ABSTRACT

Die crack is a common problem in the semiconductor industry and being able to predict the breaking force at a given loading condition could help prevent such crack problem. This paper presents the use of mechanical simulation in predicting the force at which the silicon die breaks in semiconductor package assembly process. A computer simulation with finite element analysis (FEA) technique was used. The applied force or displacement in a die bending simulation with 3 mm, 4 mm and 15 mm support span was varied until the resulting maximum principal stress of the die becomes equal to its fracture strength. Results revealed that the breaking force for the 70 µm die with 6 mm width is around 5 N for the 3 mm support span and only around 1 N for the 15 mm support span. With the good agreement between modeling and actual results, the study showed that mechanical simulation is an effective approach in predicting die breaking force and can be used to simulate different mechanical loads in the package assembly where possible die crack could happen and be avoided. This is a fast and cost-effective way of assessing risk of die crack and obtaining package assembly process parameters and specifications that are safe to the silicon die.
Keywords: Mechanical simulation; finite element analysis; die strength; die fracture strength; die crack; 3-point bend test; silicon die.

1. INTRODUCTION

Package miniaturization is the continuing trend in the semiconductor industry. As package gets smaller and thinner, the silicon die also becomes very thin. Silicon material is brittle and prone to fracture. There are many challenges with processing very thin dies in wafers for semiconductor packages. In the processing and handling of silicon wafers thinned to 50 μm and below, an external means of mechanical support is always required [1]. Mechanical strength is required to hold the thin wafer rigidly during processing [2]. During the wafer manufacturing cycle, the wafers are exposed to mechanical loads caused by sawing, manual handling, liquid jets, transport systems and pick and place equipment [3]. Die mounting (die attach) can cause die crack as reported in [4]. Thinner die has higher potential to have severe crack as compared to the larger one at die attach process [5]. There are many other processes that could induce die crack. When die is exposed to higher mechanical loads, higher die stresses are also generated. Then die crack happens when the stress in the silicon die reaches its fracture strength.

The prediction of die breaking force at a given loading condition needs the characterization of die fracture strength. Die strength is commonly measured using a 3-point bend test [6-10]. The 3-point bend test is a die strength testing method widely used in the semiconductor industry and there is already an international standard [8] for it. The setup of the 3-point bend test is also much easier. Die stress calculations can be done using finite element analysis (FEA). In a related study [11], the FEA-predicted die stress is used to predict the die failure rate compared with the experiment results. FEA is also used in evaluating the feasibility of the linear beam theory when considering geometric nonlinearity in die bending [12], analyzing die stress with ball-breaker test [13] and even for modeling monotonic bend test of flip-chip BGA package [14].

In this study, mechanical simulation with FEA was used to predict the die breaking force based on the die strength measured from the 3-point bend test. The purpose was to establish a fast and cost-effective way of obtaining the appropriate package assembly process parameters and specifications that are safe to the silicon die. Knowing the die breaking force at a given loading condition could help eliminate die crack during the semiconductor assembly as the force specifications would be set much lower than the breaking force.

2. DIE STRENGTH CHARACTERIZATION AND MECHANICAL SIMULATION

The die fracture strength characterization was done using an Instron MicroTester with a 3-point bend fixture compliant to the international standard SEMI G86-0303 for measurement of die strength [8]. The 3-point bend setup and testing procedure were based on that SEMI standard. On the other hand, the mechanical simulation was performed using ANSYS Workbench, a finite element analysis (FEA) software. Different die bending scenarios were simulated and the predicted breaking force was compared with the result from actual die bending experiment.

2.1 Die Strength Characterization

The Instron MicroTester equipment used for the die strength characterization is shown in Fig. 1. It has a load cell that measures the amount of force applied to the specimen in a 3-point bend setup as illustrated in Fig. 3. The silicon die is supported at the bottom by 2 stationary anvils and force is applied from the top with the movable upper anvil.

The Instron MicroTester measures the maximum load before the die breaks (Fig. 2) and the die strength is then calculated using the following equation [8]:

\[ \sigma = \frac{3FL}{2bh^2} \]  

(1) where,

- \( \sigma \) = die fracture strength
- \( F \) = die breaking force (maximum load before breaking)
- \( L \) = span or distance between supports
- \( b \) = die width (parallel to the support axes)
- \( h \) = die thickness

Silicon die used in the study has a thickness of 70 microns and a width of 6 mm as this is the
specification of the die considered in the mechanical simulation and actual bending experiment. The distance between anvil support, $L$, was set at 3 mm. The test speed used was 3 mm/min to avoid the impact on the test specimen. A slower speed was selected to eliminate dynamic effect. The die was placed and centered on the two stationary anvils and force was applied at midspan by the loading upper anvil. There were 15 silicon die samples used for the die strength testing. The maximum force was recorded, and the fracture strength or die strength calculated for each die according to equation (1). The average die strength obtained was then used in the FEA mechanical simulation to predict the breaking force at different die bending conditions. The calculated die strength would be the target die stress value to be achieved in the FEA simulation.

