The Use of ABAQUS in the Automated Global/Local Thermomechanical Analysis of Multichip Modules

M.S. Shephard, T.-L. Sham, L.-Y. Song, R. Garimella and R.R. Collar

Scientific Computation Research Center

Rensselaer Polytechnic Institute, Troy, NY

ABSTRACT

This paper overviews a set of software procedures for the efficient global and local thermal and thermomechanical analysis of multichip modules. This paper (i) overviews the overall global/local approach, (ii) demonstrates the automatic construction of all numerical analysis models from a CIF file specification, and (iii) overviews an ABAQUS based automated adaptive finite element methodology for the local thermal analysis.

INTRODUCTION

The effective design of a multilayer multichip module must account for its thermomechanical response to ensure the integrity of the device. To address this need a set of global/local analysis procedures capable of effectively performing thermal and thermomechanical analysis have been developed. This paper first overviews an overall approach and the automatic construction of all the analysis discretizations. Attention then focuses on the ABAQUS based adaptive thermomechanical analysis procedure for critical local 3–D regions.

A multilayer multichip module (MCM) is characterized by having a number of layers containing wires and vias in various configurations, in conjunction with a general chip and cooling structure layout. To effectively address the thermal and thermomechanical analysis requirements of these structures a coordinated set of multiple scale analysis techniques have been developed. The global analysis procedures employ averaging techniques to convert each layer into an equivalent homogeneous medium. Both the global thermal and thermomechanical analysis procedures employ a unique variational method that reduces the amount of numerical calculations required to determine the overall fields [Sham *et al.*, 1993, Tiersten *et al.*, 1993]. The local thermal analysis uses a fast floating random walk method [Le Coz and Iverson, 1992]. The local thermal stress analysis employs automated adaptive finite element modeling techniques.

Procedures have been developed to automatically construct the analysis discretizations for the commonly used MCM physical description of a CIF file. Although, this representation does lack a number of features desired in a complete physical description, an augmented version of

it combined, with a knowledge-base can be used to provide an adequate physical description. The geometric model for the local automated adaptive finite element analyses must be a non-manifold solid model derived from the CIF file. Procedures to go from the CIF file to the required non-manifold solid model are given.

Given a complete geometric definition of the selected region of the multichip module, the automated adaptive finite element procedure invokes the Finite Octree mesh generator [Shephard and Georges, 1991] to generate the initial mesh. ABAQUS analyzes the initial mesh. An error estimation procedure that deals with multi-material objects determines where, and by how much, the mesh should be refined. A multiple level refinement procedure then indicates the refinements to the automatic mesh generation procedure which carries out the required refinement through local remeshing. The components of the ABAQUS-based automated adaptive finite element system and example results are presented.

GLOBAL/LOCAL HEAT CONDUCTION AND THERMOMECHANICAL ANALYSIS OF MULTICHIP MODULES

The prediction of a mechanical failure, such as a wire fracture due to thermomechanical loading, is critical to ensuring the ability of an MCM to perform its intended electrical functions. The accurate prediction of such failures must involve the local microstructural details of the interconnect at a potential failure site in a detailed analysis accounting for nonlinear behavior. However, the analysis of the entire interconnect taking into account all microstructural details is computationally prohibitive. An alternative methodology to analyze MCMs is to employ global/local procedures, in which global analyses are performed to provide the boundary condition information needed for an accurate local analysis of critical regions where the microstructural details of wires, vias, etc. are fully represented.

The global analyses employ averaging techniques to convert each layer in the MCM into an equivalent homogeneous medium. The global analysis procedures are based on a variational technique in which the differential equations and minimal layer interface and boundary conditions are satisfied exactly. This allows a closed form integration through the thickness of the interconnect yielding a two dimensional problem independent of the number of layers in the interconnect. The resulting two-dimensional problem is then solved numerically. The requirements on the approximating functions used for the two-dimensional problem lead to the selection of smooth global functions. See references [Sham *et al.*, 1993, Tiersten *et al.*, 1993] for a description of the specific procedures developed for the global heat conduction and thermomechanical analysis of MCMs.

The local heat conduction analysis is performed using a highly efficient stochastic algorithm for solving Laplace equations [Le Coz and Iverson, 1992]. The algorithm is based on the floating random-walk method [Brown, 1956, Haji-Sheikh and Sparrow, 1967]. Briefly put, the algorithm derives from the Laplace solution on a scalable cubic domain, subject to arbitrary Dirichlet conditions. A boundary-integral solution is then found, from which an integral for the temperature at the domain center is obtained. This integral is expanded as an infinite series, and probability rules that statistically evaluate the sum are deduced. These rules define the algorithm which predicts the temperature at a specific point within the windowed local heat conduction domain.

As indicated in the later sections, the local thermomechanical analysis employs automated adaptive finite element methods. In this approach the finite element meshes are automatically generated for the detailed geometric representation in the local area under consideration.

