1997 INTERpack
Innovations in CAD/CAE Integration in Electronic Packaging
Kohala, HW, June 15-19, 1997

CAD-BASED ANALYSIS TOOLS FOR ELECTRONIC PACKAGING DESIGN
(A New Modeling Methodology for a Virtual Development Environment)

Wen X. Zhou*, Chien H. Hsiung, Robert E. Fulton
School of Mechanical Engineering,
Georgia Institute of Technology,
Atlanta, GA 30332, U.S.A

Xun Fei Yin
Hydropower Planning and Design General Institute,
MWR
Beijing 100011, P. R. China

Chao-Pin Yeh, Karl Wyatt
Corporate Software Research & Development,
Motorola, Inc.
Schaumburg, IL 29616, U.S.A

ABSTRACT

An integrated Finite Element Modeling (FEM) Methodology has been addressed for concurrent mechanical design of electronic products. The new modeling methodology (MPI/FEM) consist of three core modeling approaches: Modularized FEM (M/FEM) , Parametric FEM (P/FEM), and Interactive FEM (I/FEM). These approaches have been investigated, and then integrated into conventional FEM code. A component FE model library can be easily created by modularizing the modeling concept. The main idea of parametric modeling is to create or define a FE model template, instead of a detailed FE model by using parameters and their forming rules. The Interactive FEM approach allows the designers to analyze and visualize design models interactively. A FE model created by these approaches has the advantages of flexibility, compatibility and portability. A multi-level solution approach is applied to produce solution results. From the study results, it has been concluded that the developed methodology is a vital tool for thermomechanical design, and can be applied to the development of a virtual design environment for electronic products.

1. INTRODUCTION

Due to the lack of an appropriate mechanical modeling methodology, mechanical reliability analysis of an electronic packaging product is generally performed by mechanical experts only after electrical design [2]. Most of the available MCAD systems are used primarily for drafting, layout and parts lists, and only marginally affect the realm of engineering analysis [4,17]. The mechanical finite element (FE) model building phase generally takes up 80~90% of total analysis time. In order to reduce costs and speed up the introduction of new products with higher performance, both mechanical CAD tools and electrical CAD tools have to be integrated into a TCAD (Technology CAD) system to provide concurrent design capability. TCAD is a form of computer-based modeling used by a product designer for more efficient design processes. To respond to the growing needs of mechanical modeling and simulation in the electronic design process, a new modeling methodology (MPI/FEM) has been developed. MPI/FEM integrates three new modeling approaches: Modularized, Parametric, and Interactive approaches. It can be utilized to rapidly create a CAD-based mechanical analysis module for reliable thermo-mechanical analysis.

2. NEEDS AND ISSUES FOR INTEGRATED E/CAD AND M/CAD.

2.1 The Bridge to a Virtual Development Environment

Traditionally, the product development process can be described in terms of design-prototyping-testing-modification [19]. Electrical function design is the primary activity in the design process. Thermal or thermomechanical design issues are roughly checked during, or after the physical design process. Intensive thermomechanical design and analysis are involved only after problems are observed either on the manufacturing line or in reliability testing. In many cases, due to the time pressure of shipping products or the high cost of design modification, only short-term solutions, or no solution, can be obtained at this stage. In order to balance the design requirements and the cost of design modifications, the short-term solutions are obtained by "trail-and-error" methods. Such an approach is often costly and responsible for significant time delay in the development process, and sometime ineffective in assuring product reliability throughout the life cycle. Since the cause of failure may not be fully understood, the same failure mechanism may appear again on another new product. Thus, the life cycle cost and time-to-market are increased.

Leading electronics companies envision a virtual development environment that allows electrical engineers to collaborate with

* Current Address: Applied Simulation & Modeling Research, Corporate Software R & D, Motorola, IL 60196

mechanical engineers to design, simulate, optimize components, and to study their integrity and reliability from many aspects [2,12,13]. The drive behind obtaining this virtual environment results from the need to reduce time-to-market, decrease development costs, and improve the quality of the finished product.

2.2 Mechanical Design Limitation

To integrate mechanical design aspects with electrical design, three major limitations exist in the general product development cycle. First, it is difficult for an electrical engineer to perform thermomechanical design. Second, it takes a mechanical engineer too much time to find both the needed information and to perform a mechanical analysis. Finally, the mechanical design and modeling capabilities are greatly limited by currently available methodologies.

The generality and versatility of the FEM enable an engineer to tackle complex engineering analysis problems, but its usefulness and efficiency are largely dependent on a designer's experience. A customized FEM package intended for electronic packaging designs has to be developed to provide fast and applicable mechanical design models in an user-friendly environment.

2.3 Lack of Design Data Sharing Capability

The integration of multi-disciplinary processes of an entire product development is highly dependent on the communication of data, which includes all information associated with the product. The dataflow includes the information generated from components (parts) vendor, component library inventory, electrical design tools, physical design tools, thermomechanical design tools, and manufacturing processes. The components and product related data can currently only be transferred within a single disciplinary design environment. When it is transferred from one disciplinary tool to another, a special translator has to be used for either filtering or mapping the data from one format to anther. During the translation, part of the product information is lost. Some of the information can not be recovered, although it could be very valuable for later designs and analyses.

2.4 Gap Between Design Tool and Analysis Tool

Integration of MCAD with numerical analysis tools is another important issue in design automation. Most of the available MCAD systems are used primarily for drafting, layout, and parts lists, but only marginally affect the realm of an engineering analysis [17,18,20,25]. CAD programs, which are developed by the means of conventional information processing technologies, rely mostly on procedural representation of 3-D objects, and do not have the capability to perform analysis tasks such as finite element analysis. Although geometry information "translators" are available in major CAD and FEA software, the product design information, received by a FEA preprocessor through the translators, is not very useful. This is because: 1) Only geometry information (wire-frame, surface, solid) is transferred, while a lot of other information such as the concept of object attributes, the information of properties, the descriptions of behaviors and functions, are lost; 2) Direct transformation of a design geometry, from one CAD package to a FEA software, includes too much minute and/or non-critical geometry, which make the computational analyses practically impossible.

