MODEL REDUCTION FOR THERMO-MECHANICAL SIMULATION OF PACKAGES

E. Zukowski, J. Wilde, E. B. Rudnyi, J. G. Korvink

University of Freiburg , IMTEK

ABSTRACT

Model reduction is a new numerical technique [1, 2] that allows us to obtain an accurate low-dimensional representation of high-dimensional finite elements models. Additionally, in the case of linear models the time to perform model reduction is comparable to the solution time of a stationary problem. Hence model reduction can be employed as a fast solver for a transient or harmonic problem during the optimization process [3]. The goal of the paper is to explore new possibilities and methodology to apply model reduction during design of new packages. The described approach has been limited by us to thermo-mechanical finite element models developed in ANSYS.

1. INTRODUCTION

The qualification of packages during the development process includes an investigation of the influence of design parameters on the package life assessment. Among others, application conditions include the temperature range, so that a component qualification plan can be formulated to derive accelerated operating conditions from the application conditions.

Heat has significant influence on the package reliability [4]. It can reduce lifetime, get rise to thermal deformation, etc. In order to verify a package lifetime, thermal cycling tests are performed, because thermal stresses stimulate reliability problems, such as die cracking, interfacial delamination and solder joint fatigue.

However, experimental investigation of package design options in the development process can be costly and time consuming. Hence, computer simulation of product life cycles can be a valuable tool to evaluate designs in a virtual qualification [5, 6]. Typically analysis with the finite elements method (FEM) is employed to solve the thermal, stress and reliability issues under different parameter sets. Probabilistic design can be used to determine the effect of one or more variables on the outcome of the analysis. Nonetheless, the estimation of lifetime sensitivity to various geometry and material parameters for large FE-model requires computationally demanding simulations, particularly for non-linear materials behaviour.

Model reduction is an efficient means to build a low-dimensional model with good approximating properties directly from the finite element model [1, 2]. In our knowledge, model reduction approaches have not been used for thermo-mechanical problems so far. As a result, the goal of the present paper is to demonstrate the application of the model reduction to a thermo-mechanical problem.

We have limited ourselves to thermo-mechanical finite element models developed in ANSYS. First, ANSYS is a popular engineering tool used quite often for thermo-mechanical simulation, second, there is software available (**mor4ansys** [3], see also http://www.imtek.uni-freiburg.de/simulation/mor4ansys/) to perform model reduction directly on ANSYS models. We have chosen two typical thermo-mechanical problems to demonstrate the possibilities of model reduction:

1) A mechanical structure affected by the thermal stress under assumption of uniform temperature.
2) A mechanical structure affected by the thermal stress under a local heat source. In this way the impact of thermal distribution on the thermal-mechanical performance was also evaluated.

In both cases we limited ourselves to linear models, that is, constant material properties. We introduce model reduction in section 2, present results for the case studies in Sections 3 and 4 and discuss the results in Section 5.

2. MODEL REDUCTION OVERVIEW

Discretization in space by the finite element method produces a system of ordinary differential equations as follows

$$\begin{aligned} M\frac{d^2x(t)}{dt^2} + E\frac{dx(t)}{dt} + Kx(t) &= Bu(t) \\ y(t) &= Cx(t) \end{aligned} \qquad (1)$$

where $x(t)$ is the unknown state vector; M , E and K are system matrices, B is the input matrix, and C is the output matrix. The vector u comprises input functions (external excitations) and the output matrix specifies particular linear combinations of the state vector that case of interest to an engineer.

In the case of a thermo-mechanical problem, the state vector is composed of deformations U and temperatures T and the system matrices are a result of corresponding structural and thermal problems

$$\begin{Bmatrix} M_U & 0 \\ 0 & 0 \end{Bmatrix}\begin{pmatrix}\ddot{U}\\ \ddot{T}\end{pmatrix} + \begin{Bmatrix} E_U & 0 \\ 0 & E_T \end{Bmatrix}\begin{pmatrix}\dot{U}\\ \dot{T}\end{pmatrix} + \begin{Bmatrix} K_U & K_{UT} \\ 0 & K_T \end{Bmatrix}\begin{pmatrix}U\\ T\end{pmatrix} = Bu \quad (2)$$

with the coupling due to the K_{UT} matrix. As such, model reduction theory should work for a thermo-mechanical problem as well.

The goal of model reduction is to find a low-dimensional approximation for Eq.(1). In the case of a linear dynamic system, that is, when the system matrices are constant, there is a complete mathematical theory on it [1]. It is worthy of noting that the main advantage of model reduction is in its application to a dynamic system when the goal is to evaluate time-dependent solutions.

