Springer

Journal of Mechanical Science and Technology 23 (2009) 964~967

Journal of
Mechanical
Science and
Technology

www.springerlink.com/content/1738-494x
DOI 10.1007/s12206-009-0322-7

Study of impact energy propagation phenomenon and modal characteristics of an armored vehicle undergoing high velocity impact †

Man Hoi Koo[1], Hong Seok Lim[2], Hak In Gimm[1] and Hong Hee Yoo[2,*]

[1]*Agency for Defense Development, Deajeon, 305-152, Korea*
[2]*School of Mechanical Engineering, Hanyang University, Seoul, 133-791, Korea*

(Manuscript Received December 24, 2008; Revised March 16, 2009; Accepted March 16, 2009)

Abstract

A new manufacturing technology is being employed to build a new type of armored vehicle. While thick panels are welded together in the old manufacturing technology, relatively thin panels are welded to a frame structure in the new manufacturing technology. The structural integrity of the new type of armor vehicles can be maintained mainly by the frame structures while the panel thickness is reduced significantly to reduce the total vehicle weight. Since the dynamic characteristics of a frame-panel hybrid structure are different from those of the old type of structure which consists of only thick panels, they should be identified to achieve a good performance of the vehicle. For this purpose, a proper FE model of the hybrid type of structure needs to be developed. In the present study, FE models are proposed to represent the frame-panel hybrid type structure efficiently. The impact energy propagation, the transient response and the modal characteristics are investigated with the FE models.

Keywords: Armored vehicle; Impact analysis; Impact energy propagation; Topology optimization

---

1. Introduction

The impact analysis of an armor panel is a long standing problem. In the past a few decades, a great amount of progress has been achieved both experimentally and analytically (see, for instance [1-3]). The experimental approach can provide good results, but it is too expensive and produces only limited amount of information. On the other hand, the numerical simulation approach, such as the one employing the finite element method, has proved to be an effective alternative especially when it is combined with modeling methods developed for the structural problems undergoing high velocity impact. Commercial codes have been evolving to the level in which complex deformation phenomena during high velocity impact can be analyzed accurately. 3-D nonlinear finite element (FE) codes with advanced contact algorithms and material models have been successfully employed to simulate several types of armors subjected to the impact of projectiles moving with various velocities (see, for instance [4-9]).

Fig. 1 shows an armored vehicle with the frame-panel structure. Such structures are lighter than the existing panel welded structures. The weight and performance control of the frame-panel structure is easier than that of the existing structure. With an appropriate FE model, the characteristics of the vehicle undergoing high velocity impact (with all ranges of striking velocities) should be analyzed with the transient responses of the armoured vehicles.

The purpose of the present study is to construct an FE model by which the impact energy propagation

† This paper was presented at the 4th Asian Conference on Multibody Dynamics(ACMD2008), Jeju, Korea, August 20-23, 2008.
*Corresponding author. Tel.: +82 2 2220 0446, Fax.: +82 2 2293 5070
E-mail address: hhyoo@hanyang.ac.kr

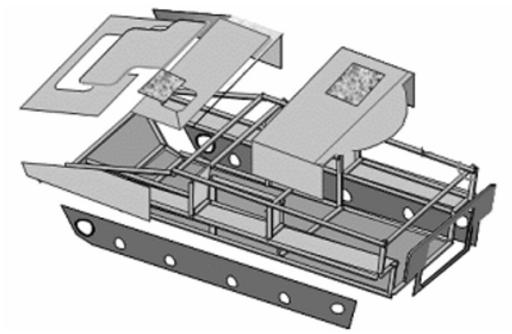


Fig. 1. Armored vehicle with frame-panel structure.

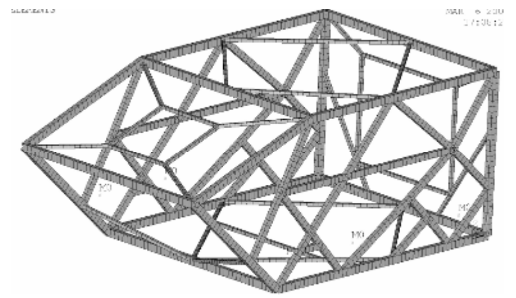


Fig. 2. Frame arrangement of the armor vehicle obtained from the topology optimization.

phenomena, the transient responses and the modal characteristics of the frame-panel structure can be obtained efficiently as well as accurately. Using LS-DYNA FE code, the transient responses and the modal characteristics of the vehicle model undergoing high velocity impact will be investigated and discussed.

2. Finite element modeling

2.1 Frame arrangement shape

The frame arrangement shape which has a structural stiffness sufficient enough to undergo specific loading conditions is obtained by using a topology optimization method. The topology optimization is executed by using a commercial FE code ANSYS. The given loading conditions consist of driving loading, dropping loading, inertial loading and twisting loading.

Fig. 2 shows the frame arrangement shape obtained from the topology optimization method. To verify the proper performance of the frame arrangement shape, structural analyses were carried out with the given loading conditions. The maximum displacement and stress are obtained and compared to the allowable

Table 1. Properties of the projectile.

