This item was submitted to Loughborough's Institutional Repository (<https://dspace.lboro.ac.uk/>) by the author and is made available under the following Creative Commons Licence conditions.

For the full text of this licence, please go to:
<http://creativecommons.org/licenses/by-nc-nd/2.5/>

A Comparison of Modelling Methods for Electronic Interconnect Structures

A.O.Ogunjimi, D.C.Whalley and D.J. Williams

Department of Manufacturing Engineering

Loughborough University of Technology

Loughborough, Leics. LE11 3TU, UK

Abstract

A comparison has been made between the use of 2-dimensional, axisymmetric and 3-dimensional geometry in the finite element analysis of a die attach assembly using conductive adhesives. The axisymmetric geometry was found to be a good approximation to the 3-dimensional geometry and gave considerable savings in computer resources and human effort. The true effect of the corners of the square die modelled was, however, not reflected in the stress level predicted by both of the more computationally efficient but simplified 2-dimensional and axisymmetric models.

Introduction

The finite element method is one of the most popular techniques for analysing the static, dynamic and thermal behaviour of physical systems, structures and components. Finite element modelling is frequently applied in the analysis of electronic interconnect and, at Loughborough, it is being used to analyse the effect of process parameters on the quality of electronic interconnect. The accuracy of this method of analysis, like all other computational methods, depends entirely on the input from the analyst. Such input parameters need to be specified to reflect as closely as possible the real system behaviour. The intricate nature and increasing sophistication of electronic interconnections are making such structures more difficult to model accurately. The use of finite element analysis in a research environment where the impact of a large number of parameters must be investigated necessitates the creation of multiple models and analyses which results in large computer memory and disk space requirements and is very labour intensive. While the accuracy of the results is vital, overall analysis efficiency must be traded off against the computational resource required and the speed of analysis iterations. There is therefore a need to develop approximate methods that can adequately represent the real system whilst minimising computer processing time and memory requirements and the overall cost of the analysis.

This paper presents a comparison of three modelling approaches for the analysis of thermally induced stress in a die attach system. The particular structure modelled is assem-

bled using conductive adhesives and the stresses of interest are those resulting from cooling the assembly from the adhesive curing temperature to ambient. The results of this comparison are however generally useful for structural and thermal analysis of geometrically similar structures. The three types of models considered were 3-dimensional (3D), axisymmetric and 2-dimensional (2D) models.

Finite Element Analysis and Modelling

Finite element analysis (FEA) divides the geometry to be modelled into discrete units called elements. The behaviour of the structure can then be analysed by solving the large number of simultaneous equations which describe the boundary conditions and the interactions between adjacent elements. The use of FEA in problem solving can be divided into four major steps [1]:

1. Definition of the physical problem
2. Creation of a finite element model
3. Analysing the model
4. Interpretation of results

Definition of the physical problem includes establishing the geometry of the structure, the identification of the loading on the structure, component or system, i.e. the boundary conditions, selection of the type of analysis required and providing the relevant material properties.

Creation of a good model is crucial to the success of FEA. The model, as stated previously, is created to represent the behaviour of the physical structure as accurately as possible. The use of symmetry can often be employed to simplify the model. The model should be more highly refined, i.e. have more, smaller, elements, in areas of interest such as near geometric discontinuities or where high stress intensity is anticipated. Accurate finite element models therefore require skills acquired through experience and good engineering judgement. There are, however, guidelines which if followed will reduce the probability of numerical difficulties and inaccurate results [1]. It should be borne in mind that FEA is an approximate method since the final model is usually an idealised representation of the real structure. The efficiency of analysis must also be taken into consideration while creating a model. Such factors as the degree of accuracy desired, computer time and space, and model generation time contribute to the efficiency of the analysis. Choos-

ing the right geometry and simplification method is crucial in saving analysis time while maintaining a reasonable level of accuracy. All real structures are 3-dimensional in nature, but reasonable approximations can often be made using 2-dimensional or axisymmetric models. This paper describes the use of such approximations of the die-attach system in which a silicon chip or "die" is mounted on a substrate using conductive adhesive.

The Die Attach Models.

Three models of the die-attach system were analysed using the SDRC I-DEAS finite element modelling system. The die-attach system modelled employed electrically conductive adhesive for bonding. The adhesive considered is a thermoset epoxy engineered for high temperature applications. The dimensions and material properties are as shown below.