Physical Description and Construction of the Numerical Analysis Models

The starting point of any engineering analysis process is a physical description of the problem being analyzed. For the analysis procedures used here the physical description consists of a geometric domain and the required analysis attribute information.

This domain definition must be complete enough to support the discretization procedures used to generate the analysis models. In addition, it is desirable that this definition be one which is convenient for MCM designers to use. The most commonly used domain description is a CIF file [Hon and Sequin, 1980, Mead and Conway, 1980]. Although convenient, a CIF file does not represent a complete and unique geometric representation of the non-manifold geometries [Weiler, 1988] of an MCM. One alternative is to insist the designer provide a complete non-manifold geometric representation of the MCM. An alternative approach to the definition of the geometric domain, which is the one taken here, is to use the CIF file as the basic information and to develop software procedures to interpret the CIF files to provide the information needed by the automatic analysis discretization procedures used here. Since the automatic finite element mesh generation procedure must operate on a non-manifold solid, specific rules are used to create the required geometric model from the CIF description.

Each of the analysis procedures requires a discretization of the domain of the MCM with the appropriate analysis attributes associated with that discretization. The form of the discretization, and the techniques used to define it depends on the analysis procedure being used. In some cases the discretization is constructed directly from the CIF file, while in others the information in the CIF file is used to construct a more appropriate geometric definition which is then discretized.

Since both global analyses are based on the variational solution procedure, they employ the same basic domain discretization. Starting from a rectangular MCM footprint, the domain discretization required is a layered description of the MCM with layer thickness, wire volume fraction and constituent material properties specified for each layer. In addition, the configuration of the chip layout on the top surface of the MCM must be specified. The chip layout configuration is constrained to rectangular chips with edges aligned with the edges of the MCM.

The local thermal analysis employs a discretization where the conductors within the region of the analysis are represented as rectangular prisms. The dimensions and positions of these prisms are directly extracted from the CIF file by the discretization procedure used by the local thermal analysis.

The finite element procedures of the local thermostress analysis employ the Finite Octree mesh generator [Shephard and Georges, 1991] which is capable of automatically generating graded meshes of general 3–D non-manifold domains. However, to operate Finite Octree requires that the domain be defined within a solid modeling system which supports the required geometric interrogations. To support this need a procedure has been developed to read the CIF file and to use that information to construct a solid model of the local region of interest in a commercial solid modeling system. In the current effort the Parasolid [Sha, 1991] solid modeling system, with extensions for the proper representation of non-manifold domains, has been used. Finite Octree then directly interacts with the geometric modeling system to generate the finite element mesh.

The procedure used to generate the multiple material solid model in an extended solid modeling system starts with a CIF file containing a 2–D description of the MCM layers in terms of rectangles, polygons, circles, and paths. When attributed with a thickness parameter, each such CIF entity can be formed into an appropriate solid primitive in the solid modeling system. The primitives of a single material type are unioned to form the collection of solid bodies of that material type. The complete geometric representation must also include the insulating material around the wires, the solder bumps and the substrate, with a proper representation of the interfaces between them. Although existing solid modeling systems can not provide such non-manifold representations, it is possible to combine their functionalities with extensions to the topological data representation to maintain the additional associativity information needed to produce the non-manifold model, and support the interrogation functions needed to mesh it. The basic steps in this process are:

1. Union all primitives of a given material type to form one or more distinct homogeneous material bodies of the current material type referred to as a material assembly.
2. Consider pairs of material assemblies to determine their interactions by performing appro-

priate modified Boolean operations.

a. If there is no interaction, proceed to the next pair of material assemblies.
b. If their interaction is only at common boundaries, record the specific interactions, which are currently represented by duplicate entities.
c. If their interactions include domain overlap, and there is a rule as to how to handle the overlay region, apply the rule which must result in only boundary interactions which are recorded.

This step terminates when all possible pairs of material assemblies have been considered.

3. Convert the extended 2–manifold topological assembly representation with duplicate entities to a full non-manifold representation without duplication.

An examination of the functionalities of Parasolid indicated the existence of all the basic geometric operators needed to allow the selective creation and retention of the redundant geometric entities needed for this class of non-manifold model. Operators to support modified union operations are used to combine the individual material assemblies. These operations create sets of coincident faces, edges, and vertices at each of the contact faces. Such redundant topology represents inconsistencies that will cause the automatic mesh generator to fail. By maintaining an understanding of the redundant relationships, the model's topological representation is post-processed to eliminate the redundant entities and construct the correct topological uses needed by a proper non-manifold representation. In the current efforts, the radial-edge topological structure [Weiler, 1988] is constructed. The upper two images in Figure 1 show a Parasolid model of a simple single chip MCM and a close-up in the vicinity of the chip.