The model reduction theory is based on the approximation of the transfer function of a dynamic system. In other words, Eq.(1) is transformed to the Laplace domain where a system of ordinary differential equations (1) becomes a system of linear equations. As a result, the ration between inputs and outputs can be written in terms of the transfer function

$$H = Y/U = C\left\{s^2 M + sE + K\right\}^{-1} B \quad (3)$$

where s is the Laplace variable, and Y and U are the output and input functions in the Laplace domain accordingly.

The approximation is achieved by the introduction of a few generalized coordinates z that describe a low-dimensional subspace V

$$x \approx Vz \quad (4)$$

with good approximation properties for the transfer function (3). In this respect, software **mor4ansys** [3] implements the implicit moment matching via the Arnoldi process.

After the low-dimensional subspace V is found, Eq.(1) is projected on it as follows in the time domain

$$\begin{gathered} V^T M V \frac{d^2 z(t)}{dt^2} + V^T E V \frac{dz(t)}{dt} + V^T K V z(t) = V^T B u(t) \\ y(t) = C V z(t) \end{gathered} \quad (5)$$

or in the Laplace domain

$$H_r = Y/U = CV\left\{s^2 V^T M V + s V^T E V + V^T K V\right\}^{-1} V B \quad (6)$$

As the dimensionality of the systems (5) and (6) is much less than that of the original systems (1) and (3), there is considerable saving in simulation time for transient and harmonic simulation.

3. MECHANICAL STRUCTURE AFFECTED BY UNIFORM TEMPERATURE

A 3D finite element model of a package on a board (Fig.1) was made by the modular modelling method [7]. In this case, the modules from a library are connected to each other only by means of contact elements (CONTA173, 3-D 4-Node Surface-to-Surface Contact and TARGE170, 3-D Target Segment). We have exploited the symmetry and modelled a quarter of the package. The SOLID185 elements (3-D 8-Node Structural Solid) have been used. The model consists of 12379 elements (including contact elements), 9500 nodes and 17355 equations in the system Eq.(1). Seven materials have been modelled such as moulding compound, silicon die, die attach, solder mask, copper pad, solder and PCB (epoxy-glass). The materials properties were assumed to be linear-elastic and temperature independent.

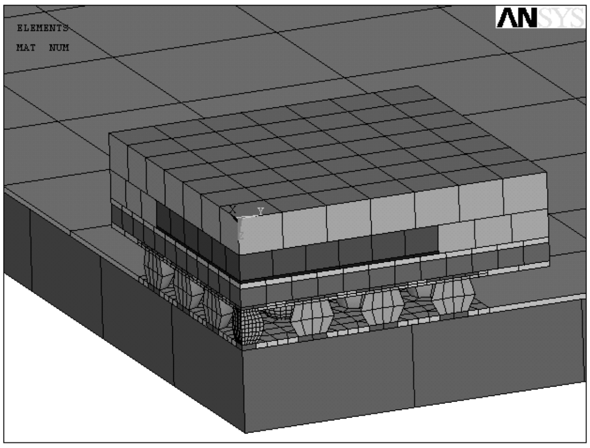

Fig. 1. A quarter 3D finite element model of a package.

When the temperature changes, the mismatch of the coefficients of thermal expansion of different materials leads to deformations as well as to stresses and strains. In our model, the mismatch causes strong bending with a high z-axis displacement. Fig. 2 shows the deformed assembly at 150 °C.

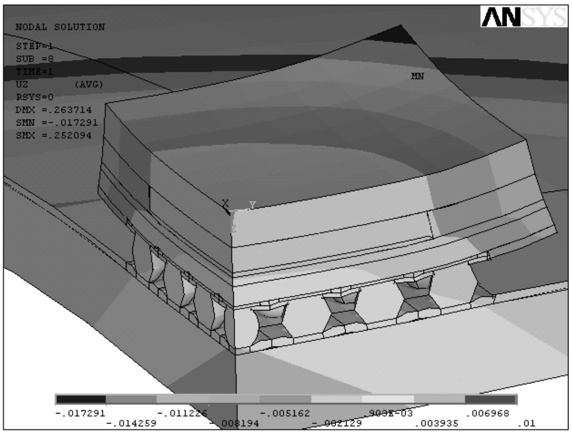

Fig. 2. Z-component of displacement in 3-D quarter symmetry FE-model at 150°C. (Max. displacement by corner node of package)

In the case of the uniform temperature we can neglect the thermal system matrices in Eq.(2) and the state vector contains the displacement only. The thermal effect in this case is modelled though the load vector B in Eq.(1). This is the standard case for **mor4ansys** [3] and it can be applied directly. **mor4ansys** reads system matrices from the binary ANSYS files, performs model reduction and writes the reduced model in the Matrix Market format [8].