The gap between the MCAD systems and analysis tools led to the emergence of new technologies into existing MCAD systems. These new technologies include knowledge-based engineering, object-oriented framework, computer algorithms, and standard definition of product information. The desired system should be able to capture both geometric and non-geometric product information, and provide different levels of product information. This information should include: components list; their geometric relation and inheritance hierarchy; geometric information to represent the shape; structural characteristics; information to represent attributes and properties; description of function; engineering judgment and analysis simplification for typical geometry; engineering rules; manufacturing constraints, and so on.

3. MODULARIZED FEM TECHNIQUE

3.1 Modularized Geometry Primitives (MGPs) Concept

A quick way of defining the modularization concept is refer to group technology. Its essential argument is that many electrical components and/or products can be grouped into classes or families of similar shapes. Each single family of topological shapes is called a Modularized Generic Primitive (MGP). A J-lead in a TSOP component is a general MGP. Individual members of a family can be distinguished by a few parameters. A new geometric shape can be created by linear transformations of an existing one. The transformations affect only the geometry size, but not the topology of a shape. Each MGP is defined topologically instead of as a detailed geometry. It may be used for creating different TSOP component models, since they have the same geometric topology. This idea is illustrated in Figure 1. The overall relationship among MGPs, components, assembly, and product can be represented as a graph tree. This tree is referred to as a Constructive Module Assembly Tree (CMAT). The CMAT is an undirected graph, or a rooted tree, where the root is the electrical product itself.

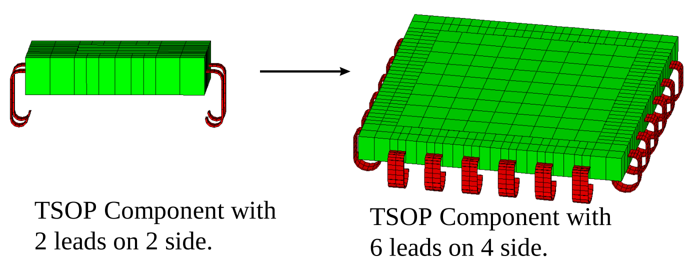


Figure 1. Two TSOP components with different configuration can be created by same component Module

3.2 Taxonomic Technique

In product design, most products are not started from scratch, but are instead chosen from an electronic component library, which contains a limited number of basic electronic components (functional modules) [3,7,10,11,21,22] such as a chip, a resistor, or a capacitor. Most of the substrate, or the interconnection, can be limited to Plate Through Hole (PTH), Surface Mount Technology (SMT), Wirebonding (WB), Taped Automatic Bonding (TAB), Flip Chip, or Ball Grid Array (BGA). Generally, there are a limited number of components used to fabricate a product, as well as limited types of interconnections in a component assembly. In this research work, only a few SMT components are studied, and they can be classified into several categories. Note that these approaches may be extended to PTH and MCM technology. According to the characteristics of geometry and topology of electrical components, the defined categories include: (1) active, (2) passive, (3) chip-level attachment, (4) BGAs, (5) footprint, (6) vias, (7) solder joint formation. The taxonomy of active components is showed in Figure 2. Other components, such as switches, fuses, connectors, and lamps, are omitted in this research work, because they generally have little design and reliability implications in mechanical design scenarios.

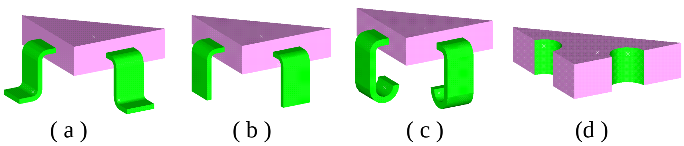


Figure 2. Taxonomy of active component: (a) Gull-lead; (b) I-lead; (c) J-lead; (d) Leadless

3.3 Library of FE Models for Electronic Components

Since the component model saved in a CAD system preserves the geometry and topology, the same model and its associated mesh information can be reused, as long as the physical properties remain invariant. When the finite element analysis for a component is required, the user may request the information stored in the CAD system by inputting a Constructive Module (CM) tree for that component. Hence, the model of the component, once created with the help of a solid modeler, can be stored in a CAD system, and reused for another component that is geometrically and topologically equivalent to this master model.

In mechanical analysis, these electronic products may be treated as an assembly of typical components, which can be classified and saved into a mechanical component library. Each component can be defined as a finite element module. This module is different from a meshed FE model, because its geometry, mesh shape, and density are created by parameters and form rules. Each module may have different types of analysis capabilities, such as static-structural, static-heat transfer, transient heat transfer, or modal analysis. Each module may be further encapsulated into a higher level module. Using these standard parts (models) will reduce the number of components that must be generated from scratch. The original model-primitives in the component library, after initial construction, will never have to be remodeled. These model-primitives are generated and validated by experts, and later can be used by the product designer to create a model for predictive analysis.

3.4 Modularized representation for a BGA FE Model

This section gives an example of modularized FE modeling of a BGA structure. It will show the flexibility of a BGA FE model. The parts needed to create a BGA interconnection are:

- Chip(0-level) substrate
- upper solder pad for eutectic solder
- upper eutectic solder connection
- Non-eutectic solder ball
- lower eutectic solder connection
- lower solder pad for eutectic solder
- Board (1st-level) substrate

These parts are defined (Figure 3) and stored in a library for a BGA model assembly. Each part of a geometric shape and mesh grid are flexibly controlled by build-in parameters. For example, the solder ball may be stretched vertically or horizontally to form a taller or shorter elliptical ball.

