

Technical Paper

Int'l J. of Aeronautical & Space Sci. 11(3), 221–226 (2010)
DOI:10.5139/IJASS.2010.11.3.221



A Numerical Study of the High-Velocity Impact Response of a Composite Laminate Using LS-DYNA

Jeoung-Hee Ahn*, Khanh-Hung Nguyen**, Yong-Bin Park**, Jin-Hwe Kweon*** and Jin-Ho Choi****

Research Center for Aircraft Parts Technology, School of Mechanical and Aerospace Engineering, Gyeongsang National University, Jinju 660-701, Korea

Abstract

The failure of a Kevlar29/Phenolic composite plate under high-velocity impact from an fragment simulation projectile was investigated using the nonlinear explicit finite element code, LS-DYNA. The composite laminate and the impactor were idealized by solid elements, and the interface between the laminas was modeled as a tiebreak type in LS-DYNA. The interaction between the impactor and laminate was simulated using a surface-to-surface eroding contact algorithm. When the stress level meets the given failure criteria, the layer in the element is eroded. Numerical results were verified through existing test results and showed good agreement.

Key words: High-velocity impact, Composite laminate, Kevlar, Delamination, LS-DYNA

1. Introduction

As the application of composite materials to aircraft structures has increased in various fields (Niu, 2005; Park et al. 2009), ballistic-tolerant design concepts have become a topic of interest for designers. Particularly in the design of fighters, it is necessary to consider the ballistic resistance of structures that are exposed to the potential danger of ballistic impact. In helicopters, survivability after a ballistic impact is required for the rotor blades, control, and the hub. The application of composite materials to armor already has a long history due to not only the ballistic resistance but also the excellent specific stiffness and strength offered by these materials.

The behavior of composite structures during a high-velocity impact is affected by a variety of factors, including the impact velocity, shape, mass and stiffness of the impactor, and laminate properties. The complicated failure mechanism of a composite laminate is an especially important obstacle in numerical simulations. Accordingly, numerical approaches regarding a high-velocity impact are

considered to be a challenging issue. For this reason, mainly tests have been used in this area (Jung et al., 2006; Lee et al., 2001; Liu, 2004).

There are two approaches regarding numerical methods, i.e., microscopic and macroscopic. In the microscopic model, the constituents of composite materials such as the reinforcing fiber and matrix are treated as separate materials (Ji and Kim, 2006). In another microscopic method, composite materials are idealized as a combination of unit cells consisting of the fiber and the matrix (Tabiei and Ivanov, 2002; Unosson and Buzaud, 2000). The macroscopic model assumes a composite laminate to be a structure made of a homogenous and anisotropic material. It is useful and efficient for simulating the structural behavior, as the constituents of the composite materials are not separately handled. However, it is somewhat limited in simulations of failure due to the combined effect of the fiber and the matrix (Ji and Kim, 2006).

While a number of papers (Kong et al., 2007; Lee et al., 2008; Park et al., 2008) are available related to low-velocity impacts on composite structures, a limited number of studies

© * Ph.D.
** Ph.D. Candidate

*** Professor, Corresponding author
E-mail: jhkweon@gnu.ac.kr Tel: +82-55-752-6391 Fax: +82-55-757-5622
**** professor

related to high-velocity impacts are available to the public. Even in the available literature, only limited information is provided, and data such as the stacking sequence and material properties are frequently excluded.

Tabiei et al. (2001, 2002) performed an analysis of a high-velocity impact on flexible Kevlar fabric using a homogenized microscopic unit cell approach. They then compared their results with test results. LS-DYNA provides the user defined material (UMAT) function to simulate a failure through impact on composite structures. Using this method, Hoof (1999) and Yen (2002) conducted numerical analyses of laminated composite structures.

The present paper has used a numerical analytic method for a Kevlar29/Phenolic fabric laminate under the high-velocity impact of a fragment simulation projectile (FSP). A nonlinear explicit finite element program, LS-DYNA version 971 (Livermore Software Technology Corporation, 2007), was applied using the element eroding contact algorithm. In the algorithm, where the stress level exceeds a certain criterion, the associated element is eroded from the finite element model. Delamination is an important failure mode in impact problems. The tiebreak contact interface was used to simulate delamination between composite layers. The finite element results were compared with the test results in an earlier study (Hoof, 1999).