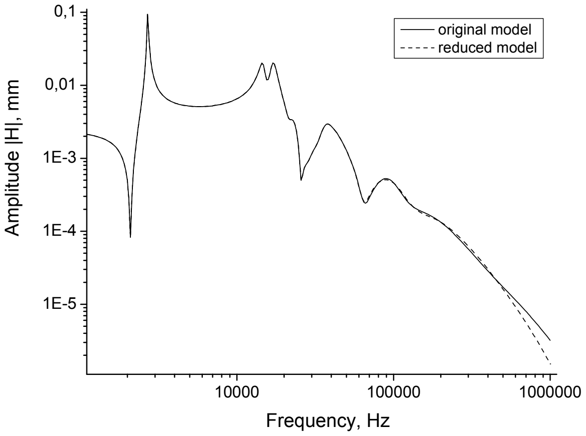

Fig. 3. The transfer function Log(|H|) of the 3-D quarter symmetry FE-model for the IC corner node in z-direction (max. displacement).

In Fig. 3, a transfer function of the original model of dimension 17355 is compared with the transfer function of the reduced system model .The plot has been made for U_Z of the corner node of the IC. The difference between the two curves is very small in the beginning and becomes slightly noticeable above 10^5 Hz. This means that the reduced model captures accurately the dynamic effects of the original model. We show the transient response of the model for the same degree of freedom in Fig. 4 in the case of the step response. We plot both curves for the original and reduced model and here the difference is even smaller than in the previous case. We show the difference plot in Fig. 5.

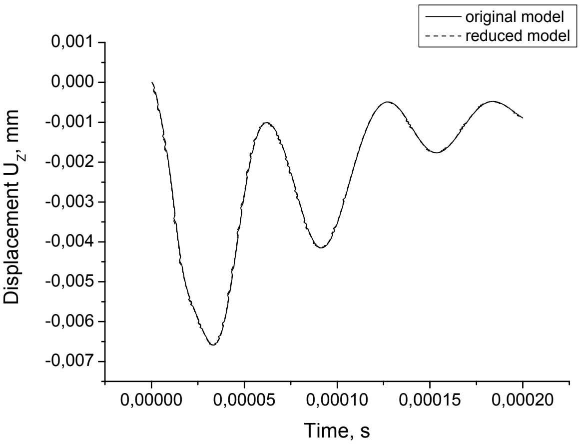

Fig. 4. The transient response U_Z of the IC corner node in z-direction.

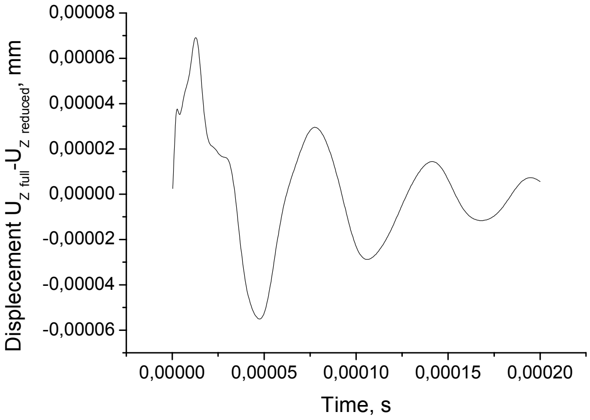

Fig. 5. The difference $U_{Z\ full}$-$U_{Z\ reduced}$ for the transient response of the original and reduced systems from Fig 4.

The time for model reduction was 33 s and it is similar the time for the stationary solution in ANSYS (26 s). At the same time, the time for harmonic analysis in ANSYS to evaluate the transfer function of the original problem (17355 equations) shown in Fig. 3 was almost 6 hours and the time to evaluate the transfer function of the reduced system (30 equations) was just 9 s. The similar ratio in computational time was found for the transient analysis of the original and reduced systems.

4. MECHANICAL STRUCTURE AFFECTED BY NON-UNIFORM TEMPERATURE

In the second case, a local heat source has been introduced. This means that the temperature distribution changes dynamically and is non-uniform within the model. We have modelled a solder joint shown in Fig. 6 to demonstrate this effect. The SOLID5 element type (3-D Coupled-Field Solid) has been used, as in this case it was necessary to introduce both the displacements and the temperatures. The model consists of 1740 elements, 2234 nodes and 8810 equations.