Dimensions

Die (Silicon): 10mm x 10mm x 0.5mm

Adhesive (Epoxy): 50 μ m thick + fillet

Substrate (Alumina): 25.4mm x 25.4mm x 0.64mm

Material Properties

	E(GPa)	$\alpha(10^{-6}/\text{C})$	ν
Silicon	190	3.06	0.279
Epoxy	4.65	55.9**	0.35
Alumina	380	7.80	0.34

E - Young's Modulus of Elasticity
 α - Thermal Coefficient of Expansion
 ν - Poisson Ratio

** Average over the temperature range considered

A linear analysis of the thermally induced stress created by cooling the system from the curing temperature (275°C) to ambient (taken as 25°C) was performed.

The main assumptions used in the model are as follows:-

1. All material properties were considered to be linear (i.e. stress relief due to plastic deformation of the adhesive was not taken into consideration).
2. The adhesive layer is homogeneous and void free.
3. The system is in equilibrium at the curing temperature (i.e. strain free).
4. Isotropic material properties were assumed for silicon.

Figure 1 shows the plan view of the 3D geometry of the die attach assembly with the axisymmetric and 2-dimensional approximations highlighted using dashed lines.

The 3-Dimensional Model

The problem was modelled using 3D geometry taking planes of symmetry along the diagonal (z'y-plane) and mid-plane (xy-plane) of the assembly. This reduces the model size by a factor of 8. A total of 3038 10-node tetrahedral

Figure 1: Plan view of the Die-attach assembly

triangular solid elements were employed. Attention was paid to generating an effective distribution of elements to ensure good element density for a reasonable degree of accuracy.

The Axisymmetric Model.

A 2-dimensional axisymmetric model of the problem was also built. The origin of the axes was taken to be the centre of the structure, with the vertical (y) axis as the axial direction and the horizontal (x) axis as the radial direction. A total of 860, 6-node parabolic axisymmetric solid elements were used, ensuring adequate mesh density. As this model is 2-dimensional an approximation has to be made to allow the prediction of the stresses along the diagonal of the the real assembly. This was achieved by choosing the length of the model to be the same as the length along the diagonal of the real assembly.

The 2D Model

A 2D model of the same problem was built using 6-node plane stress parabolic triangular elements. The structure was modelled in the xy plane with a plane of symmetry at the centre of the structure. A total of 926 elements were used.

Figure 2: (a) Maximum Principal Stress and (b) Vertical Component of Displacement

There are two possibilities for modelling the two dimensional stress system. The out of plane thickness stress, σ_{zz} , is assumed to be zero or the through thickness strain, ϵ_{zz} , is assumed to be zero.

The first case is the plane stress situation. While this assumes the out of plane stress to be zero, the thickness of the model and hence the effect of the strain out of plane is not totally ignored in the calculation of the stiffness matrix of the model. Hence the variation in thickness of the various components of the model influences the stiffness. The latter case, the plain strain situation, is appropriate when the model is of a large and uniform thickness and hence the out of plane component of strain can be safely ignored. In view of the varying layer thicknesses in the structure being modelled the plane stress case was considered most appropriate.

Results

The output from the models are stresses and displacements. The results were therefore compared by examining the stress distributions and the displacements in the system. Figure 2 shows the maximum principal stress and the vertical component of displacement in the 3D model.

Displacements

The primary result of an FEA is normally the nodal displacements of the structure from which the stresses are then calculated [1]. Accurate prediction of displacements is usually readily achieved and reasonable displacements may be predicted even with a coarse mesh. Figure 3 shows the vertical component of displacements in the die. There is a clear correlation in the displacements predicted by the 2D model and the xy-plane of the 3D model on one hand and between the axisymmetric and the z'y-plane of the 3D model on the other. This therefore implies that, since the largest displacements occur along the diagonal of the die, the axisymmetric model is better than the 2D model at predicting the worst case displacement in the die.

Figure 3: Vertical Component of Displacement in the Die

Stresses

The stress distributions in the models are as shown in Figures 4 to 7. Figure 4 shows the maximum principal stress component of the stress distribution on the top of the die. As expected, the worst case once again occurred along the diagonals of the die as shown by the z'y-plane of the 3D model. There is a large difference in the stress predicted by the 2D models when compared with the axisymmetric and the 3D models. Where maximum stress is the criterion for evaluation of possible failure modes, the axisymmetric model predicts a closer value to that expected on the xy plane of the structure. The correlation in the position of the

maximum stress between the axisymmetric and the z'y plane of the 3D model is as a result of the length of the diagonals assumed in the axisymmetric model. The stress level in the axisymmetric model is however closer to that predicted along the mid-plane of the 3D model. The average stress predicted by the 2D model on the other hand is higher than the 3D model suggests.