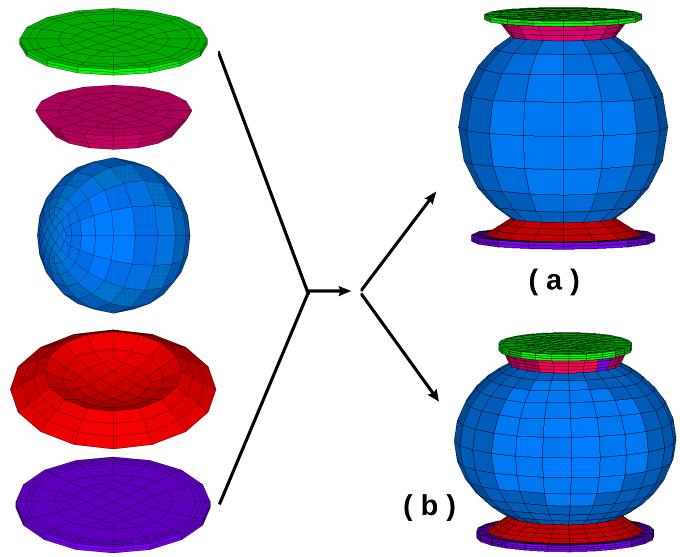


Figure 3. Modularized parts for a BGA modeling.
Figure 4. Two BGA FE model based on a single modularized FE model

The modularized parts in a library are virtual models without real detail geometry data for the parameters. Some parameters are user inputs, while others depend on the parameter values of neighboring parts. Construction of two BGA connections with different configurations are shown on Figure 4. In each construction process, the parts are called one at a time to assemble a BGA connection. The geometry relations and element connections are maintained by a graph tree. All the parts of the two BGAs have different configurations and mesh densities. In Figure 4 (b), the BGA has a shorter and wider ball with a fine mesh, and in Figure 4 (a) is a perfect sphere with a coarse mesh. The configuration and mesh distribution for each part of a BGA model may be modified even after model creation. Other connected parts will be modified automatically and simultaneously, since they are all controlled by the graph tree.

4. PARAMETRIC FEM APPROACH

4.1 Feature-Based Parametric Representation.

The parametric modeling approach has several advantages, which include flexibility, interchangeability, and portability. Flexibility of a model implies that the model may be modified easily to create a new one in the initial design process, but may also be able to answer "what if" questions during the design analysis process. Interchangeability refers to the compatibility of unitizing a model, which may be modified later, if necessary. Since there is no data, or number, employed in the procedure for defining a model template, the processes of actual model creation and model definition are separated. Model hierarchy and abstraction can be easily achieved. Internal changes of a model template and model encapsulation can be independently implemented without altering the manipulating methods and techniques for the model. Transportability suggests that the developed model templates are independent of either software platforms or design applications. The development of a template involves only the definitions of forming rules and parameters used for describing these rules. The rules are created by design applications, not the software actually implementing it. Thus, the model defined by a model template, in a software environment, can be readily transferred to another environment.

4.2 Parametric Representation for FE Modeling

As mentioned in previous sections, a geometric solid model is used to help a designer visualize the design concept, but it can also be used for more comprehensive analysis and prediction. The same geometric model, however, has to be able to perform different design studies. In order to achieve this, not only geometric parameters, but also analysis parameters have to be built into the model template.

As a completed model, which can perform a specified FE analysis, the following parameters have to be explicitly defined:

- element types
- element attributes
- initial condition
- material properties
- loading
- solution criteria

This information has to be provided in parametric form to allow flexible changes and reuses.

4.3 Parametric Representation for MGPs

Modularized generic primitives (MGPs) introduced in Section 3.1, are parametrically modeled for a one-to-many modeling capability. The modeling process involves three steps:

1. Description of a MGP geometry variations
2. Definitions of parameters for both geometry and FE modeling
3. Construction and testing of a MGP

During the geometry shape definition, the flexibility of a MGP has to be explicitly specified. Though a MGP can be stretched to form a new shape, it is not yet constrained. The more the flexibility of a MGP, the more difficult it is to define constrain rules. Consequently, the structure of a MGP becomes more complex and difficult to model. The definitions of parameters for the geometry and FE modeling are primarily dependent on the needs of MGPs in later design and analysis. The implementation of construction and testing for a MGP, in the final step, may vary on different software platforms, but the concepts and procedures remain the same.

It is important to note that both the component module and its pre-defined parameters are saved in the component library. The parameters include not only geometric parameters, but also parameters for materials, mesh control, analysis type, and assembly control. These parameters are categorized and linked with different resources. For example, the geometric parameters are linked with a user's input, the material parameters are linked with a database, etc. Figure 5 shows a TSOP component and its different data groups. The geometric parameter for a TSOP component with gull-lead can be input by users through a GUI window, shown in Figure 6.