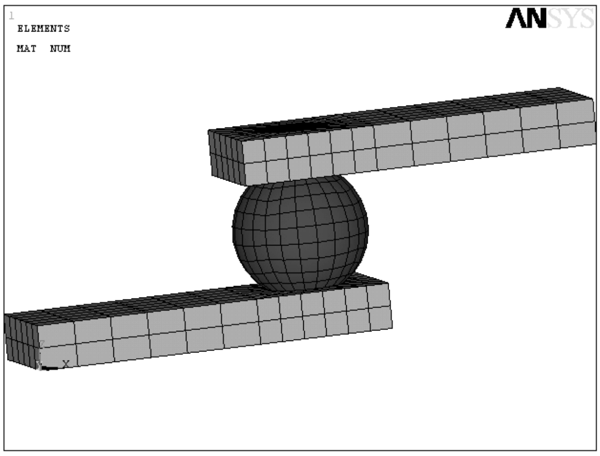

Fig. 6. A solder joint 3D FE- model.

The heat source was located within a solder ball and convection cooling was assumed over the entire surface. The stationary temperature distribution and the solder joints displacement are displayed in Fig. 7.

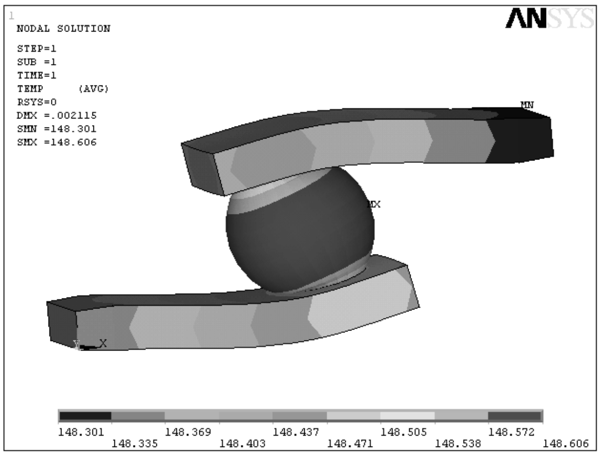

Fig. 7. The temperature distribution in the solder joint under the local heat source.

ANSYS couples a structural and a thermal problem via the load vector, that is, it does not create the coupling matrix K_{UT} explicitly. We needed to generate this matrix by ourselves from element load vectors by assuming that the element thermal strain is caused by an average element temperature. The transfer function of the original system of dimension 8810 and the reduced system of dimension 30 are shown in Fig. 8. The plot is made for U_Z-displacement of the node 757 (max. displacement). The difference is within the line thickness and the difference plot is presented in Fig. 9. This again shows that the reduced model captures very accurately the dynamic effects of the original model.

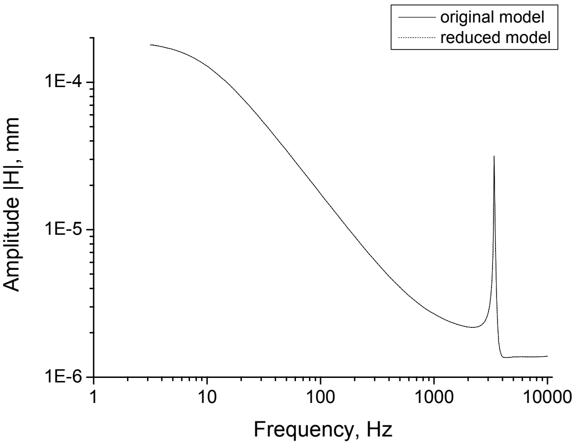

Fig. 8. The transfer function [H| of the 3-D quarter symmetry FE-model for the node 757 (max. displacement) in z-direction.

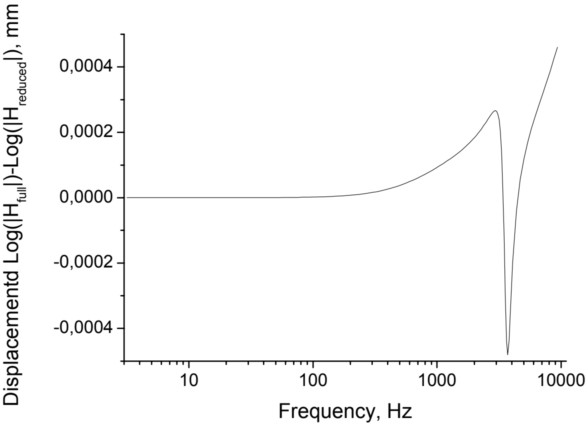

Fig. 9. The difference plot Log(|H_{full}|)- Log(|$H_{reduced}$|) for the curves from Fig. 8.