2. Finite Element Analysis

2.1 Problem description

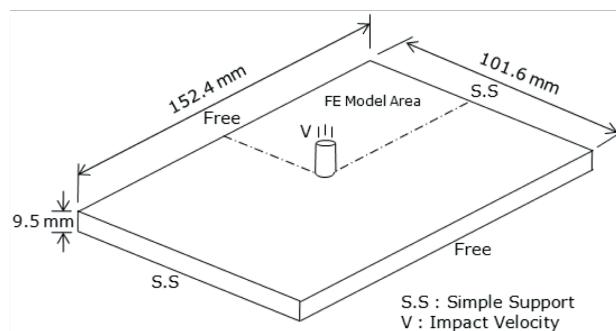


Fig. 1. Composite laminate under a ballistic impact.

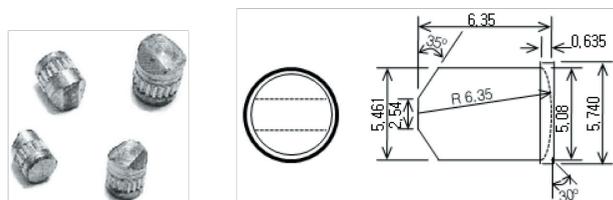


Fig. 2. Configuration of the fragment simulation projectile (mm).

Finite element analysis was conducted for the Kevlar29/Phenolic fabric laminated composite plate shown in Fig. 1. The plate has dimensions of 152.4 (length) × 101.6 (width) × 9.5 (thickness) (unit: mm). The long edges are free, and the short edges are simply supported. The impact velocities considered are 483, 545, and 586 m/s. The impactor FSP shown in Fig. 2 has a diameter of 5.59 mm and weight of 1.1 g. The FSP impacts upon the center of the plate. For finite element modeling, a symmetry condition is used. The material properties of the composite and the impactor are given in Tables 1 and 2, respectively (Hoof, 1999).

2.2 Finite element modeling

Key issues for finite element analysis of a composite laminate under a high-velocity impact include definitions of the material properties, development of a proper contact algorithm, and failure criteria. It is very difficult to define material properties that are valid for a wide range of impact velocities. Accordingly, it is important to find an adjustment factor for material properties to correlate the results of finite element analysis with test results. In the early design phase, many parameters for operating LS-DYNA were adjusted through iterative comparisons between the test and analytic results. To adjust the parameters, we use a trial and error method. First, we set the parameters as they are by default in the program and undertake analysis. We then change these parameters slightly when there is a mismatch between the results of numerical analysis and the test. After several iterations, suitable parameters can be found.

The finite element model is shown in Fig. 3, where a quarter of the plate is idealized using the symmetry condition. An

Table 1. Material properties of Kevlar29/Phenolic panels

Young's modulus	$E_{11} = E_{22} = 18.5 \text{ GPa}$ $E_{33} = 6.0 \text{ GPa}$: dynamic
Poisson's ratio	$\nu_{12} = 0.25$ $\nu_{13} = \nu_{23} = 0.33$
Shear modulus	$G_{12} = 0.77 \text{ GPa}$ $G_{13} = G_{23} = 5.43 \text{ GPa}$
Density	$\rho = 1.23 \text{ g/cm}^3$
Strength	$X = Y = 1,850 \text{ MPa}$ $Z = 1,200 \text{ MPa}$ $S_{12} = 77 \text{ MPa}$ $S_{23} = S_{13} = 543 \text{ MPa}$
Inter-laminar tensile strength	$S_n = 34.5 \text{ MPa}$
Inter-laminar shear strength	$S_s = 9.0 \text{ MPa}$

Table 2. Material properties of the impactor

Young's modulus	$E = 207 \text{ GPa}$
Poisson's ratio	$\nu = 0.25$
Yield strength	$Y = 1,034.2 \text{ MPa}$
Hardening modulus	$H = 685 \text{ MPa}$

eight-node hexagonal solid element was used to simulate the plate and the impactor. Each layer of the plate was modeled using two solid elements through the thickness, and the minimal element size was 0.25 mm × 0.83 mm at the center of the plate, which was the point of impact. The failure criterion for the composite material is a stress-based equation from Schweizerhof et al. (1998).

$$f = \frac{4[\sigma_1 - \frac{X_t - X_c}{2}]^2}{(X_t + X_c)^2} + \frac{4[\sigma_2 - \frac{Y_t - Y_c}{2}]^2}{(Y_t + Y_c)^2} + \frac{\sigma_{12}^2}{S_{12}^2} + \frac{\sigma_{13}^2}{S_{13}^2} + \frac{\sigma_{23}^2}{S_{23}^2} - 1 \quad (1)$$

In Eq. (1),

σ_1, σ_2 : Stresses in the fiber and transverse directions, respectively.