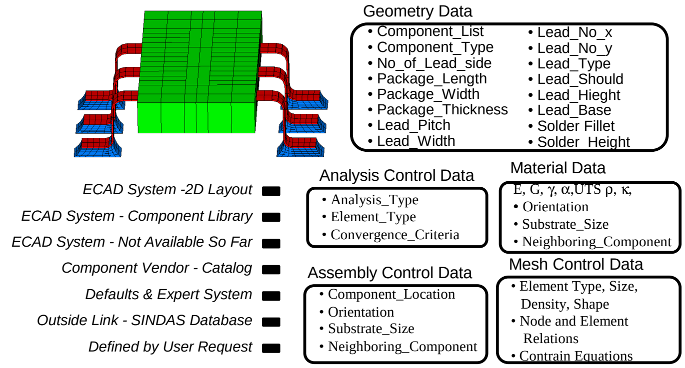


Figure 5. A TSOP component and its parameters (data) group

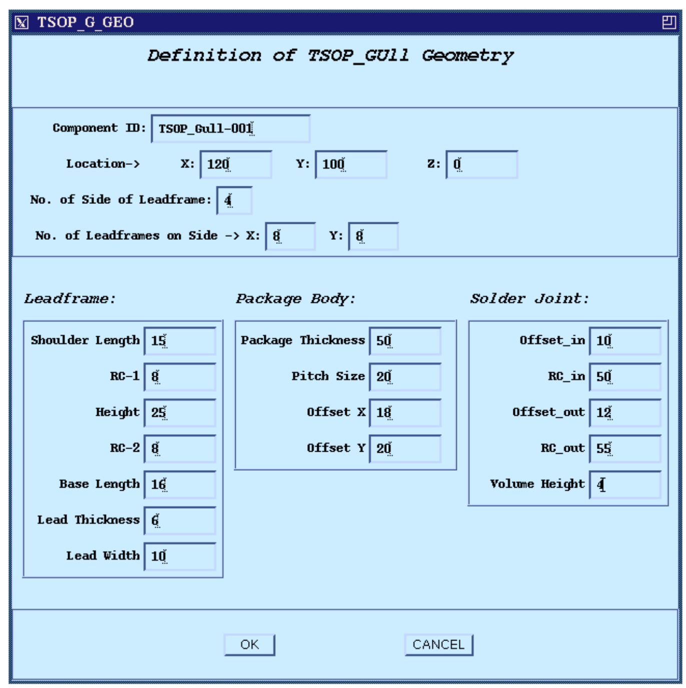


Figure 6. Definitions of parameters for a TSOP component with a gull-lead

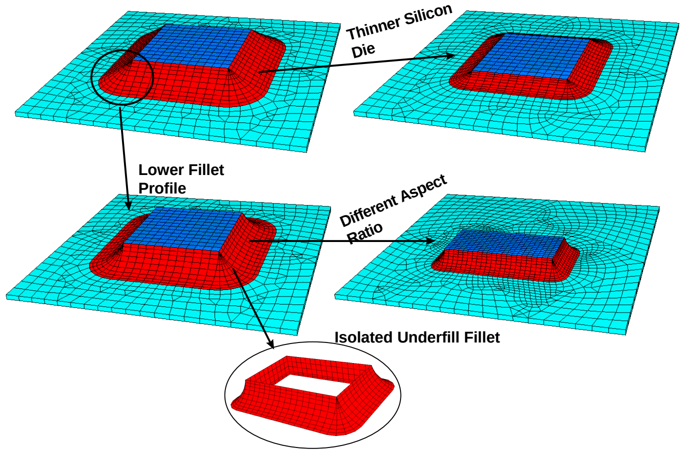


Figure 7. Parametric Modeling for DCAs

Parametric modeling of MGPs are performed after the modularized modeling procedure. Parametric modeling of DCAs, an extended modeling part in the development of MGPs, is shown in Figure 7.

5. INTERACTIVE FEM APPROACH

5.1 Deficiencies of Current FEM Interfaces

Today, most finite element analysis systems let users control the finite element models through pre-processor, solution-solver, and post-processor. This means that changes to a particular structure are first made to the geometric model, then the revised FE model, the solution, and finally, the results review.

Complexity of geometric modeling and FEM development created a wide range of engineering software for the engineering industry. This led to not so many skills of using different software packages for solving design problems of a single product. For example, an electric engineer has to be trained in order to perform a thermomechanical FE analysis [5,17,49]. Currently, a pre-processor has a limited interactive-control capability for users to develop a FE model. The load and BCs have to be explicitly specified by the users to obtain a solution. The numerical outputs from FEA programs are often long, multi-valued, and not easily read or understood by inexperienced users. It is therefore important that an interactive user interface is available that will let a user perform a FEA task in a simple manner. This may include:

- Automatically link geometric model and appropriate FE mesh
- Allowing an user to graphically described load and BCs, and make a simple change in geometry
- Allowing an user to visualize the result as the model is modified

In short, since most FEA applications are based on physical models, geometric visualization of the output can greatly assist the understanding of the mechanics. The goal in this approach is to analyze and visualize the FEA output of a pre-developed model interactively. This model is built to represent a class of similar physical products, such as the models developed in previous sections. By this approach, it allows an user to study the output results of model modifications, such as changing loads, deforming shapes, and element sensitivity.

5.2 Steps for Interactive FE Model Computing

Steps for interactive FEM approach may vary in different applications. The key concept is allowing an user to interactively modify the model, and at same time, visualize the resulting changes. While a more comprehensive procedure is still under development, a few important steps are found as:

1. Create a graphics user interface for analysis visualization
2. Generate a modularized and parametric FE model
3. Pre-define different analysis type
4. Divide a large FE model into a few sub-domains for easy and fast computing
5. Specify solution accuracy criteria
6. Use mesh parameters and/or adaptive mesh for automatic element modification [14,23]
7. Compute and display solution results

5.3 An Simple Example: Steered 2D Plane Stress Analysis and Visualization

In this section, a simple example, Steered 2D Plane Stress Analysis, is constructed to show the basic concept of the interactive FEM approach. To be consistent with a traditional FEM approach, this example is explained in three aspects: pre-processing, solution computing, and post-processing. However, the difference is that all three steps are performed concurrently.

The first part is termed pre-processing, where the input data is created and organized. The class of 2D elements used is the isoparametric quadrilateral, a four-sided convex polygon of arbitrary shape. Using a mouse pointer in a graphical user interface, the user is able to drag the nodes of the 2D elements to interactively deform a 2D model. In addition, the user can add, modify, move, combine, and delete nodal loads by dragging graphical arrow icons (Figure 8).