Since the local finite element analysis procedure is intended to consider only a small local portion of the MCM in an analysis, the procedure to construct the local non-manifold solid models has been extended to efficiently build the non-manifold solid only within a prespecified rectangular parallelepiped. This construction is easily supported by first intersecting each primitive of interest constructed from the CIF file with the given parallelepiped before performing the required union operations. To make this process computationally efficient, CIF file entities that do not interact with the selected parallelepiped are not included in the primitive construction and Boolean operation processes. For example, layers with z-values that do not overlap the parallelepiped need not be considered, while entities in layers of interest that do not overlap the xy-rectangle of the parallelepiped need not be considered. The two lower images in Figure 1 shows the Parasolid model of two local windows of the one chip interconnect built this way.

With the non-manifold structure automatically constructed using the solid modeler operators, the finite element model is automatically generated by Finite Octree using its operator interface to

the solid modeler (Fig. 2).

The successful performance of the four step analysis process requires the communication of analysis result information for one analysis procedure to the other. The global thermoelastic analysis needs the temperature field information from the global thermal analysis to define the initial strain field. The local thermal analysis needs the temperatures on the boundary of the local region to define the required boundary conditions. The local thermal stress analysis requires the temperature field from the local thermal analysis to define the initial strains, and requires the displacements on the boundary of the local domain from the global thermostress analysis

Since both global analyses employ the same form of expansions, an effective means to provide the global thermal stress analysis with the domain temperature information is to give it the numerical values of the coefficients of the temperature expansion solved for in the thermal analysis. Since the local analyses use entirely different solution techniques from each other and the global analysis, there is no convenient method to provide spatial distribution of the quantities of interest. This restriction does not cause a problem since both of the local procedures can operate effectively using pointwise values of the quantities of interest. Therefore, a pointwise evaluation operator which can evaluate the quantity of interest at any given point in the domain is used.

AUTOMATED ADAPTIVE FINITE ELEMENT PROCEDURE FOR LOCAL THERMOMECHANICAL ANALYSIS

Prediction of mechanical failure of interconnect components such as a wire requires a detailed understanding of the local stress fields in that wire. The most effective means to determine such detailed local information is to employ an automated adaptive finite element methodology. Automated analysis is needed to eliminate the time restrictive, and error prone process of generating valid finite element meshes in multiple material domains local to critical features in the MCM, while adaptive techniques are needed to ensure that the final mesh provides the desired level of solution accuracy.

The main components of an automated adaptive finite element procedure are:

1. an automatic mesh generator that can create valid graded meshes in arbitrarily complex domains
2. finite element analysis procedures capable of solving the given physical problem
3. a posteriori error estimation procedures to predict the mesh discretization errors and to indicate where it must be improved
4. mesh enrichment procedures to update the mesh discretization

Automatic mesh generation and mesh enrichment is performed by the Finite Octree mesh generator outlined below. With an initial mesh generated, the finite element analysis is performed by ABAQUS [Hib, 1985]. The error estimator calculates the solution error and is used by the multiple level refinement procedure to determine where and by how much the mesh is to be refined or coarsened. This information is then used by the Finite Octree mesh generator to enrich the mesh for the next analysis step. This process is continued until the solution accuracy is deemed accurate enough.

Automatic Mesh Generation

The Finite Octree mesh generator [Shephard and Georges, 1991] performs a two step discretization in the creation of a finite element mesh. In the first step the geometric domain is discretized into a set of discrete cells which are stored in a regular tree structure, referred to as the Finite Octree. In the second step the individual cells within the Finite Octree are discretized into finite elements, with specific care to ensure the proper matching to the elements in the neighboring cells. All of the individual operations performed during the mesh generation process are performed using 19 geometric interrogation operators [Georges, 1990], and are performed in such a manner to ensure the result is a valid discretization of the original domain [Schroeder and Shephard, 1991, Shephard and Georges, 1992]. For a more complete technical explanation of the Finite Octree mesh generation procedure the interested reader is referred to reference [Shephard and Georges, 1991].

Error Estimation and Indication

A residual-based error estimator [Baehmann *et al.*, 1992] is used to calculate the solution error for the local thermomechanical analyses. The construction of the error estimator begins with the variational form [Hughes, 1987] for a linear elliptic, or linearized increment of a nonlinear elliptic problem. This form can be stated as find $\underline{u} \in \mathcal{H}^1_E(\Omega)$ such that

$$A(\underline{u},\underline{v})_\Omega = (\underline{f},\underline{v})_\Omega + \langle\hat{\underline{\sigma}},\underline{v}\rangle_{\partial\Omega_2}, \quad \forall\ \underline{v} \in \mathcal{H}^1_0(\Omega) \subset \mathcal{H}^1(\Omega) \tag{1}$$