$\sigma_{12}, \sigma_{13}, \sigma_{23}$: Shear stresses.

Y_t, Y_c : Tension and compression strengths, respectively, in the fiber direction.

S_{12}, S_{13}, S_{23} : Shear strengths.

When the index, f , is equal to or greater than 0, the element is assumed to fail. It is interesting that in the equation, only five stress components, viz., $\sigma_1, \sigma_2, \sigma_{12}, \sigma_{13}$, and σ_{23} , affect the failure criterion, with the exception of σ_3 . In the analysis, an additional constraint is imposed on the strain for the case in which further calculation is not possible due to a large amount of deformation. Although an element does not fail in terms of the stress criterion, it is eroded when the strain exceeds a critical value. An eroding surface-to-surface contact algorithm was adopted for the simulation of contact between the composite plate and the impactor (Hoof, 1999; Livermore Software Technology Corporation, 2007). The tiebreak contact interfaces are defined between adjacent layers of the composite plate to determine the delamination, as shown in Fig. 4. When the combined value of the inter-

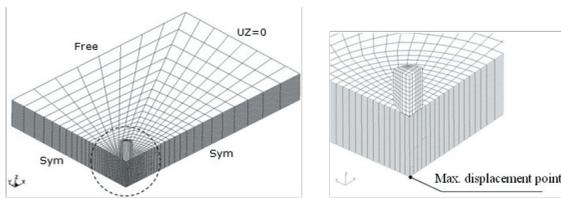


Fig. 3. Finite element model of a composite plate and an impactor.

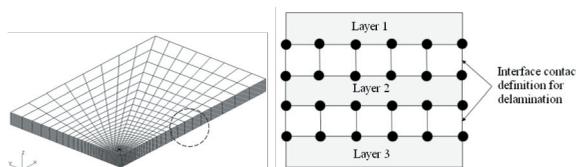


Fig. 4. Definition of the interfacial contact.

laminar normal and shear stresses exceeds a critical value, the tiebreak contact interface is deleted and delamination occurs (Hoof, 1999; Ji and Kim, 2006; Livermore Software Technology Corporation, 2007). The failure of the tiebreak contact interface is determined by a combination of the fracture modes, I, II, and III, as shown in Fig. 5. The failure criterion for the tiebreak is given in Eq. (2).

$$\left(\frac{|\sigma_n|^2}{S_n} \right) + \left(\frac{|\sigma_s|^2}{S_s} \right) \geq 1. \quad (2)$$

In Eq. (2),

S_n : Inter-laminar tension strength.

S_s : Inter-laminar shear strength.

The steel impactor was modeled with an elastic-plastic material. The von-Mises yield criterion for the impactor is defined as follows (Livermore Software Technology Corporation, 2007):

$$\phi = J_2 - \frac{\sigma_y^2}{3} \quad (3)$$

$$J_2 = \frac{1}{2} s_{ij} s_{ji} \quad (4)$$

with the following notation.

J_2 : Second order stress invariant

s_{ij} : Deviatoric stresses.

The yield stress, σ_y , is defined in terms of the effective

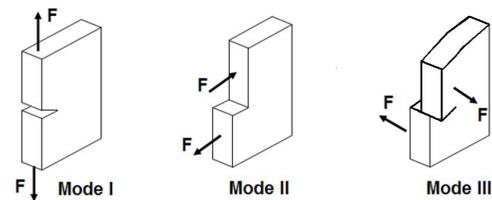


Fig. 5. Inter-laminar fracture modes.