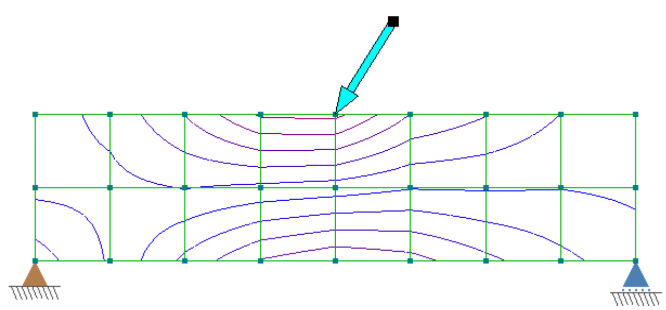


Figure 8. Contour display as a graphical arrow load-icon is applied

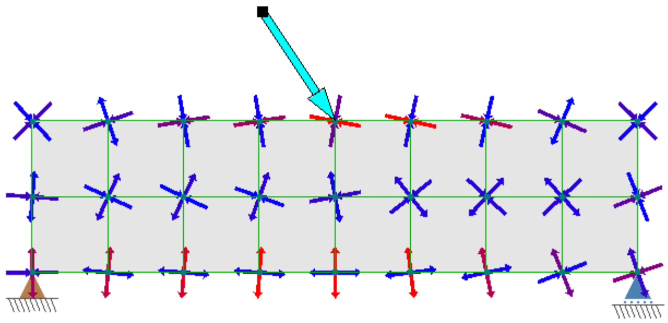


Figure 9 The directions of principle stress are dynamically changed with the change of loading arrow

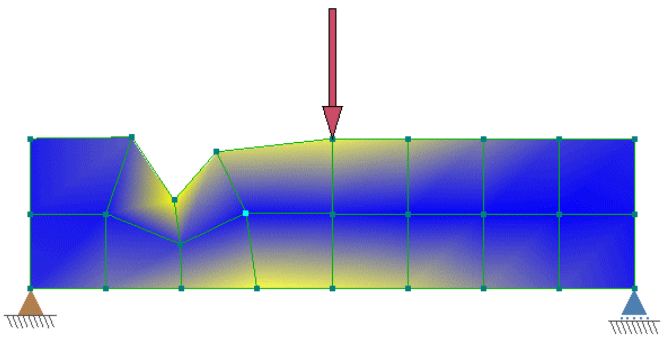


Figure 10. Model reshape by graphically moving a node

The second part is the actual computation of the finite element method. The direct-stiffness method is chosen for implementing this computation.

The third part is termed post-processing, and it is in this part that the output data is visualized graphically. The data to be visualized is classified as a 2D field of 3-vector floating point values. The 2D field represents the nodes of the quadrilateral elements, and the 3-vector values are the three plane-stress values: normal stresses σ_{xx}, σ_{yy}, and shear stress τ_{xy}. Analyses are made to these 3-vector values by computing the principle stresses σ_1 and σ_2 with angular directions, as well as the corresponding maximum shear stress τ_{max}. This is done using the Mohr circle equations, the 2D version of solving for eigen-values and eigen-vectors of general stress tensor. To visualize these data, three methods are used, with a color map legend illustrating the stress levels in MPa.

One important aspect of this example is the real-time steering of the visualization. At every change, including the changes of BC, nodes and loads, all the stress values of each node are re-computed again. The pre-processing routines, finite element analysis routines, and post-processing routines are all in the same program, allowing immediate update of the visualization based on user input. For example, the user will be able to stretch and/or rotate the loading arrow dynamically and watch the stress contours "flow" or the principle stresses' arrows rotate. This is illustrated in Figure 9.

By moving the nodes of the elements, the user may also reshape the model, as shown in Figure 10. This allows studying of stress concentrations in the model due to area of high-curvature, as well as the sensitivity of the element due shape distortion. The lighter shading around the V-cut illustrates high stress value.

The result of the this simple example is self-evident. The ability to dynamically modify the values to be visualized, coupled with steering of the visualization greatly enhances one's understanding of the analyzed problem. In addition, one can easily see how distorting of the elements, showed in Figure 10, may introduce artifacts in the FEA and probe the sensitivity of the elements. Furthermore, by watching the flow of the color contours during the dragging of the nodes, one might find an optimal nodal layout. One can then perform a more detailed analysis using another FEA system, using a decomposition of this optimal nodal layout.

6. MULTI-LEVEL SOLUTION APPROACH

Multi-level analysis provides the capability for different levels of analysis. Two different types of analysis are applied in global and local analyses: structural analysis and reliability prediction. Structural analysis evaluates the characteristics of package architecture in terms of stress/temperature distribution, and hot spots on PWBs or substrates, for geometrically coarse representations of the system. Structural analysis is to provide first-order insight into the behavior of the system, to determine the parametric sensitivities and inflection points, and to confirm that the final design can perform within the imposed specifications. The information obtained at this level will be the input for calculating the stress concentration and distribution, and for reliability prediction of a selected part of a package. The reliability analysis calculates the time to failure for dominant failure mechanisms in a assembly. For different failure mechanisms, formula based tools are applied at this level of analysis.

For an electronic product, there is very complex geometry and thousands of tiny parts. Producing detailed results for the entire product is neither practical nor necessary. During the first step of the global/local analysis procedure, the intent is to capture a global behavior when a whole product model is present. The global results may be warpage, stress or strain distribution, or "hot spot" on the product. Although the results may be approximated, they could indicate trouble spots for more detailed analysis. By focusing on a spot of interest, other parts are removed from its boundary, and previous results are applied as a boundary load. More detailed analysis is then performed on this part. Based on this detailed analysis, a formula based calculation can be performed to predict reliability data.

A multi-level analysis example, built by modularized and parametric FEM approach, is shown in the following. This example model has a test PWB with 13 components, includes 2 DCA, 2 TSOP with gull lead, 1 TSOP with J-lead, and 8 passive components.

Step 1. Model Generation Phase

The components are placed onto the board in sequence one at a time. First, a DCA is placed on the lower left corner of a board, as shown in Figure 11.

The DCA has a squared silicon die. As seen on the picture, the DCA is a meshed model. The geometry and mesh density are determined by its definition in a component library. Followed by this component, more components are continuously placed onto the board, as seen in Figure 12. They are one TSOP with gull leads on four sides, one TSOP with J-lead on four sides, and two small passive components with 3 solder joints on sides. Notice that one passive component is placed horizontally, while another is placed vertically. This can be easily achieved by specifying the angle of rotation as 90° with respect to another.