Figure 4: Stress on the Top Surface of the Die

Figure 5: Stress on the Edge of the Die

A plot of the stress on the edge of the die as shown in figure 5 reveals that the axisymmetric models give a closer and a more similar distribution to the true case along the diagonal of the die while the 2D model, although having a similarly shaped stress distribution, predicts a stress level that is neither close to the value on the xy nor the z'y planes.

Figure 6: Stress on the Die-Adhesive Interface

Figure 7: Stress on the Adhesive-Substrate Interface

The stresses on the interface, figures 6 and 7 show a better correlation between the axisymmetric and the 3D models except for the stress level at the edges. The worst case, as expected, occurs along the diagonal of the die.

Discussion

There are four significant regions of interest in the die-attach assembly modelled.(Figure 8)

Figure 8: Possible effects of Thermal Stress due to Adhesive Curing

- The surface of the die; to predict whether stresses are high enough to cause cracking.
- The edge of the die; to investigate the possibility of cracking.
- The two interfaces of the adhesive; to check for the possibility of failure by peeling.
- In the bulk of the adhesive to check for the likelihood of plastic yielding of the resin.

Whilst a 3D model will give the most accurate results, considerable savings can be made by using the two approximate models. Figure 9 shows a comparison of the computer time taken for the three models on a SUN SPARC 1+ workstation. The system time is the time used in file handling and associated processes while the analysis time is the actual time taken to analyse the model. It is apparent that a substantial amount of computer resource is required for the analysis of the 3D model.

Furthermore, a considerable amount of engineering time (not included in the figure!) was spent in preparing the mesh for the 3D model, in contrast to both the axisymmetric and the 2D models which took very little time to build and mesh. In a research environment, in which sensitivity and similar studies are conducted on a regular basis, computer memory and disk space requirements and speed of analysis are limiting factors.

In all instances the axisymmetric model predicted closer values to the 3D model when compared to the 2D model.

Figure 9 : Comparison of Computer time taken for the Analysis

The major discrepancy between the 3D and axisymmetric models occurred in the stress level predicted on the surface of the die. This is due to the fact that the axisymmetric model does not take into account the stress raising effect of the corners of the rectangular die. The displacement plots of the 3D model suggest a largely axisymmetric pattern of displacement. The subsequent results also justify the use of axisymmetric geometry for modelling surface mount components by Hall [3]. It should, however, be borne in mind that the peak stress along the diagonal of the square could be as high as 1.85 times that predicted by the axisymmetric model (which is the same level of stress predicted for the mid-plane). Similarly the position of occurrence of the peak stress is geometry dependent and will therefore depend on the length of the model. Figure 10 shows a comparison of the structures that the different models actually represent.

Conclusions

In this work, the axisymmetric model has proved to be an acceptable approximation to the 3D model with the advantage that it reduces the total computer time taken by 98%. However, it should be noted that the stress on the surface of the die along the diagonal is greater than that on the mid-plane by a factor of about 1.85. Hence precautions should be taken when using the axisymmetric approximation to examine the level of stress on the surface of the die. It also follows from the statements above that a circular silicon die will result in a lower and more uniform stress level at the surface of the die when compared with a rectangular die.

A. 3-D Geometry

1. True representation of geometry
2. Very expensive to analyse
i.e. Large time, computer resources requirement

B. Axisymmetric Geometry

1. Axisymmetric representation of structure
2. Easy to construct and to mesh model
3. Ignores the effect of corners
4. Cheap to analyse

C. 2-D Geometry

1. Planar representation of structure
2. Easy to construct and to mesh model
3. Incomplete picture of structure
4. Cheap to analyse

[2] INSPEC, Properties of Silicon, Institute of Electrical Engineers, London, 1992.

[3] P.M Hall, "Forces, Moments, and Displacements During Thermal Chamber Cycling of Leadless Ceramic Chip Carriers Soldered to Printed Boards", Vol. CHMT-7, pp314-327, December 1984.

Figure 10 : Comparison of Models Geometry

This result encourages the use of axisymmetric models in sensitivity analysis in which geometric parameters such as the die size, bond-line thickness etc. for the assembly must be investigated. This approximation method can be applied to the analysis of structures having similar geometry in the manner of that already applied to the simplified analysis of surface mount joint deformation [3].

Acknowledgements

The authors acknowledge the support of a consortium of industrial collaborators including Lucas Aerospace, Lucas Advanced Engineering Centre and Rolls Royce and the financial support of the ACME Directorate of SERC under grant GR/H21241.

References

[1] National Agency for Finite Element Methods and Standards, A Finite Element Primer, Glasgow, 1986.