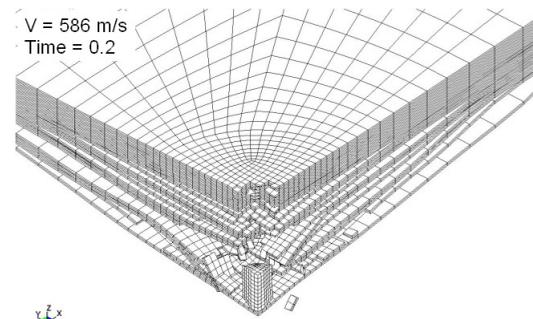


Fig. 6. Perforating behavior of a composite panel ($t = 200 \mu s$).

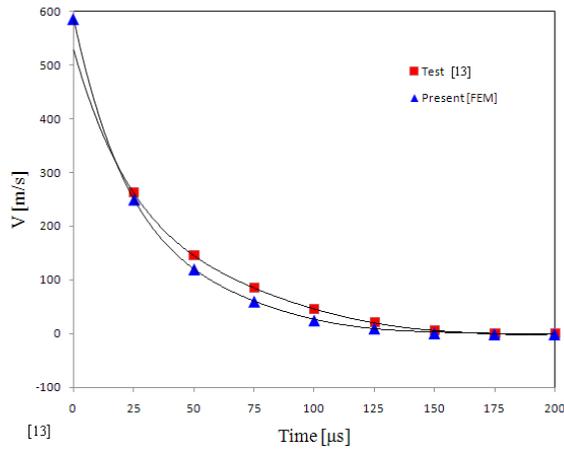


Fig. 7. Temporal variation of the fragment simulation projectile velocity ($V = 586$ m/s).

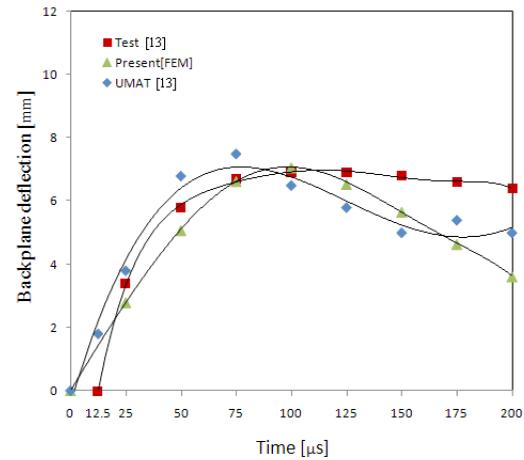


Fig. 9. Temporal variation of the backplane deflection ($V = 545$ m/s).

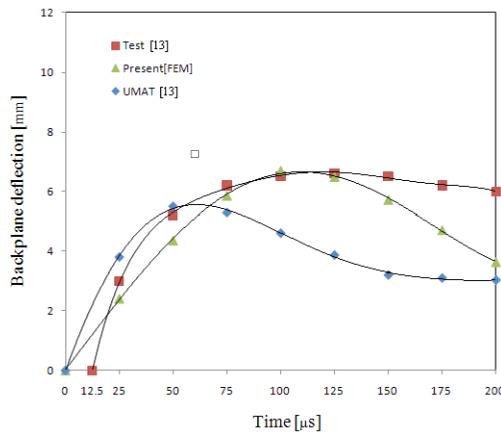


Fig. 8. Temporal variation of the backplane deflection ($V = 483$ m/s).

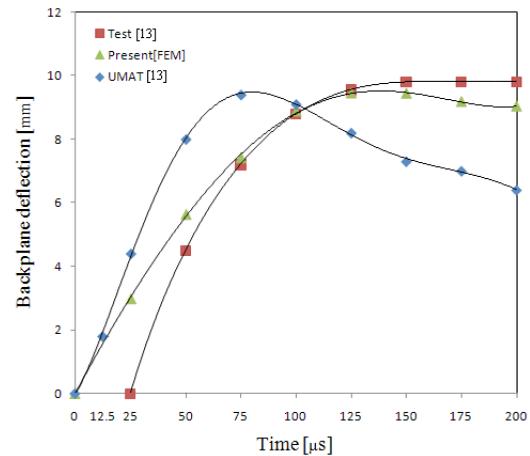


Fig. 10. Temporal variation of the backplane deflection ($V = 586$ m/s).

plastic strain, ϵ_{eff}^p and the plastic hardening stiffness, E_p .