5. DISCUSSION

We have shown that model reduction is working very well for linear thermo-mechanical models in the case of transient and harmonic simulation. In the case of ANSYS finite element models and the uniform temperature distribution, the model order reduction software **mor4ansys** allows us to apply model reduction directly "out-of-the box". In the case of a local heat source it is necessary to form the coupling matrix K_{UT} in Eq.(2) before the model reduction process. In any case, our results confirm the previous observations that 30 generalized degrees of freedom are usually enough to accurately describe the dynamic behaviour of a high-dimensional finite element model.

Nonetheless, the question how to use model reduction in engineering practice remains open. There are two problems to address. First, thermo-mechanical simulation of packages requires highly nonlinear models. Second, such simulation is usually performed in quasi-stationary mode when the time derivatives of the state vector in Eq (1) and (2) are ignored.

In our view, a linear material behaviour can be assumed during a rapid assessment of parameter sensitivity in the design phase. Then, a detailed nonlinear simulation can be performed specifically for selected values of parameters. This can solve the first problem specified above. On the other hand, even a stationary problem in the case of many design parameters, for example as follows

$$\left(K + \sum_i p_i K_i\right) x = B \qquad (7)$$

requires solution of many systems of linear equations for different values of parameters. This can be also time consuming in the case of a high-dimensional finite element model. At the same time, Eq.(7) is similar to the model reduction problem in the Laplace domain (compare Eqs. 3 and 7) when instead of one Laplace variable we have several design parameters. We believe that model reduction described in the present paper can be successfully generalized to this case [9, 10] and this will addresses the second problem.

6. REFERENCES

[1] A. C. Antoulas, D. C. Sorensen, Approximation of large-scale dynamical systems: An overview, Applied Mathematics & Computer Science 11 (5)(2001) 1093-1121.

[2] E. B. Rudnyi, J. G. Korvink, Review: Automatic model reduction for transient simulation of MEMS-based Devices. Sensors Update v. 11, p. 3-33, 2002.

[3] J. S. Han, E. B. Rudnyi, J. G. Korvink. Efficient optimization of transient dynamic problems in MEMS devices using model order reduction. Journal of Micromechanics and Microengineering 2005, v. 15, N 4, p. 822-832. http://www.imtek.uni-freiburg.de/simulation/mor4ansys/

[4] Thoben, M; Xie, X.; Silber, D.; Wilde, J.: "Reliability of Chip/DCB solder joints in AlSiC base plate power modules: influence of chip size." *Microelectronics Reliability.* vol. 41 (2001) no. 9-10, p.1219-1223.

[5] Zhang, G.Q.; Bisschop, J.; Maessen, P.: "Virtual thermo-mechanical prototyping of microelectronics products - towards optimized designing in reliability", *Advancing Microelectronics,* vol. 28, (2001) no. 1.

[6] Ansorge, F.; Becker, K. F.; Braun, T.; Reichl, H.; Voigt, K.: "Design tools and electrical, thermal and mechanical simulators for creating "mechatronic" systems", *Elektronik,* vol. 50, (2001) no. 20, p.91-96.

[7] E. Zukowski, E. Deier, J. Wilde: Correct modelling of geometry and of materials properties in thethermo-mechanical Finite-Elements-Simulation of Chip Scale Packages. 6th IEEE EuroSimE , Berlin, Germany, 18-20 April, 2005

[8] R. F. Boisvert, R. Pozo, K. A. Remington, The matrix market exchange formats: Initial design, National Institute of Standards and Technology, NIST Interim Report 5935 (December 1996).

[9] L. Daniel, O. C. Siong, L. S. Chay, K. H. Lee, and J. White, A multiparameter moment-matching model-reduction approach for generating geometrically parameterized interconnect performance models, IIEEE Transactions on Computer-Aided Design of Integrated Circuits and Systems, vol. 23, pp. 678-693, 2004.

[10 L. H. Feng, E. B. Rudnyi, J. G. Korvink. Preserving the film coefficient as a parameter in the compact thermal model for fast electro-thermal simulation. IEEE Transactions on Computer-Aided Design of Integrated Circuits and Systems, December, 2005.