2.2 Mechanical Simulation and Actual Bending Experiment

Mechanical simulation was carried out with different die bending conditions. Table 1 summarizes the bending conditions included in the simulation. The different support span values were chosen arbitrarily with consideration of the actual bending setup capability. The mesh size of the die was maintained for all the die bending conditions analyzed to ensure consistent results. In this specific simulation, the body mesh size is maintained at 0.20 mm with 2 elements along the die thickness. The two bottom anvils or supports and the upper loading anvil were included in the FEA model as well as the specific anvil geometry. The FEA simulation was using linear elastic material properties. The silicon die has an elastic modulus of 169 GPa and a Poisson's ratio.
of 0.23. On the other hand, the anvils are made of stainless steel with a modulus of 193 GPa and a Possion’s ratio of 0.31.

In ANSYS Workbench software, contacts between the bottom anvils and the bottom surface of the die were defined. Additional contact pair was also defined to simulate the contact between the top surface of the die and the upper anvil. A downward displacement was applied to the upper loading anvil and the force was taken as the reaction force result at the anvil support fixed base opposite to the displacement direction. The boundary conditions and loads are shown in Fig. 3 for the quarter symmetry model and Fig. 4 shows the whole FEA model with the mesh shown.

The breaking force was taken as the reaction force at a certain displacement applied that induces die stress equal to the average die strength obtained from die strength characterization. The FEA mechanical simulation runs were done iteratively until a die stress value equal to the die strength was obtained. The predicted force values obtained by mechanical simulation were compared with the actual breaking force recorded from the die bending experiment. Instron MicroTester was also used in the experimental study for the different die bending conditions described in Table 1. The actual breaking force was determined directly from the actual testing output from the MicroTester.

3. RESULTS AND DISCUSSION

The representative load-deflection curves for the die strength characterization are shown in Fig. 5. Only the curves for the first 6 die samples are displayed. Each curve describes the relationship between the load applied (N) and the corresponding die deflection (mm) from the initial position. The load drop signifies the breaking of the die after reaching the maximum force. From the die strength testing, an average value of 823 MPa was calculated based on equation 1. This die strength is for the specific silicon die used in all the actual bending experimental conditions in this study. This stress value was used as the stress limit in the mechanical simulation.

### Table 1. Bending conditions

| Bending Condition/ Loading Setup | Anvil Radius |
|---------------------------------|-------------|
| Die bending with 3 mm span       | 0.3 mm      |
| Die bending with 4 mm span       | 0.3 mm      |
| Die bending with 15 mm span      | 2.5 mm      |

Fig. 3. FEA boundary conditions (BC) and loads
Mechanical simulation result for the die bending with 3 mm span is shown in Fig. 6. The predicted breaking force is 5.37 N. This indicates that this amount of force can induce a stress in the die equal to 823 MPa, the die strength of the silicon die. Since the die breaks when the induced stress reaches its fracture strength, the 5.37 N load is taken as the breaking load from FEA mechanical simulation. The maximum stress, as shown in the die stress contour plot, occurs at the bottom of the die under the location of the applied load. The high stress area is more concentrated and runs parallel to the loading anvil axis. This high stress location is where the die crack is expected to initiate. The die bending with 4 mm span also has similar die stress contour plot but with a breaking force of 3.98 N.

For die bending with 15 mm span, result shown in Fig. 7 indicates higher die deflection but lower breaking force. It would only take 1.06 N force to break the die, and this is much lower compared to the result for die bending with 3 mm span. As shown in the die stress contour plot, the high stress area for this die bending with 15 mm span is wider compared to die bending with 3 mm.

Actual die bending experiment resulted in the following breaking loads: 5.38 N for 3 mm span, 4.68 N for 4 mm span, and 0.82 N for 15 mm.
span. As expected, the die broke along the high stress area indicated in the mechanical simulation die stress contour plots. It can be seen in Fig. 8 that the predicted values using FEA mechanical simulation are close to the actual values obtained from die bending experiment using Instron MicroTester. This good agreement between mechanical simulation results and actual results implies that mechanical simulation can be used to predict silicon die breaking force for establishing process specifications that could ensure nonoccurrence of die crack problem during semiconductor package assembly.