$$\sigma_y = \sigma_0 + E_p \epsilon_{eff}^p \quad (5)$$

In Eq. (5),

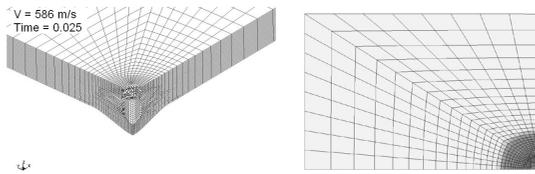
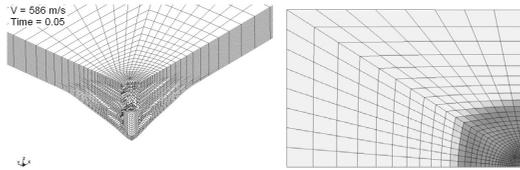
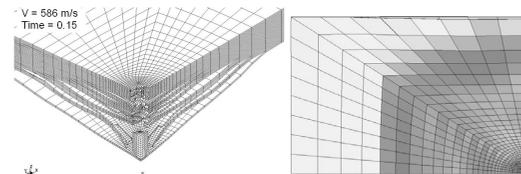
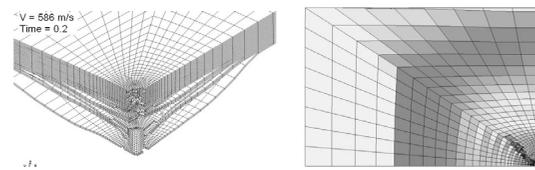
σ_0 : Specified threshold stress.

3. Results and Discussion

The failure, perforation, backplane deflection, and velocity of the impactor are the important parameters in the analytic results for a high-velocity impact on a composite plate. The behavior of the composite panel 200 μ s after the initial contact from a 586 m/s-velocity impactor is shown in Fig. 6. As shown in the figure, the panel was nearly completely perforated by the impactor. Extensive delamination occurred between the bottom plies of the panel, whereas this was not observed in the upper plies. Figure 7 shows the velocity-time history of the impactor. The present results of finite-element

analysis are in good agreement with the aforementioned test results (Hoof, 1999).

The results regarding the backplane deflection of a composite plate under various impact velocities are shown in Figs. 8-10 in comparison with the test and numerical results based on the UMAT model (Hoof, 1999). As discussed in a relevant study, namely (Hoof, 1999), the initial time of commencement of deflection was not precisely measured during the test. Except for the very early stage of the impact event, the deflection was believed to have been measured correctly. Accordingly, the authors of the aforementioned study, namely (Hoof, 1999), did not shift the test curve to position the starting point at the origin. This is also true for the present paper. The finite element results here show excellent agreement with the test results, including the maximum deflection, apart from the early stage of the event. The springback phenomenon is natural after the maximum deflection under the impact occurs. While the test results do

Fig. 11. Failure of a composite plate ($t = 25 \mu\text{s}$).Fig. 12. Failure of a composite plate ($t = 50 \mu\text{s}$).Fig. 13. Failure of a composite plate ($t = 150 \mu\text{s}$).Fig. 14. Failure of a composite plate ($t = 200 \mu\text{s}$).

not account for this phenomenon, it is easily observable in the finite element analyses.

Figures 11-14 show the progress of damage in the composite plate at an impact velocity of 586 m/s. The entire process from the initial impact to the final penetration of the impactor through the plate was successfully simulated. Figure 11 shows the damage at 25 μs after the initial strike by the impactor. Regardless of the short time for the response, the impactor penetrated into more than half the plate and the backplane deflection reached nearly 3 mm. However, the damage zone was limited to the region close to the impact point. The finite elements where the impactor passed through failed and were therefore eroded. Figure 12 shows the results at 50 μs after the initial impact. The maximum backplane deflection was approximately 6 mm, which was double the value at $t = 25 \mu\text{s}$. Extensive delamination around the impactor and through the thickness are observed in the figure. Figures 13 and 14 show the damage of a composite plate at $t = 150$ and 200 μs , respectively. Delamination in the layers close to the backplane occurred over nearly the entire plate, and the impactor began to leave the plate after penetration.