where

$$A(\underline{u},\underline{v})_\Omega = \int_\Omega \left(\nabla\underline{v}^{\mathrm{T}}\underline{a}\nabla\underline{u} + \underline{v}^{\mathrm{T}}\underline{b}\underline{u}\right) dxdydz$$

$$(\underline{f},\underline{v})_\Omega = \int_\Omega \underline{v}^{\mathrm{T}}\underline{f}\,dxdydz, \qquad \langle\hat{\underline{\sigma}},\underline{v}\rangle_{\partial\Omega_2} = \int_{\partial\Omega_2} \underline{v}^{\mathrm{T}}\hat{\underline{\sigma}}\,dxdydz$$

an underscore denotes a vector or matrix, Ω is a bounded region, $\partial\Omega$ is the boundary of Ω, $\underline{a}(x,y,z) > 0$ and $\underline{b}(x,y,z) \geq 0$ are smooth functions on Ω and $\hat{\underline{\sigma}}$ is given on $\partial\Omega_2$. The subscripts E and 0 further restrict functions to $\hat{\underline{u}}$ or 0 on $\partial\Omega_1$, respectively.

Finite element approximations of equation (1) consist of determining $\underline{U} \in \hat{\mathcal{H}}^1_E(\Omega) \subset \mathcal{H}^1_E(\Omega)$ such that

$$A(\underline{U},\underline{V})_\Omega = (\underline{f},\underline{V})_\Omega + \langle\hat{\underline{\sigma}},\underline{V}\rangle_{\partial\Omega_2}, \quad \forall\ \underline{V} \in \hat{\mathcal{H}}^1_0(\Omega) \subset \mathcal{H}^1_0(\Omega) \tag{2}$$

where $\hat{\mathcal{H}}^1(\Omega)$ consists of continuous piecewise polynomials defined over individual elements of a partition of Ω.

To find an expression for the error, replace $\underline{u}$ in equation (1) by $\underline{U}+\underline{e}$, where $\underline{e}$ is the discretization error to yield

$$A(\underline{e},\underline{v})_{\Omega_i} = (\underline{f},\underline{v})_{\Omega_i} - A(\underline{U},\underline{v})_{\Omega_i} + \langle\underline{a}\underline{u}_{\mathrm{n}},\underline{v}\rangle_{\partial\Omega_i}, \qquad \forall\ \underline{v} \in \mathcal{H}^1_0(\Omega_i) \tag{3}$$

where $\underline{a}\underline{u}_n$ is replaced with $\hat{\underline{\sigma}}$ when $\partial\Omega_i$ coincides with $\partial\Omega_2$. Approximations to the error $\underline{e}$, symbolized as $\underline{\mathrm{E}}$, are obtained by solving equation (3) in a finite-dimensional subspace $\tilde{\mathcal{H}}^1(\Omega_i)$ of $\mathcal{H}^1(\Omega_i)$. Typically, $\tilde{\mathcal{H}}^1(\Omega_i)$ consists of polynomials of higher degree than those used for $\hat{\mathcal{H}}^1(\Omega)$. Babuska and Yu [Babuska and Yu, 1987] have studied the contributions of the error terms in equation (3) and have shown that the interior term $(\underline{f},\underline{V})_{\Omega_i} - A(\underline{U},\underline{V})_{\Omega_i}$ in equation (3) is the dominant term of the error when $\hat{\mathcal{H}}^1(\Omega)$ consists of even order functions.

The error estimation used here, and presented in reference [Baehmann *et al.*, 1992], employs these results to estimate the error $\underline{\mathrm{E}}$ of piecewise quadratic approximations on meshes of tetrahedral elements. The approach taken is to calculate only the interior residual portion of the error on the elements. By neglecting the boundary term in equation (3), the calculation of the error becomes

$$A(\underline{\mathrm{E}},\underline{V})_{\Omega_i} = (\underline{f},\underline{V})_{\Omega_i} - A(\underline{U},\underline{V})_{\Omega_i}, \qquad \forall\ \underline{V} \in \tilde{\mathcal{H}}^1_0(\Omega_i) \tag{4}$$

which is a local problem that can be solved for one element i at a time. $\tilde{\mathcal{H}}^1_0(\Omega_i)$ is a finite-dimensional subspace of $\mathcal{H}^1_0(\Omega)$ containing functions that are of a higher polynomial degree than those in $\hat{\mathcal{H}}^1_0(\Omega)$ used for the finite element solution. The space $\hat{\mathcal{H}}^1_0(\Omega)$ spanned by the piecewise polynomials are complete quadratics in both two- and three-dimensions. In three-dimensions the space $\tilde{\mathcal{H}}^1_0(\Omega_i)$ is spanned by quartic and quintic basis

