BGA Impact Analysis: Numerical Study

Rasha Mohamed Soliman, Medhat Awad El-Hadek, Shaban Ibrahim Abdou
Department of Mechanical Design & Production, Faculty of Engineering, Port Said University, Egypt.

Abstract
The detailed micro electronic and mechanical systems (MEMS) for a mobile microprocessor complex shape were analyzed using Finite Element (FE) processing.

Keywords: MEMS; Impact; Stress Analysis

1. Introduction
The literature survey of the past theoretical studies of predicting the mechanical and physical behavior of the components in varies assemblies indicated that the finite element method is a very essential tool used by engineers in the field of MEMS devices design

Through the literature, it was found to be a challenge to manufacture BGA packages for high performance applications with high thermal and electrical requirements in industry.

<u>The scope of the investigation</u>

1 - The present work comprises the use of both Ansys and Abaqus for the need of the benchmark.

2- Due to the lack of studies related to the deformations caused by the exposure to impact load, the submitted work emphasizes on getting an accurate and complete FEA of this type of deformations.

2. Benchmark analysis numerical model
2.1 The Theory
The model used was a thin solid circular plate of radius (a) with a clamped edge under a uniform load (P_o). Ventsel and Krauthammer [1] have concluded the closed form solution of the circular model under the following boundary conditions in their book.Where they stated that researchers could obtain the deflection from Equation (1).

$$w = \frac{Po}{64D}(a^2 - r^2)^2 \quad (1)$$

At the same time the maximum deflection that occurs at the center of the plate is determined by Equation (2).

$$w_{max} = \frac{p_{o\,a}{}^4}{64D} \quad (2)$$

Where D is the modulus of rigidity.

2.2 Benchmark FE Model
Two separate 3D models were conducted for the previously mentioned numerical circular plate and meshed using the same suitable element size. The exact same boundary conditions were applied in both cases, where the model was clamped from the edge. A uniform pressure was distributed over the surface of the first model, as shown in figure (1).

Figure (1) Uniformly distributed pressure

2.3 Results of the benchmark
The relation between the location and the deformation corresponding to the results of the benchmark FEA along with the theory results is presented in figure (2) for each case individually.

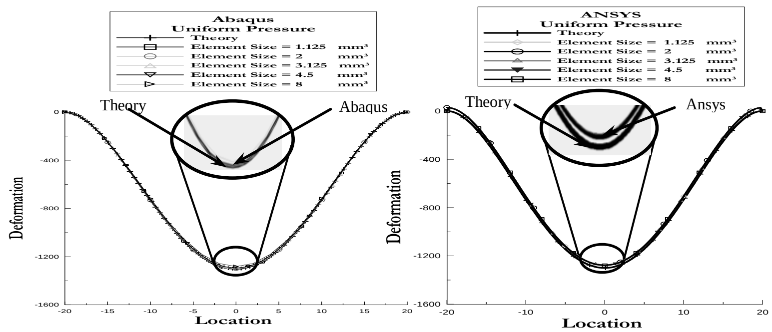

Figure (2) The relation between the location and the deformation in case of the uniform pressure loading condition using Abaqus & Ansys.

2.4 Benchmark FEA Conclusion
As presented in figure (3) it is clearly that the values gained from Abaqus were more accurate and closer to the theory results than Ansys. In case of uniform pressure, the error calculated between the fit equations of Abaqus and Ansys was of the value of 1.44 %.

Therefore, Abaqus was the software used in the upcoming part of BGA analysis under impact loading conditions.

3. BGA Model
This BGA has perimeter solder balls with 35 balls along each periphery of the package and five rows of balls along each side.The solder balls are made of 63Sn/37Pb solder.

The Von Mises stress analysis in addition to studying the strain energy criteria on a multi-layer 600-pin BGA with the package size of 45 mm square were carried out. The BGA model was constructed as presented in figure (3). The BGA model consists of eight layers using the following materials: polyimide/glass, copper, Tin-Lead solder, dry film solder mask, polyimide tape, and Finally, Mold compound, and epoxy adhesive. Rohde and Swearengen [2], Darveaux and Mawer [3] and ASM [4] reported that these materials are the most suitable for the BGA application

