Journal of Applied Mechanical Engineering

Her and Lan, J Appl Mech Eng 2014, 3:3
DOI: 10.4172/2168-9873.1000e130

Editorial Open Access

Dynamic Analysis of an Aluminum Plate Subjected to Drop Test

Shiuh-Chuan Her* and Shien-Chin Lan

Department of Mechanical Engineering, Yuan Ze University, Chung.Li, Taiwan

Introduction

Aluminum panels are widely used in lightweight structures especially in aerospace and automobile industries because of their high strength, stiffness and low weight. However, they are susceptible to impacts caused by runway debris, hailstones, dropped tools, etc [1]. Although the induced damage may be barely visible, especially for low-velocity impacts, the strength and reliability of the structure can be affected. Hence, the behaviour of aluminum structures under impact has received increasing attention. Impact is a complex phenomenon that includes the interaction between projectile and structure, motion of the projectile and the dynamics of the structure [2]. Complete modeling of an impact event can always be done using a three-dimensional finite element model, which involves the interface/contact between the impacting bodies. Gupta et al. [3] investigated the behavior of thin aluminum plates subjected to impact by blunt and hemispherical-nosed projectiles. Corran et al. [4] studied the effects of projectile mass, nose shape and hardness on penetration of steel and aluminum alloy plates of varying thickness. Camacho and Ortiz [5] carried out the impact analysis of aluminum plates by conical-nosed projectiles. Borvik et al. [6] investigated the impact behavior of steel plates with blunt nose projectiles. In this work, the commercial finite element software ANSYS/LS-DYNA is applied to compute the impact acceleration and dynamic strain on the aluminum plate during the drop impact. The finite element results are compared with the experimental measurements of acceleration and strain with good correlation between simulation and drop testing.

Experimental set up of drop test

The shock test machine (King Ton model DP-1200-60) as shown in Figure 1 is used to conduct the free-fall drop test. This machine can provide three different types of impact impulse including half sine pulse, trapezoid pulse and square pulse, which are dependent on the target material. The half sine pulse will be chosen in this study with four half-spheres of Teflon as target material as shown in Figure 1. The specimen of aluminum plate is mounted on the drop table through a steel fixture. The two shorter edges of the test specimen were clamped on the fixture. Between the plate and drop table, 20 mm standoff is added to allow aluminum plate bending. The drop table is dropped from a height of 50 mm along the four guiding rods onto four half-spheres of Teflon as shown in Figure 1. The dynamic responses of the aluminum plate including the acceleration and strain are measured during the drop test. Two accelerometers are placed on the top surface of the aluminum plate, one on the center of the plate and the other closed to the right hand clamped edge of the plate. Two strain gauges are also placed on the back side of the aluminum plate, one on the center of the plate and the other closed to the right hand clamped edge of the plate. A high speed and multi-channel of data acquisition system is set up to record the accelerations and in-plane strains of the aluminum plate.

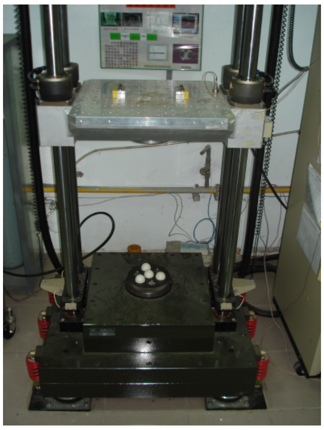

Figure 1: Experimental setup of drop test.

Finite Element Analysis

The commercial finite element software ANSYS/LS-DYNA which incorporates the pre and post processes of ANSYS with the solver of LS-DNYA is employed to investigate the impact response of the drop test. LS-DNNA classified the contact bodies into two categories named

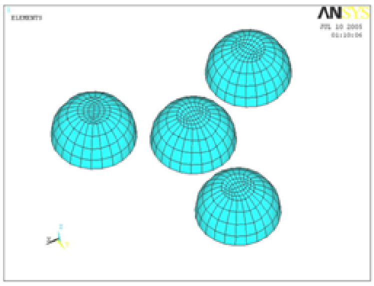

Figure 2: Finite element mesh of the target.

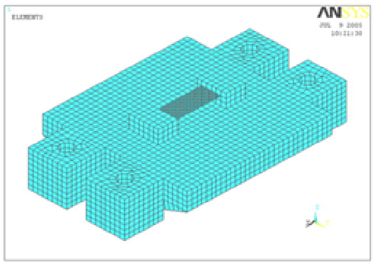

Figure 3: Finite element mesh of the drop table and test specimen.

***Corresponding author:** Shiuh-Chuan Her, Department of Mechanical Engineering, Yuan Ze University, Chung. Li, Taiwan, Tel: 03-46388002385; E-mail: mesch@saturn.yzu.edu.tw

Received April 02, 2014; **Accepted** May 23, 2014; **Published** June 06, 2014

Citation: Her SC, Lan SC (2014) Dynamic Analysis of an Aluminum Plate Subjected to Drop Test. J Appl Mech Eng 3: e130. doi:10.4172/2168-9873.1000e130

Citation: Her SC, Lan SC (2014) Dynamic Analysis of an Aluminum Plate Subjected to Drop Test. J Appl Mech Eng 3: e130. doi:10.4172/2168-9873.1000e130

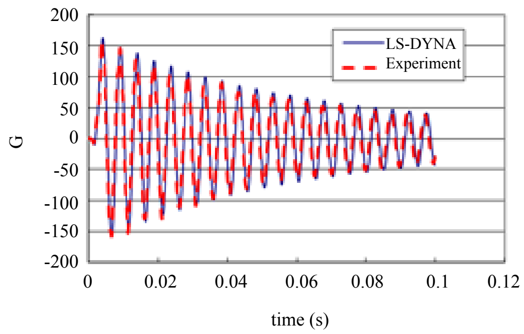

Figure 4: Acceleration at the center of specimen.