$$\begin{aligned}
\psi^*_1 &= \xi_1\xi_2\xi_3\xi_4 = \xi_1\xi_2\xi_3(1-\xi_1-\xi_2-\xi_3),\\
\psi^*_2 &= \xi_1\psi^*_1 = \xi_1^2\xi_2\xi_3(1-\xi_1-\xi_2-\xi_3),\\
\psi^*_3 &= \xi_2\psi^*_1 = \xi_1\xi_2^2\xi_3(1-\xi_1-\xi_2-\xi_3),\\
\psi^*_4 &= \xi_3\psi^*_1 = \xi_1\xi_2\xi_3^2(1-\xi_1-\xi_2-\xi_3)
\end{aligned} \tag{5}$$

where (ξ_1, ξ_2, ξ_3) are volume coordinates and the redundant volume coordinate is written in terms of the other area coordinates as $\xi_4 = 1 - \xi_1 - \xi_2 - \xi_3$. The individual shape functions ψ^* are "bubble" functions that vanish on element boundaries. The restriction of $\underline{\mathrm{E}}$ to element i has the form

$$\underline{\mathrm{E}}_{\Omega_i} = \sum_{j=1}^{4} C_{ij} \psi_j^* \tag{6}$$

The error determined from equations (4) and (6) may be used to obtain the local elemental errors in strain energy

$$\left\| \underline{\mathrm{E}}_j \right\|_{\Omega_i}^2 = \frac{1}{2} A(\underline{\mathrm{E}}, \underline{\mathrm{E}})_{\Omega_i} \tag{7}$$

and the global error in strain energy for each material j

$$\left\| \underline{\mathrm{E}}_j \right\|^2 = \sum_{i=1}^{\mathrm{N_j}} \left\| \underline{\mathrm{E}}_j \right\|_{\Omega_i}^2, \qquad j = 1, \text{number of materials} \tag{8}$$

where N_j is the total number of elements in material j in the mesh.

Multiple Level Refinement Procedure and Local Remeshing

The element errors are used to determine where and how to refine or coarsen the mesh. The allowable element error is determined by

$$\left\| \underline{\mathrm{E}}_j \right\|_a = \frac{\eta_j^*}{100} \left(\frac{\left\| \underline{U}_j \right\|^2 + \left\| \underline{\mathrm{E}}_j \right\|^2}{\mathrm{N}_j} \right)^{1/2} \tag{9}$$

where η_j^* is the user supplied error tolerance, eg. 5% error. The error on each element is examined and the new desired element size is calculated as

$$h_i^{new} = h_i^{old} \left(\frac{\left\| \underline{E}_j \right\|_a}{\left\| \underline{E}_{\Omega_i} \right\|} \right)^{1/2} \tag{10}$$

This information can be provided directly to an element refinement or remeshing procedure. However, it has been found that when the requested element sizes are substantially different from that of the original data, the new mesh defined with this data will not have an optimal refinement distribution. The basic problem is that the information provided by equation 10 represents a discontinuous distribution of requested element sizes over the domain. A more continuous distribution of element sizes is constructed from this information by employing the hierarchy of mesh entities as well the relationship, referred to as classification, of the mesh entities with respect to the original model.

In basic terms, the element sizes are assigned to mesh entities by considering the hierarchy from mesh vertices, to mesh edges, to mesh faces, to mesh regions. By considering the element sizes requested for the mesh regions using a particular mesh entity, the requested element size is determined. Since the critical regions are at domain boundaries (the interface between material regions is part of the boundary) the assignment of the requested element sizes for mesh entities classified on the boundary take specific account of that factor using the smallest element size requested by any mesh region using that mesh entity.

Once the distribution of desired element sizes has been determined, the mesh needs to be updated. In the current procedure this is done through functionality of the Finite Octree mesh generator. This process is made efficient by directly employing the underlying octree structure to perform local remeshing. When the desired element sizes indicate the elements sizes change enough to alter the tree level, it is altered and elements within the altered octants and their edge and face neighbors are regenerated. The unaffected octants maintain the mesh from the previous step.

The use of this intelligent multiple level refinement procedure has been found computationally effective. The procedure typically obtains the desired level of accuracy in one adaptive step, with some problems requiring two. The mesh has a nearly optimal distribution of elements since the convergence rate obtained is nearly equal to the best possible for an h-refinement method.

Test examples indicate that the overall process is computationally efficient. In cases where one adaptive step is needed, the initial mesh generation, the initial analysis, error estimation, element size distribution determination and remeshing are less than 30% of the total computational effort, with the other 70% of the computational effort in the analysis of the final mesh. In cases where two adaptive steps were required, around 60% of the total computational effort was still in the final analysis. Since the final mesh has an optimal distribution of elements, it is likely that even when a user was to spend the time and effort needed to interactively define mesh gradations, the total computational effort required for the solution of a mesh that gave the same accuracy would be more than that of the entire automated adaptive procedure.