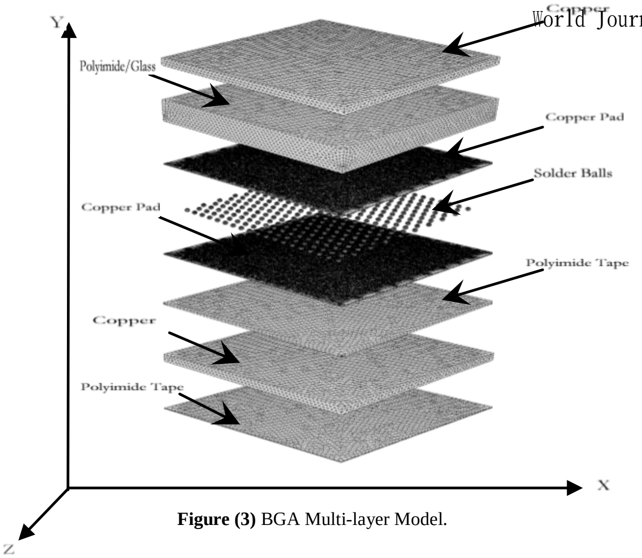

Figure (3) BGA Multi-layer Model.

The element type used the meshing process was C3D10M, Diehl and Carroll [5] offered a concentrated study on the benefits of using the C3D10M element modeling approach. The number of elements generated was 385945, which is a large number that will take long time in analysis and high configuration hardware. As a result, I took a cross section through the model, as shown in figure (4), exactly 1/16 of the symmetrical model to reduce the analysis time.

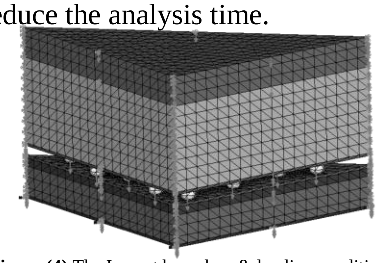

Figure (4) The Impact boundary & loading conditions.

4. Boundary Conditions

The model was clamped from the bottom and pinned it from the two cut sides, as presented in figures (4). In order to apply the impact loading condition, the applied loading conditions simulated a fallen BGA device from a constant distance.

5. Results

5.1 Impact Analysis

Figure (5) presents the maximum Von Mises stresses at the different velocities, based on a fixed falling distance of 1.5 meter.

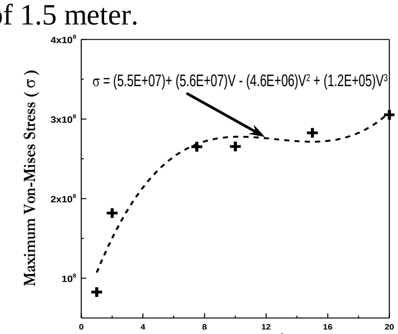

Figure (5) The relation between the impact velocity and the maximum Von Mises stresses at the moment of impact.

The Stain Energy for the whole element was also computed to be able to determine the increase in energy associated with the deformation of the member under the Impact loading condition. Figure (6) represents the relation between the strain energy and the velocity of impact.

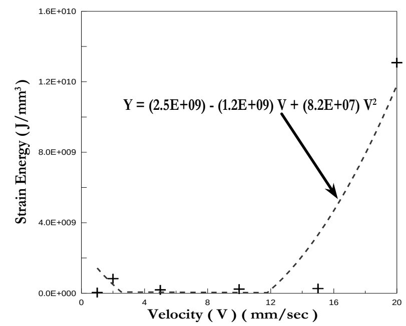

Figure (6) The relation between the impact velocity and the Strain energy.

6. Conclusion

Several important points throughout this research could be extracted as follows:

1- The benchmark non-linear analysis showed that the results gained from running the model using Abaqus were closer to the closed form solution of the circular plate than Ansys.

2- Impact boundary conditions were applied to simulate the most common loading conditions that affect the MEMS daily applications according to the in-depth literature survey.

3- Impact results showed a normal distribution of maximum Von Mises stress over the different impact velocities.

4- Strain energy manifests approximate close and small ranges over the impact velocities until the 36 km/hr, and then it starts to increase tremendously reaching to the maximum at the extreme velocity condition of 7g.

References

1- Ventsel, E., and Krauthammer, T. (2001).Thin Plates and Shells, Theory, Analysis and Applications, pp. 95-103.

2- Rohde, R.W., and Swearengen, J. C. (1980). Deformation Modeling Applied to Stress Relaxation of Four Solder Alloys, ASME, Vol. 102, pp. 207-214.

3- Darveaux, R., and Mawer, A. (August 1995). Thermal and Power Cycling Limits of Plastic Ball Grid Array Assemblies, SMI Proceedings, San Jose, CA, pp. 1-12.

4- ASM International (1979). Electronic Materials Handbook, Volume 1, Ohio.

5- Diehl, T., and Carroll, D. (2000). Utilizing ABAQUS' 10-Node tet for analysing Impact problems Involving Thin-Walled Structures, Abaqus Users' Conference Proceedings.