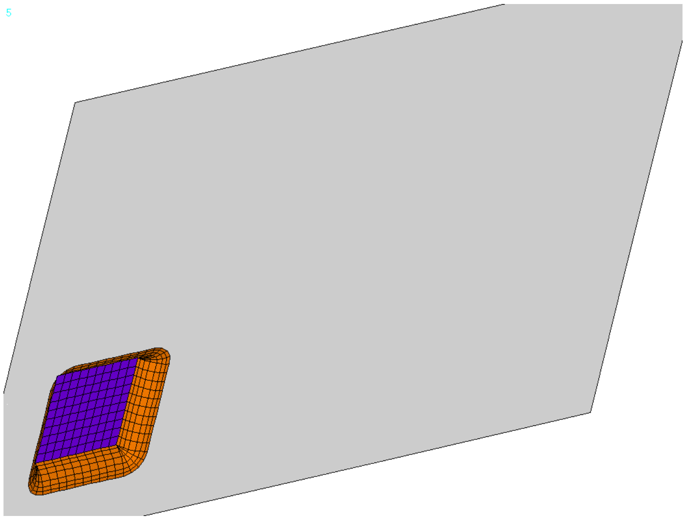


Figure 11. A DCA component is placed on a board

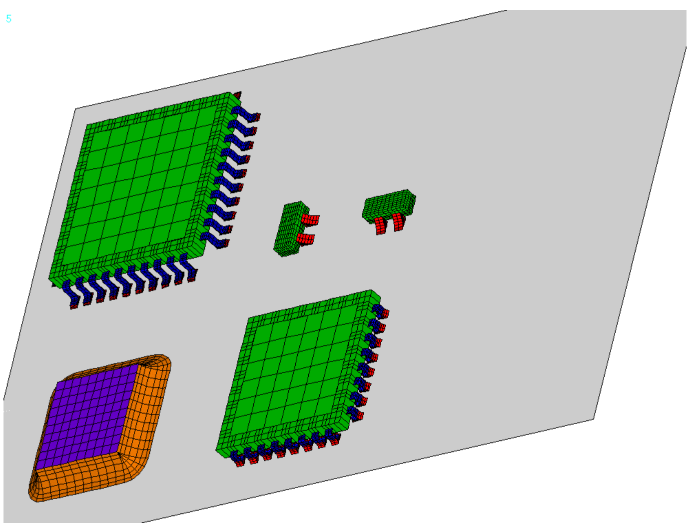


Figure 12. More component are continuously placed onto the board.

The entire model is completed by placing the rest of the components onto the board. Eight more components are mounted onto the board: 4 small passive components with solder joints on edge side; 2 large passive components with solder joints on edge side; one TSOP with gull lead with two side lead frame; and one DCA with a slim rectangular silicon die. The new DCA, on the top-right corner, is basically duplicated from the previous DCA on lower-left corner, but is of a different geometric size. As geometry sized changed, the meshing element number changed correspondingly, to keep the same mesh density. Similar to the duplication of a DCA, the newly placed TSOP with gull leads on two sides is created by changing the value of the module parameter, LEADSIDE, from 4 to 2. Figure 13 shows a board-level model with all the components.

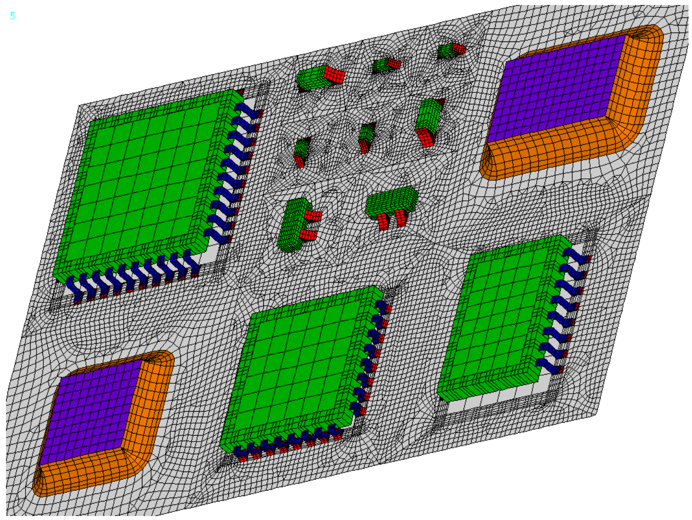


Figure 13. Completed model assembly.

Step 2. Global Analysis

The thermal analysis is applied as a test solution. The thermal analysis is to simulate the power switching process. The process assumes that all the components have a room temperature at the initial state. When the power switch is turned on, the components start to heat up. Because of the temperature loading, stresses are created in the components and solder joints. For simplicity, all active components are assumed to have the same power of 2W. The board is assumed to be nearly isolated from the environment, as if it is insulated in a case. Since it is a free convection and the heat removing capability is small, the heat dissipation is modeled as conduction. The room temperature is assumed to be 20°C. The solution is done at the state when the whole assembly has reached a steady state temperature. The left side of the board is clamped, as if the board is slided into a board slot.

Due to the CTE mismatch, the board is warped at the free edge. The strain distribution of the entire board is shown in Figure 14.

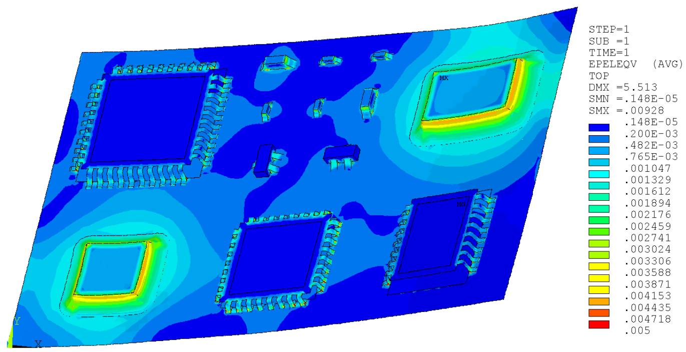


Figure 14. Strain distribution of the board under a thermal load.