Example

Figure 3a shows the three-dimensional model of a highly local region that was targeted for thermalstress analysis in a 25chip MCM. The dielectric in the model is polyimide and, the wires, ground plane and power plane are made up of copper. Although the ground and power planes are, in reality, separated by a very thin ceramic layer, this has not been modeled here. Figure 3b shows the automatically generated initial mesh. The contour plot in figure 3c shows the mises stress distribution after the first analysis. Note that the visualization procedure used does not smooth stresses across material interfaces, leading to the observed discontinuities in contours.

Figure 3d shows the mesh after adaptive refinement following analysis, error estimation and mesh prediction. As expected, the mesh is refined in areas with large stress gradients and the peak stress predicted has increased by 10%. Table 1 shows that the strain energy for each material region has decreased in the refined mesh. This is the expected direction of change since this problem was formulated with Dirichlet boundary conditions.

CLOSING REMARKS

This paper has outlined a general global/local solution methodology for heat conduction and thermomechanical analysis of MCMs. The complexities of an MCM clearly indicate the need for the development of a global/local analysis methodology. Emphasis in the development of the current global/local methodology has been placed on highly effective analysis procedures for each component analysis, and on the complete integration of these techniques through the use of automated discrete analysis model construction techniques. This paper has emphasized the process of automatically constructing the analysis discretizations using the information in a CIF file, and outlined the automated adaptive finite element techniques used as one of the local analysis techniques. References [Le Coz and Iverson, 1992, Sham *et al.*, 1993, Tiersten *et al.*, 1993] contain a more detailed description of the global variational approximation procedures, and provides results of the individual techniques as well as the global/local solution process.

ACKNOWLEDGMENTS

This work was supported by Wright-Patterson AFB under Contract No. F333615-91-K-1717. The ABAQUS finite element program was made available under academic license by Hibbitt, Karlsson & Sorensen, Inc.

REFERENCES

Ivo Babuska and Dehao Yu. Asymptotically exact a-posteriori error estimator for biquadratic elements. *Finite Elements in Analysis and Design*, pages 341–354, 1987.

Peggy L. Baehmann, Mark S. Shephard, and Joseph E. Flaherty. A posteriori error estimation for triangular and tetrahedral quadratic elements using interior residuals. *Int. J. Numer. Meth. Engng.*, 34:979–996, 1992.

G. M. Brown. Monte carlo methods. In E. F. Beckenbach, editor, *Modern Mathematics for Engineers*. McGraw Hill, NY, 1956.

M. K. Georges. Geometric operators for the Finite Octree mesh generator. Technical Report SCOREC Report # 13-1991, Scientific Computation Research Center, Rensselaer Polytechnic Institute, Troy, NY 12180-3590, 1990.

A. Haji-Sheikh and E. M. Sparrow. The solution of heat conduction problems by probability methods. *Trans. ASME*, C-89:121–131, 1967.

Hibbitt, Karlsson and Sorensen, Inc., 100 Medway Street, Providence, RI. *ABAQUS Theory and User's Manual*, July 1985.

R. W. Hon and C. H. Sequin. A guide to lsi implementation, 2nd edition. Technical Report SSL-79-7, XEROX, Palo Alto, CA, 1980.

T. J. R. Hughes. *The Finite Element Method: Linear Static and Dynamic Finite Element Analysis*. Prentice Hall, Englewood Cliffs, NJ, 1987.

Y. L. Le Coz and R. B. Iverson. A stochastic algorithm for high speed capacitance extraction in integrated circuits. *Solid State Electronics*, 35:1005–1012, 1992.

C. Mead and L. Conway. *Introduction to VLSI Systems*. Addison-Wesley Publ. Co., Reading, MA, 1980.

W. J. Schroeder and M. S. Shephard. On rigorous conditions for automatically generated finite element meshes. In J. Turner, J. Pegna, and M. Wozny, editors, *Product Modeling for Computer-Aided Design and Manufacturing*, pages 267–281. North Holland, 1991.

Shape Data Limited, Parker's House, 46 Regent Street, Cambridge CB2 1DB England. *PARASOLID v4.0 Programming Reference Manual*, August 1991.

T.-L. Sham, H.F. Tiersten, P.L. Baehmann, L.-Y. Song, Y.S. Zhou, B.J. Lwo, Y.L. Le Coz, and M.S. Shephard. A global-local procedure for the heat conduction analysis of multichip modules. In P. A. Engel and W. T. Chen, editors, *Advances in Electronic Packaging 1993*, volume 2, pages 551–562, New York, NY, 1993. ASME.

M. S. Shephard and M. K. Georges. Automatic three-dimensional mesh generation by the Finite Octree technique. *Int. J. Numer. Meth. Engng.*, 32(4):709–749, 1991.

M. S. Shephard and M. K. Georges. Reliability of automatic 3-D mesh generation. *Comp. Meth. Appl. Mech. Engng.*, 101:443–462, 1992.