Material	Diameter [mm]	Length [mm]	Mass [kg]	E [GPa]	S_Y [MPa]
AL1100	20	50	0.067	69	40

Table 2. Properties of the armored vehicle.

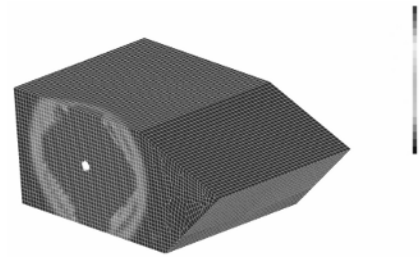
	Material	Thickness [mm]	Density [kg/m^3]	E [GPa]	S_Y [MPa]
Panel Welding Structure	AL7039	38.1	2850	71.7	330
Frame -panel Structure	AL7039	25.4	2850	71.7	330

displacement and stress. It was found that the obtained displacement and stress are less than the allowable displacement and stress.

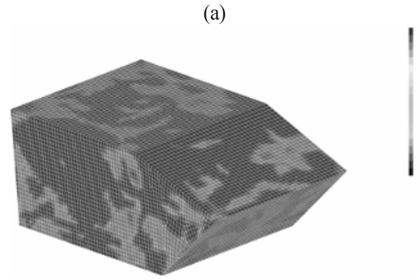
The section shape of each frame is assumed to have a hollow rectangular section. The dimension of each section is obtained by executing the section optimization method. Total frame weight obtained through the procedure is 334.24 [kg].

2.2 Modeling for the impact analysis

For the analysis of an armored vehicle undergoing high velocity impact, an FE modeling of the projectile and the armor vehicle is carried out. The projectile is modeled with solid elements, the panel is modeled with shell elements and the frame is modeled with beam elements. The properties of the projectile are given in Table 1. Properties of the armor vehicle are in Table 2 and Fig. 3 and Fig. 4 show the FE model of the armored vehicle. The external form of the FE model does not show the detailed composition of the structure which consists of two types of structures. The existing panel structures are made by welding the edges of the panels directly. On the other hand, the frame-panel structures are made by constructing the frame first and the panels are attached to the frame later. So such a structure is composed of beam elements along with shell elements. The difference of the two models mentioned above is that the panel thickness of the frame-panel structure is thinner than that of the welded panel structure. The total mass of the welded panel structure is 2208.1[kg] while that of the frame-panel structure is 1806.3[kg]. Therefore, the total mass of the frame-panel structure is

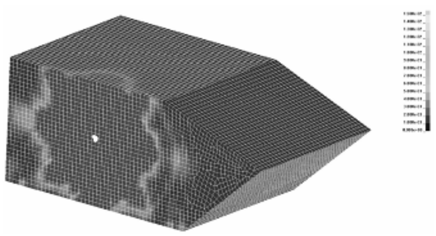


(a)

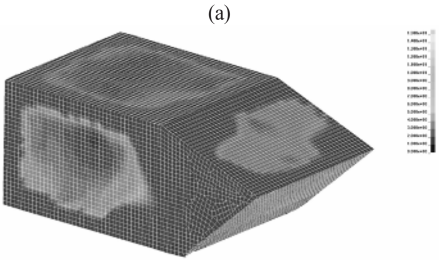


(b)

Fig. 3. Internal energy density of the welded panel structure versus time after the impact. (a) t=0.001[s] (b) t=0.05[s].



(a)



(b)

Fig. 4. Internal energy density of the frame-panel structure versus time after the impact. (a) t=0.001[s] (b) t=0.05[s].

decreased about 18% compared to the welded panel structure. For the boundary conditions of the armored vehicle, four corner points of the bottom of the vehicle are fixed in space.

3. Numerical results

The analysis of the developed 3-D FE model is carried out by using the plastic-kinematic material model. In the FE analysis, the surface-to-surface contact

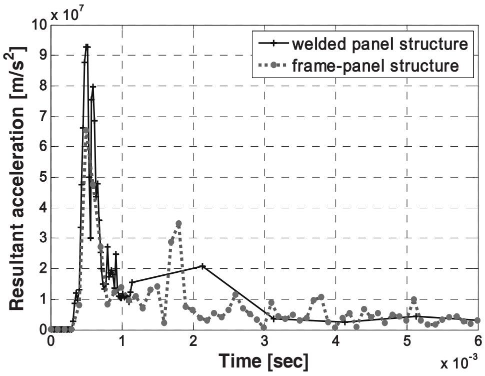


Fig. 5. Acceleration versus time after the impact for the two vehicle structures at panel center.

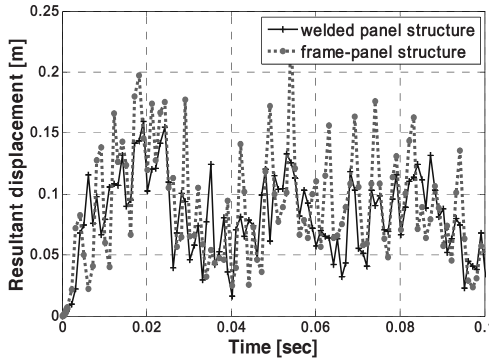


Fig. 6. Acceleration versus time after the impact for the two vehicle structures at panel corner.

algorithm is employed to simulate the contact behavior between the surfaces during the impact.