4. Conclusions

A finite element model using LS-DYNA was developed to simulate the high-velocity impact response of a Kevlar29/Phenolic composite plate. The interface between the laminas was modeled by a tiebreak contact algorithm, while the interaction between the impactor and the laminate was simulated using a surface-to-surface eroding contact algorithm. Numerical analyses were conducted at impact velocities of 483, 545, and 586 m/s of a steel impactor. The

results of finite element analysis were in good agreement with the test results and in especially excellent agreement regarding the maximum backplane deflection. The present finite element model also successfully simulated the progress in damage from the initial impact to the final penetration of the composite plate.

Acknowledgements

This work was supported by Priority Research Centers Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education, Science and Technology (2009-0094015) and the Degree and Research Center for Aerospace Green Technology (DRC) of the Korea Aerospace Research Institute (KARI) funded by the Korea Research Council of Fundamental Science & Technology (KRCF).

References

- Hoof, J. V. (1999). *Modeling of Impact Induced Delamination in Composite Materials*. PhD Thesis, Carleton University.
- Ji, K. H. and Kim, S. J. (2006). Direct numerical simulation of composite laminates under low velocity impact. *Journal of the Korean Society for Composite Materials*, 19, 1-8.
- Jung, W. K., Ahn, S. H., Lee, W. I., Kim, H. J., and Kwon, J. W. (2006). Fracture mechanism of ceramic/glass-fiber-reinforced-composite laminate by high velocity impact. *Journal of the Korean Society of Precision Engineering*, 23, 170-176.
- Kong, C. D., Kim, Y. K., and Lee, S. H. (2007). Comparative study on low-velocity impact behavior of graphite/epoxy

composite laminate and steel plate. *Journal of the Korean Society for Composite Materials*, 20, 1-6.

Lee, B. W., Lee, S. H., Kim, S. G., Yoon, B. I., and Paik, J. G. (2008). A study on the low speed impact response and frictional characteristic of shear thickening fluid impregnated Kevlar fabrics. *Journal of the Korean Society for Composite Materials*, 21, 15-24.

Lee, B. L., Walsh, T. F., Won, S. T., Patts, H. M., Song, J. W., and Mayer, A. H. (2001). Penetration failure mechanisms of armor-grade fiber composites under impact. *Journal of Composite Materials*, 35, 1605-1633.

Liu, D. (2004). Characterization of impact properties and damage process of glass/epoxy composite laminates. *Journal of Composite Materials*, 38, 1425-1442.

Livermore Software Technology Corporation (2007). *LS-DYNA Keyword User Manual*. Livermore, CA: Livermore Software Technology Corporation.

Livermore Software Technology Corporation (2006). *LS-DYNA Theory Manual*. Livermore, CA: Livermore Software Technology Corporation.

Niu, M. (2005). *Composite airframe structures*. Hong Kong: Hong Kong Conmilit Press, Ltd.

Park, I. J., Jung, S. N., Kim, D. H., and Yun, C. Y. (2009). General purpose cross-section analysis program for

composite rotor blades. *International Journal of Aeronautical and Space Sciences*, 10, 77-85.

Park, J. B., Kim, D. R., Kim, H. G., and Hwang, T. K. (2008). Evaluation of residual strength of CFRP pressure vessel after low velocity impact. *Journal of the Korean Society for Composite Materials*, 21, 9-17.

Schweizerhof, K., Weimar, K., Munz, T., and Rottner, T. (1998). Crashworthiness analysis with enhanced composite material in ls-dyna-material and limits. *Proceedings of the 5th LS-DYNA World Conference*, Southfield, MI.

Tabiei, A. and Chen, Q. (2001). Micromechanics based composite material model for crash worthiness explicit finite element simulation. *Journal of Composite Materials*, 14, 264-289.

Tabiei, A. and Ivanov, I. (2002). Computational micro-mechanical model of flexible woven fabric for finite element impact simulation. *International Journal for Numerical Methods in Engineering*, 53, 1259-1276.

Unosson, M. and Buzaud, E. (2000). *Scalar and Vectorized User Defined Material Routine in LS-DYNA. Methodology Report*. Tumba: FOA Defence Research Establishment.

Yen, C. F. (2002). Ballistic impact modeling of composite materials. *Proceedings of the 7th International LS-DYNA Users Conference*, Detroit, MI.