H.F. Tiersten, T.-L. Sham, B.J. Lwo, Y.S. Zhou, L.-Y. Song, P.L. Baehmann, Y.L. Le Coz, and M.S. Shephard. A global-local procedure for the thermoelastic analysis of multichip modules. In P. A. Engel and W. T. Chen, editors, *Advances in Electronic Packaging 1993*, volume 1, pages 103–118, New York, NY, 1993. ASME.

K. J. Weiler. The radial-edge structure: A topological representation for non-manifold geometric boundary representations. In M. J. Wozny, H. W. McLaughlin, and J. L. Encarnacao, editors, *Geometric Modeling for CAD Applications*, pages 3–36. North Holland, 1988.

TABLES

Region name	Strain Energy	
	Initial mesh	Refined mesh
Dielectric	0.3530E-08	0.2938E-08
Ground plane	0.1699E-07	0.1441E-07
Power plane	0.1595E-07	0.1401E-07
Wire 1	0.1872E-09	0.1393E-09
Wire 2	0.1970E-09	0.1160E-09
Wire 3	0.1279E-09	0.0799E-09
Wire 4	0.1634E-09	0.0796E-09

Table 1 Strain energy of material regions in the initial mesh and refined mesh

FIGURES

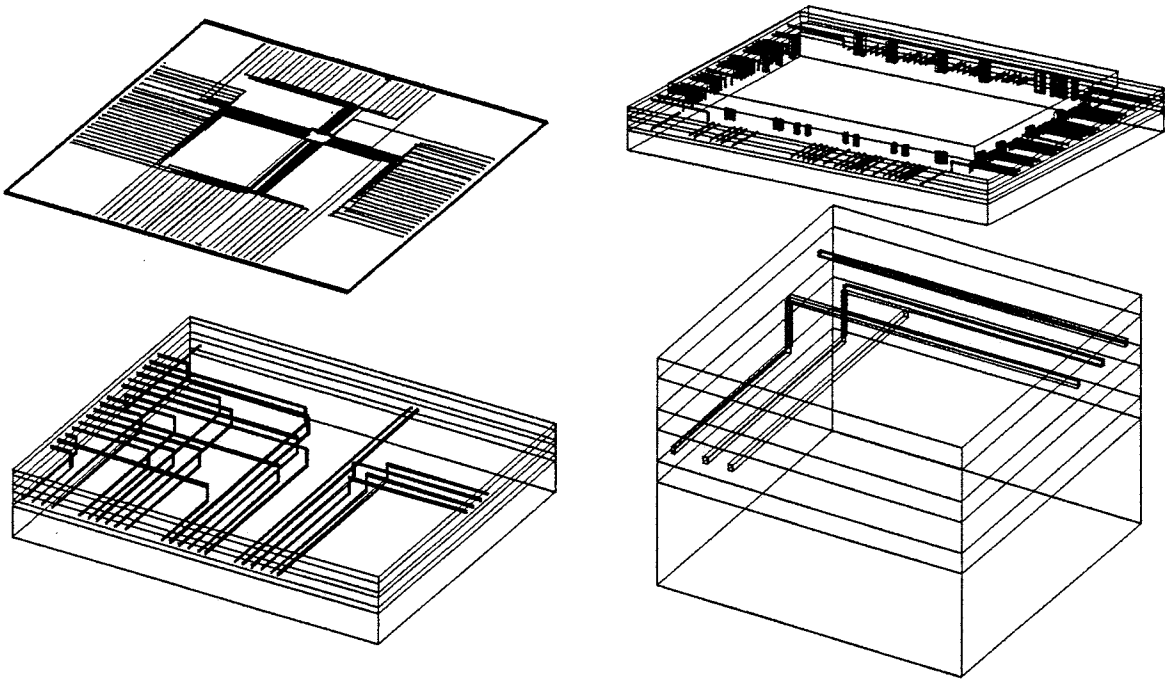

Figure 1. Views of a Parasolid model of a simple one chip interconnect created from a CIF file.

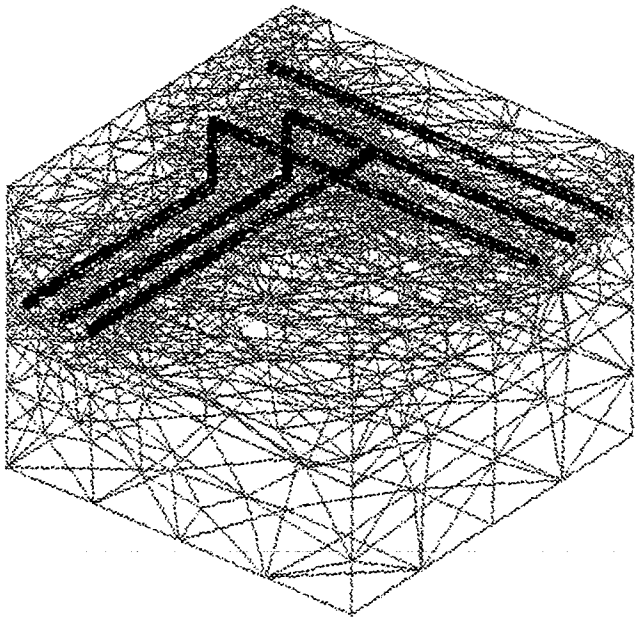

Figure 2. Finite element mesh of the non-manifold model within the smallest local analysis window.