Fig. 3 shows the variation of internal energy density of the welded panel structure versus time after the impact and Fig. 4 shows the variation of internal energy density of the frame-panel structure after the impact. The impact velocity of the projectile is 400m/s and the projectile collides with the center of the side panel of the vehicle in normal direction. The energy propagation shapes obtained with the two structures remain similar until the vibration wave reaches the edge of the pane where the panel is adjoined to the frame for the frame-panel structure. As can be observed in Fig. 3 and Fig. 4, the energy is more concentrated around the frames for the frame-panel structure while it is almost uniformly spread out for the welded panel structure.

Fig. 5 shows the acceleration (at the projectile impact point) versus time for the two vehicle structures. As can be observed from the figure, the maximum value of the acceleration of the frame-panel structure is much smaller than that of the welded panel structure. However, the acceleration of the frame-panel structure remains relatively large compared to that of the welded panel structure after the impact occurs. Fig. 6 shows the displacement at the corner of the panel versus time for the two vehicle structures. As can be observed from the figure, the displacement remains relatively large in the frame-panel structure compared to the welded panel structure. Therefore one can say that the frame-panel structure has a disadvantage in vibration characteristics while it has an advantage in maximum acceleration characteristics.

4. Conclusions

Two FE models for the dynamic analysis of the frame-panel vehicle structure and the welded panel structure undergoing high velocity impact have been developed. For the frame-panel vehicle structure, especially, a topology optimization method was employed to obtain the frame configuration. Using the FE models structural dynamic analyses were performed with a commercial code (LS-DYNA). The numerical results show that the transient and modal characteristics of the two models are significantly different. The maximum acceleration of the frame-panel structure is much smaller than that of the welded panel structure. However, the magnitude of the vibration acceleration (or displacement) remains large in the frame-panel structure after the impact while it disappears relatively fast in the welded panel structure. Therefore, for the frame-panel vehicle structure, the residual vibration after the impact should be absorbed effectively by using damping materials and designing joint dampers in the frame structure.

Acknowledgment

This work was supported by the Defense Acquisition Program Administration and Agency for Defense Development under the contract UD070060CD and the Second Brain Korea 21 Project in 2008, Korea.

References

[1] W. Goldsmith, *Impact: the theory and physical behavior of colliding solids*, Dover publication, INC., Mineola, New York, USA, (1960).

[2] G. G. Corbett, S. R. Reid and W. Johnson, Impact loading of plates and shells by free-flying projectiles: a review, *International Journal Impact Engineering*, 18 (1996) 141-230.

[3] W. Goldsmith, Review: non-ideal projectile impact on targets. *International Journal Impact Engineering,* 22 (1999) 95-395.

[4] H. Mahfuz, Y. Zhu, A. Haque, A. Abutalib, U. Vaidya, S. Jeelani, B. Gama, J. Gillespie and B. Fink, Investigation of high-velocity impact on integral armor using finite element method, *International Journal Impact Engineering,* 24 (2000) 203-217.

[5] J. A. Zukas and D. R. Scheffler, Impact effects in multilayered plates, *International Journal Solids Structures,* 38 (2001) 3321-3328.

[6] Junyan Guo, Guangyu Shi, Yuyong Wang and Chun Lu, Efficient modeling of panel-like structures in perforation simulations, *Computers and Structures,* 81 (2003) 1-8.

[7] H. Kurtaran, M. Buyuk and A. Eskandarian, Ballistic impact simulation of GT model vehicle door using finite element method, *Theoretical and Applied Fracture Mechanics,* 40 (2003) 113-121.

[8] L. Kwasniewski, H. Li, R. Nimbalkar and J. Wekezer, Crashworthiness assessment of a paratransit bus, *International Journal Impact Engineering,* 32 (2006) 883-888.

[9] J. Li, X. J. Li, Z. Zhao, Y. X. Ou and D. A. Jiang, Simulation on projectile with high rotating speed penetrating into the moving vehicular door, *Theoretical and Applied Fracture Mechanics*, 47 (2007) 113-119.

[10] ANSYS, *User's Manual, Structural Analysis Guide*, ANSYS Inc, (2007).

[11] J. O. Hallquist, *LS-DYNA keyword user's manual, version 970*, Livermore Software Technology Corporation, Livermore, CA, USA, (2003).

Hong-Hee Yoo received a B.S. and M.S. degree in Mechanical Engineering from Seoul National University in 1980 and 1982. He then went on to receive his Ph.D. degree from Michigan State University in 1989. Dr. Yoo is currently a Professor at the School of Mechanical Engineering at Hanyang University in Seoul, Korea. His research interests are in the area of Flexible body dynamics, vibration.