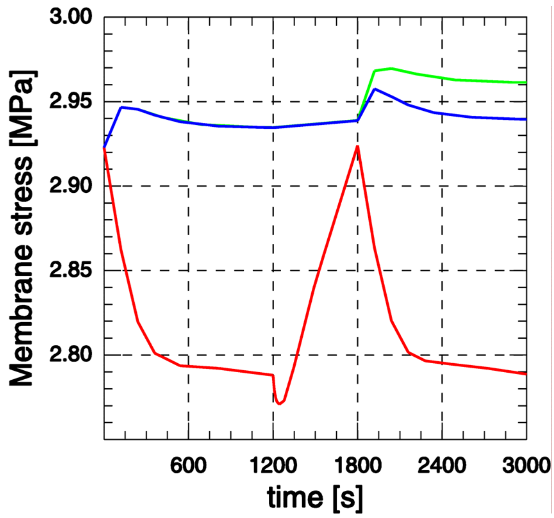

Figure 3: Membrane stress variation

A 3d FE model was created in order to determine the influence of the cubic shape of chip and case compared with the circumferential electrodes inside. Shell elements were combined with solids again, and 1/8 symmetry was utilised, Fig. 4. Furthermore, corrugations close to the membrane boundary were evaluated with respect to their number, width, height, and facet angle, and different kinds of membrane fixing have been analysed by means of FE. Finally, a numerically recommended solution for Si-microphone packaging could be pinpointed, and FE techniques have been proved to be well-applicable in early microphone design.

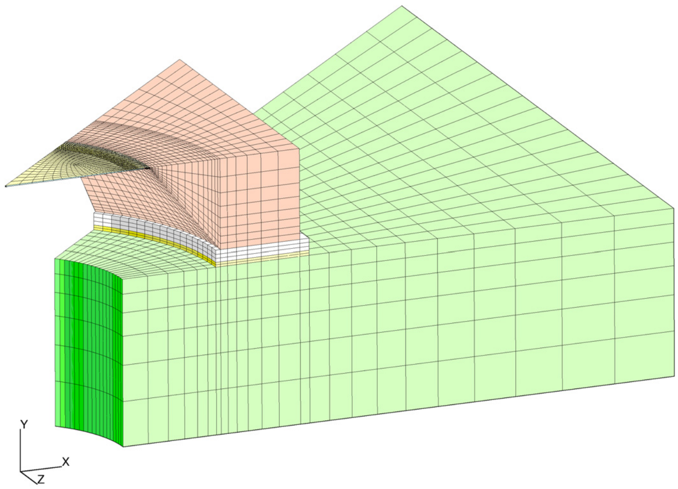

Figure 4: 3d FE model (46,000 nodes)

Pull-in test

In order to measure the prestress in the membrane, a so-called pull-in test can be used. If it is possible to analyse this process numerically, the shape and the membrane stress of the Si microphone can be pre-optimised for best acoustic properties. As an example, Fig. 5 shows the principle of an axisymmetric FE mesh consisting of shell elements for the membrane and the counter electrode and solid elements for isolating layer and the chip outside. A complicated controlling of applied load was necessary due to the position-dependent and strongly non-linear electrostatic forces, Fig 6.

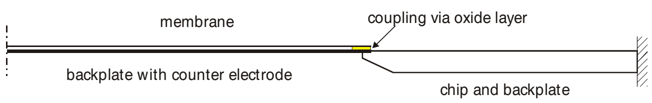

Figure 5: Axisymmetric FE mesh for the pull-in analysis

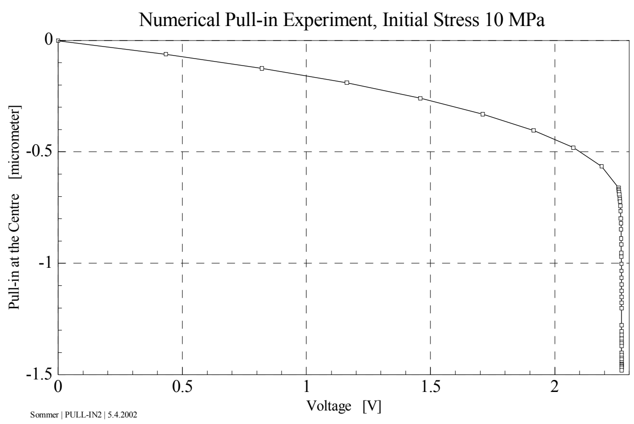

Figure 6: Strongly non-linear electro-static force control during the pull-in experiment

Several technologically interesting questions could be discussed numerically. As an example, how do warped backplates influence the functionality of the microphone? It could be shown by means of FE analyses, that low warpage can be neglected. Higher one yields in a membrane, which doesn't snap to the backplate in the centre, but comes in first contact at a ring, Fig. 7.

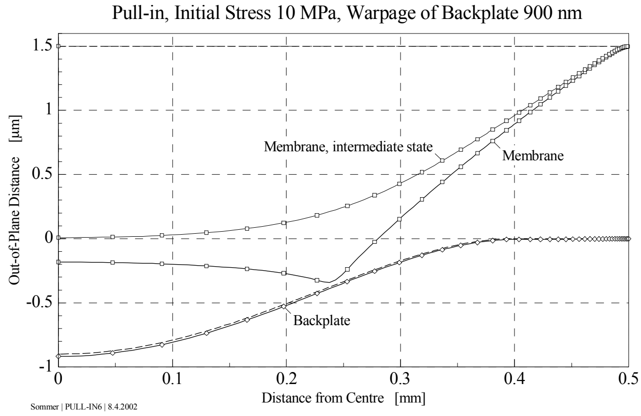

Figure 7: Pull-in-test for a strongly warped backplate: Shape and position of the membrane at several time levels

The membrane and the backplate consist of several layers which have been deposited at different temperatures and special ambient conditions. Usually the assembling process results in an intial stress state which deforms these layers. Furthermore, the initial deformation is influenced by geometric parameters as well. Assuming all initial stresses to be of thermal nature, a numerical sensitivity analysis was carried out to decide wheater a parameter is of strong influence or not. Based on an improved FE model with composite shell elements and incorporated corrugations many technologically interesting variants have been evaluated. Fig. 8 is a typical deformation state of membrane and backplate.

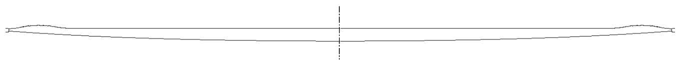

Figure 8: Warpage of membrane and counter electrode (20-times enlarged)

For example, the influence of the diameter of the etching cavity and the thicknesses of the different layers have been evaluated in detail.

Summary

FE techniques have been proved to be well-applicable especially in early design steps of innovative and advanced microelectronic systems. However, reliable numerical results can only be expected based on reliable material data for all components. This is important preferably for adhesives and other plastics. Neglecting their visco-elastic and temperature-dependent properties, a realistic thermo-mechanical response from the device under thermal load cannot be expected.

References

[1] Sommer, J.-P., Wittler, O., Manessis, D., Michel, B.: Design Process Supporting Simulations on Wafer Level Packages. Proc. Conf. Design, Test, Integration and Packaging of MEMS/MOEMS (DTIP 2003), Mandelieu (F), May 5-7, 2003, 45-49.

[2] Sommer, J.-P.: Support of the Design process for Advanced Electronic Components by Means of Finite Element Analysis. Series Micromaterials and Nanomaterials, issue 03 2004, 268-273.