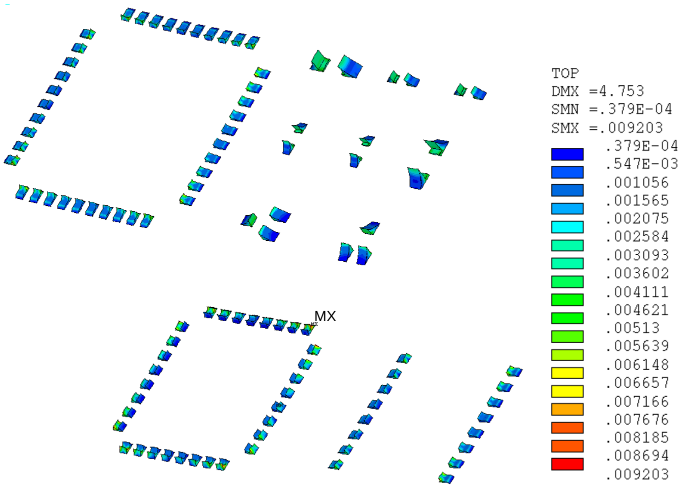


Figure 15. Strain plot for solder joints of active components.

In Figure 15, all the solder joints for the SMT components are plotted. The stress value of all the solder joints are sorted in the solution process. The joint, with highest stress, is marked by "MX" above the solder joint of a TSOP. The maximum equivalent strain value for this solder joint is found to be 0.92%. The strain may not represent the real value. However, Since all the solder joints are calculated under the same assumption, they should have similar percentage of error. Thus, by looking this plot, it immediately indicates, that under this specific thermal load, the J-lead joint, marked by "MX" should fail first.

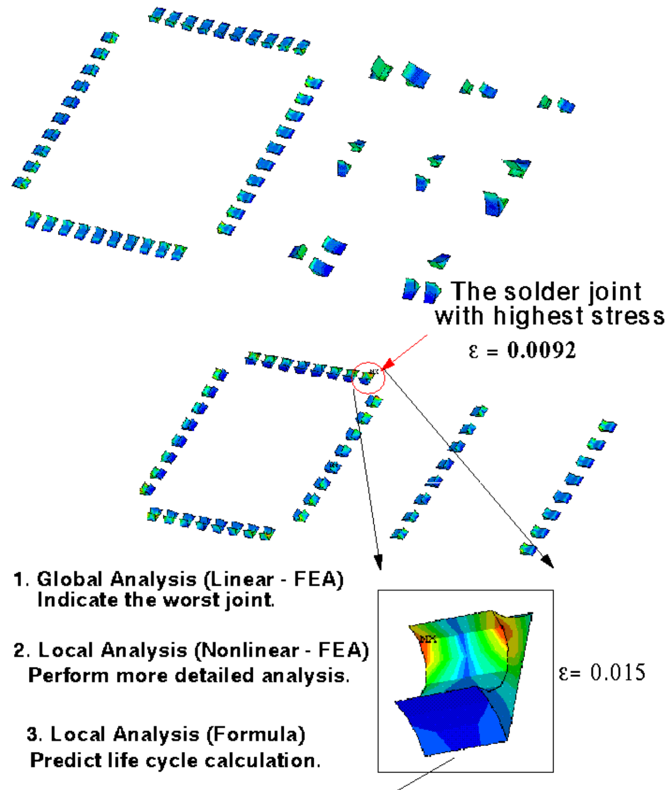
Once the trouble joint is found, more detailed analysis is needed to answer more questions, such as "how bad is this joint?". This type of analysis is then performed by a more detailed local analysis.

Step 3. Local Analysis

Based on the joint indicated in global analysis, a non-linear FE analysis is performed to examine its extended thermomechanical behavior. The non-linear results obtained are used to perform a formula based analysis, which is to predict the life-cycle of the joint.

As all J-lead solder joints are generated from a master model, that is, all the J-lead solder joints have the same geometry and mesh pattern, the only differences of the bad joint from others are the nodal values of the joint.

In order to perform more detailed analyses on the joint, only the stress and displacement values, associated with this joint, are needed. These stress values are picked by first selecting the nodes of the joint and then sorting the node numbers. These displacement and stress results are used as boundary and loading conditions to perform a non-linear FE analysis. The non-linear material property of the solder joint is modeled as bi-linear. It is assumed that the stress remains constant. Though the true stress on the solder joint may decrease when the non-linear material properties are applied, the change of stress is not considered in this local analysis. This is because the obtained non-linear results will represent the upper-bound condition, or the worst case. The non-linear strain obtained at this local joint is 1.5% (Figure 16), compared to 0.92% in the global analysis. The formula used for life-cycle prediction is the modified Coffin-Manson [1,6,15] relation for low cycle fatigue. The formula based results is also shown in Figure 16.



$$\overline{N_f} = \frac{1}{2}\left(\frac{\Delta\varepsilon^P}{2\varepsilon_f}\right)^{1/c} = 1407 \quad \text{(cycle)}$$

Figure 16. Global and local analysis for solder joints.

7. CONCLUSION

To reduce the modeling repetition, concepts of taxonomy and thermomechanical component library have been introduced. A thermomechanical component can be generated as a shared, manageable FE model. It can be handled as an icon (modularized FE model) for computational simulation of the movement/replacement of an electronic component.

Since a geometric sensitivity and uncertainty study is one of the key parts in thermomechanical design and optimization, flexible modeling capability is greatly needed. Pure traditional geometry parameter modeling and/or FEM no longer satisfy these needs. To overcome the deficiency, the parametric finite element modeling presented in this research includes parameters both for geometry creation and for mesh control in the modeling process. In addition, relations, such as "with/without" and "if-then" relations, are extensively treated as parameters in creating a model. A single model can be used for different tasks, materials, configurations, and conditions. Significant time savings have been found as the result of this one-to-many modeling.