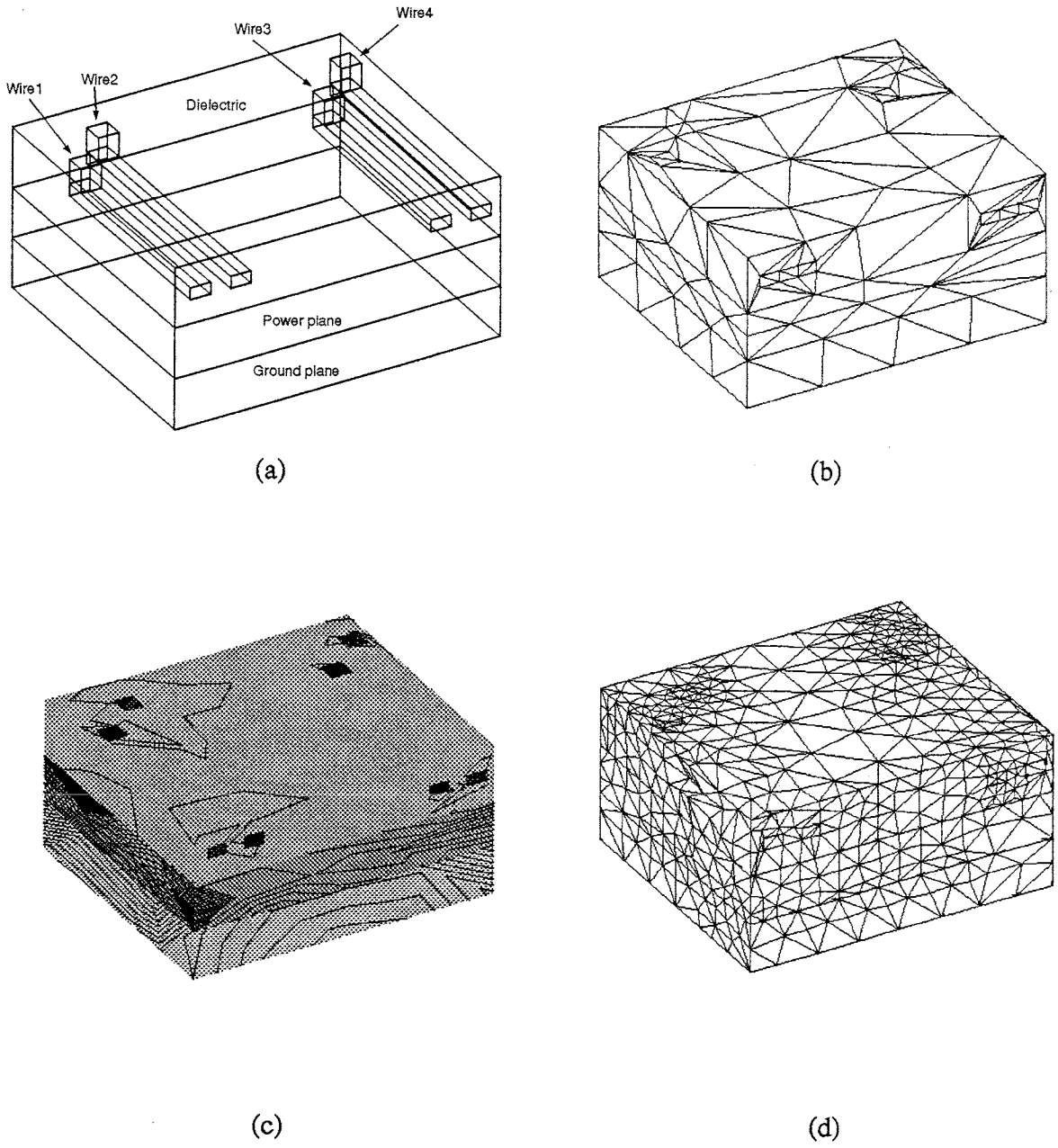

Figure 3. Three-dimensional example with initial and adaptively refined finite element meshes (a) Geometric model of local window (b) Initial mesh of local model (c) Contour plot of mises stress after first analysis (d) Adaptively refined mesh