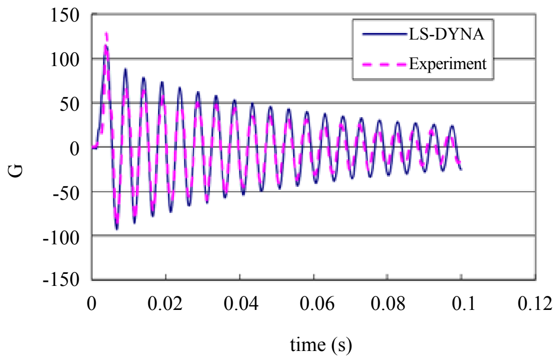

Figure 5: Acceleration at the side of specimen.

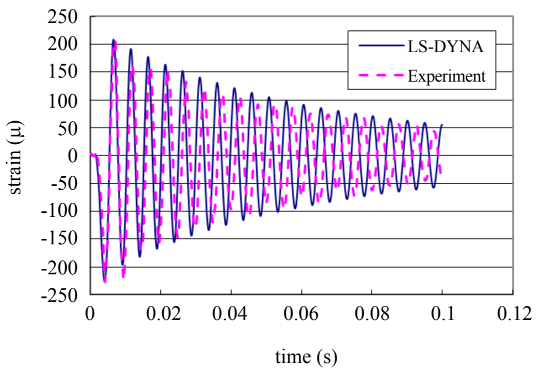

Figure 6: Dynamic strain at the center of specimen.

impactor and target. There are several contact methods provided by the LS-DYNA to simulate the impact. In this study, Surface to Surface (STS) contact is chosen. The finite element model of drop test consists of three parts including aluminum plate, drop table and the target (four half-spheres of Teflon). Since the thickness of the aluminum plate was quite small compared with the length and the width of the aluminum plate, it was modeled as shell elements (shell 163). The drop table and target used solid elements (solid 164). Figures 2 and 3 show the finite element meshes of the target, and drop table with test specimen, respectively. All the nodes of the ground in the four half-spheres are fixed. The nodes along the clamped edges of the aluminum plate were merged to the nodes on the fixture as fixed constraints.

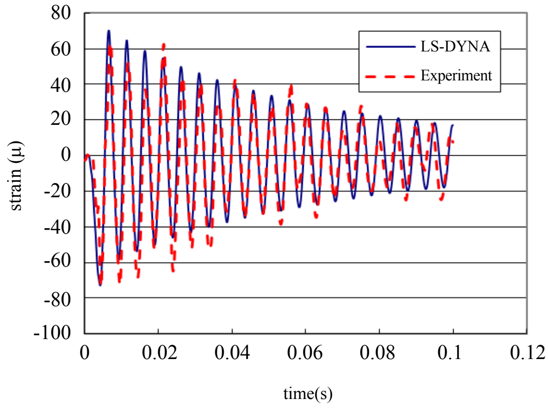

Figure 7: Dynamic strain at the side of specimen.

Results and Discussion

The accelerations at the center of the specimen calculated from the ANSYS/LS-DYNA and measured by the accelerometer are shown in Figure 4. Good correlations have been achieved between the finite element analysis and experimental measurement. The maximum acceleration obtained from the finite element simulation is 160.3G, while the experimental measurement is 159.6G. Figure 5 shows the acceleration of the point located 50 mm from the right hand clamped edge. It can be seen that the acceleration of the center point has a higher magnitude compared to that of the side of the specimen. Figures 6 and 7 shows the dynamic strain responses at the center and side points of the test specimen, respectively. It appears that the strain for the center point is larger than that of near the clamped edge. Good agreement between the numerical simulation and experimental measurement demonstrates that the dynamic responses of the impact can be evaluated via ANSYS/LS-DYNA.

References

1. Foo CC, Chai GB, Seah LK (2008) A model to predict low-velocity impact response and damage in sandwich composites, Composites Science and Technology 68: 1348-1356.
2. Pashah S, Massenzio M, Jacquelin E (2008) Prediction of structural response for low velocity impact. International Journal of Impact Engineering 35: 119-132.
3. Gupta N K, Iqbal M A, Sekhon G S (2006) Experimental and numerical studies on the behavior of thin aluminum plates subjected to impact by blunt- and hemispherical-nosed projectiles. International Journal of Impact Engineering 32: 1921-1944.
4. Corran RSJ, Shadbolt PJ, Ruiz C (1983) Impact loading of plates-an experimental investigation. Int J Impact Eng 1: 3-22.
5. Camacho GT, Ortiz M (1997) Adaptive Lagrangian modelling of ballistic penetration of metallic targets. Comput Meth Appl Mech Eng 142: 269-301.
6. Borvik T, Hopperstad OS, Berstad T, Langseth M (2001) Numerical simulation of plugging failure in ballistic penetration. Int J Solids Struct 38: 6241-6264.