![Fig. 6. FEA die stress result for die bending with 3 mm span](image)
Fig. 7. FEA die stress result for die bending with 15 mm span

Maximum die stress = 873.8 MPa at 1.06 N

Fig. 8. FEA-predicted breaking load vs actual breaking load
4. CONCLUSION

The mechanical simulation and actual experiment done in this study showed that the silicon die breaking force in bending is higher when the distance between supports is shorter. For the 70 µm silicon die with 6 mm width, the breaking force is around 5 N for the 3 mm support distance and only around 1N for the 15 mm support distance. Good agreement between the predicted breaking force and the actual breaking force has been achieved. Therefore, FEA mechanical simulation can be used to predict silicon die breaking force in different die bending conditions. It can be considered an effective approach in predicting the breaking force of die undergoing bending in different processes of the package assembly where possible die crack could happen. The predicted value can be used as basis for assessing the risk of die crack and obtaining the right package assembly process parameters and specifications that are safe to the silicon die. Further study on mechanical simulation involving a load from a pointed tip such as the ejector needle pushing the die in the die attach process could be done.

DISCLAIMER

The products used for this research are commonly and predominantly use products in our area of research and country. There is absolutely no conflict of interest between the authors and producers of the products because we do not intend to use these products as an avenue for any litigation but for the advancement of knowledge. Also, the research was not funded by the producing company rather it was funded by personal efforts of the authors.

ACKNOWLEDGEMENTS

The author would like to thank the management of STMicroelectronics for the support provided in this study especially to the New Product Development and Introduction (NPD-I) group.

COMPETING INTERESTS

Author has declared that no competing interests exist.

REFERENCES

1. Olson S, Hummler K, Sapp B. Challenges in thin wafer handling and processing.
2. Farrens S, Bisson P, Sood S, Hermanowski J. Thin Wafer Handling Challenges and Emerging Solutions. ECS Transactions; 2010.
3. Wang PA. Industrial Challenges for Thin Wafer Manufacturing. IEEE 4th World Conference on Photovoltaic Energy Conference; 2006.
4. Fan P, Wang J. IC Chip Crack Issues due to Mounting Process for Ultra-thin IC Smart Card Module. International Conference on Electronic Packaging Technology & High Density Packaging (ICEPT-HDP); 2008.
5. Annaniah L, Devarajan M. Investigation on Influence of Die Size Against Die Crack at Die Attach Process. Journal of Optoelectronics and Biomedical Materials. 2017;9:1.
6. Petefish WG, et al. High Performance Laminated Chip Package Technology. Semiconductor Packaging Symposium SEMICON West; 1998.
7. Heng LT, et al. 40μm Die Strength Characterization. International Conference on Electronic Materials and Packaging; 2008.
8. SEMI G86-0303: Test Method for Measurement of Chip (Die) Strength by Mean of 3-Point Bending; 2011.
9. Shih HC, Tsai F, Shih MK, Targ D, Hung CP. An Experimental Investigation Into Thin Silicon Die Strength Evaluation. 14th International Microsystems, Packaging, Assembly and Circuits Technology Conference; 2019.
10. Finn DS, Lin Z, Kleinernt J, et al. Study of die break strength and heat-affected zone for laser processing of thin silicon wafers. Journal of Laser Applications; 2015.
11. Hu G, Luan J, Baraton X. Characterization of Silicon Die Strength with Application to Die Crack Analysis. 33rd International Electronics Manufacturing Technology Conference; 2008.
12. Tsai MY, Huang PS, Yeh JH, et al. Evaluation of Three-Point Bending Strength of Thin Silicon Die with a Consideration of Geometric Nonlinearity. IEEE Transactions on Device and Materials Reliability. 2019;19:4.
13. Chen PC, Su YF, Yang SY, et al. Evaluation of Die Strength by Using Finite Element Method with Experiment Validation. IEEE Transactions on
14. McCann S, Lee T, Ramalingam S. Finite Element Modeling Methodology for Monotonic Bend Test of Flip-Chip BGA Package. IEEE 70th Electronic Components and Technology Conference (ECTC); 2020.

© 2021 Talledo; This is an Open Access article distributed under the terms of the Creative Commons Attribution License (http://creativecommons.org/licenses/by/4.0), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

Peer-review history:
The peer review history for this paper can be accessed here:
http://www.sdiarticle4.com/review-history/67199