Utilization of the concepts of modularization and feature based parameters provide for coupling of thermomechanical design with electronic design. When a new electronic component is added to existing design, a corresponding mechanical module can be assembled to the model, and be ready for a solution. "What-if" scenarios can be performed with design changes. Failure modes can be recognized and eliminated before final product design, and thus generate a "failure-free" product, which is "correct by design". This single pass design will result in time and cost savings.

Compared to traditional FEM approach, interactive FEM integrates pre-processor, solution, and post-processor into a single user interface. It allow an user (designer) to graphically modify the loading and BC conditions, as well as make minor changes to the geometry and elements. By adding interactive FEM capability into a traditional FE modeling package, a developed FE model can be used as both as an analysis tool and as a design tool.

The mechanical complexity of advanced electronic systems and the throughput limitations of existing computing platforms preclude the use of numerical models for detailed design. This research has developed a modeling methodology applying finite element solution within a few integrated controlling algorithms. Though this research project is still in its preliminary phases, the advantages of this modeling methodology have been significant. It is believed that a fully developed MPI/FEM method is a vital tool to create the bridge to a virtual design environment for electronic products. As a result, the innovative concepts addressed in this research should be of particular value to design engineers concerned with product reliability, performance, cost, and "time-to-market".

REFERENCES

Solomon, H. D., "Fatigue of 60/40 solder", IEEE Transaction on Components, Hybrids and Manufacturing Technology, Vol. CHMT-9, pp.423 - 432, Dec. 1988.

Bar-Cohen, A, "Physical Design of Electronic Systems - Methodology Trends, and Future Challenge", Advances in Thermal Modeling of Electronic Components and Systems. Pub. name, time.

Ling, S. X., Dasgupta, A., "A Design Model for Through-hole Components Based on Lead-Fatigue Considerations". Advances in Electronic Packaging, ASME, Vol. 4-1, pp. 217-225, 1993.

Ikegami, K., "Some Topics of Mechanical Problems in Electronic Packaging", Advances in Electronic Packaging, ASME, pp. 567-573, 1993.

Ladd, S. K., and Mandry, J. E., "Solving Real World MCM Design Problems", MCM 94 proceeding, pp.567- 571.

Keegan, K. K., Kearney, D., and Monghan, P. F., "Thermal Fatigue Life Prediction for a Middle Gull-Wing Joint Using Finite Element Analysis", Advances in Electronic Packaging, ASME, Vol. 4-1, pp. 499 - 507, 1993.

Godfrey, W. M., et al, "Interactive thermal Modeling of Electric Circuit Boards", ASME J. of Electronic Packaging, pp65 - 71, 1994.

Bhatti., P.K., Gschwend, K., Kwang, A. Y., and Syed, A. R., "Computer simulation of leaded surface mount devices and solder joint in a Thermal Cycling Environment", Advances in Electronic Packaging, ASME, Vol. 4-2, pp. 1047 - 1054, 1993.

McBridge, R. R., Zarnow, D. F., and Brown, R. L., "The essential role of custom Design Kits for Cost-effective Access to the MCM Foundry", MCM proceeding, 1994, pp.535 - 539.

Heinrich. S. M. and Liedtke, P. E., etal, "Effect of Chip Pad Geometry on Solder Joint Formation in SMT",Advances in Electronic Packaging, ASME, pp. 603 - 609, 1993.

Shah, M.K., 1990, "Analysis of Parameters Influencing Stress in the Solder Joints of Leadless Chip Capacitors," ASME J. Electronic Packaging, Vol. 112, No. 2, pp.147 - 153.

Lau, J. H., "Chip on board technology for multichip modules",Van Nostrand Reinhold,

Demaria, D., "CAD tool and Design for Manufacturing. Part I" Printed circuit Design. pp10 -14. May 1991.

Shephard, M. S., "Approaches to the Automatic Generation and Control of Finite Element Meshes." Applied Mechanics, ASME, vol 41, No. 4, Apr. 1988.

Engelmaier, W., "Long-Term Surface Mount Solder Joint Reliability in Electronic Systems with Multiple Use Environments and a Multiplicity of Components." Advances in Electronic Packaging, ASME. vol. 4-1, pp479 -486, 1993.

Nolan, L., "Mechanical CAD/CAM/CAE Contribution to Qualify in Concurrent Engineering." Manufacturing Aspects in Electronic Packaging, ASME. vol. 2, pp1 - 6,1992.

Dally, J. W., and et al "Electronic Packaging Education for Mechanical Engineers: a Panel Response." Advances in Electronic Packaging, ASME. pp1099 - 1012, 1992.

Demmin, J. C., "Choosing an MCM Technology for Thermal Performance." MCM'94 Proceedings, pp 549 -554, 1994.

Tummala, R., and Rymaszewski, E. J., "Microelectronics Packaging Handbook." Van Nostrand Reinhold, 1989Cadigan, Mike, " Trends in Electronic Packaging," Circuits

Classon, Frank, "Surface Mount Technology for Concurrent Engineering and Manufacturing", McGraw-Hill, New York,1993

Keister, Geald, "Surface-Mounted Components," Electronic Components News, pp.25 -39, Feb. 1990.

Electronic Packaging Handbook: The Institute for Interconnection and Packaging Electronic Circuits(IPC).

Baehmann, P. L., Sham, T.-L., Song, L.-Y., and Shephard, M.S., "Thermal and Thermo-mechanical Analysis of Multichip Modules using Adaptive Finite Element Techniques." computer Aided Design in Electronic Packaging, ASME 1992. Vol 3, pp.57-63

Burkhart, Art, "Recent Developments in Flip Chip Technology", Surface Mount Technology, July 1991, pp.41-44.

"Needs and Chanlleges of Electronic Packaging for Next Centery." Georgia Tech Package Research Center. 1994