ID-1193

STUDY ON IMPACT BEHAVIOR OF COMPOSITES USING THE FINITE ELEMENT METHOD

Volnei Tita and Jonas de Carvalho
Department of Mechanical Engineering, S. Carlos Engineering School,
University of S. Paulo, Brazil

SUMMARY: During the last two decades, research on the development of structural components with high crashworthiness has been carried out by the automobile industries, aeronautics, naval, trains and elevators. The use of composite materials in these components has grown up higher and higher and many methodologies have been created to predict the absorption energy capacity of composites structures loaded with impact forces using the Finite Element Method (FEM). Thus, it becomes necessary to model the non-linear problems involved , such as the contact between bodies that collide, the plastic deformations of material and large displacements on the structure. In order to address all these problems, this work proposes a methodology to evaluate crashworthiness in composite structures, and gives a first contribution on the modeling of the non-linear contact problem. A case study where a composite plate is impacted by one steel sphere is studied and the results are described and compared with those from the literature.

KEYWORDS: composites, impact, finite element method

INTRODUCTION

Among all the numerical methods available, the Finite Element Method (FEM) has been increasingly used to predict the absorption energy capacity of composites structures loaded with impact forces (Kindervater and Georgi) [1] (Haug and De Rouvray) [2]. More recently, some works have carried out numerical analyses and proposed methodologies to predict composite structure behavior under impact loads. Pérez Galán, Climent and Le Page [3] performed numerical simulations for metallic and composite fuselage structures using a commercial explicit finite element code (PAM-CRASH). These numerical simulations were validated by a full-scale drop test. Vicente, Béltran, and Martínez [4] made simulations to reproduce accurately the non-linear dynamic response and complex collapse mechanisms of composite frames using a commercial explicit finite element code (ABAQUS). Thus, several modeling strategies were tested, eventually leading to the development of equivalent isotropic material models for representing the mechanical behavior under the conditions of interest. The strategy adopted by authors was to capture macroscopically the failure of the material instead of representing composite divided in plies. Therefore, it was necessary to develop an equivalent material characterized by a few parameters. After several attempts, it was decided to define an elastic-plastic constitutive model with a failure criterion governed by the maximum equivalent plastic strain of the material. In such approximation, the involved parameters of the models had to be calibrated. Such calibration was performed differently , depending on the expected importance of the part. Kindervater et al [5] realized numerical crash and high velocity impact simulation of metallic and composite aircraft sub-components using a commercial explicit finite element code (PAM-CRASH). The major tasks were focused on material studies, failure mechanisms and criteria, generation of new composite fabric models, new design concepts, numerical simulations and correlation with experimental data. Thus, a methodology was developed for simulating the response of composite aircraft structures to high velocity impacts (HVI), such as bird strike or foreign object damage on wing leading edges, engine fan blades etc.. Procedures to measure the mechanical properties of composite materials under large strain and high strain rate loading were developed. New materials constitutive laws and failure models for composites under HVI loading were developed and implemented into explicit dynamic codes.

This work proposes a methodology to predict absorption energy capacity of composites components loaded with impact forces using numerical simulations and experimental tests. In a preliminary implementation, the numerical simulations use a commercial explicit finite element code (ANSYS/LS-DYNA) and the experimental tests are formed by quasi-static tests as well as dynamic tests (Charpy impact test and drop-test). Quasi-static tests are used to estimate material properties of laminate in tensile, compression and flexure. The Charpy impact test is used to calibrate the material model implemented into finite element code considering the influence of strain rate. Later on, the drop-test is used to validate the model of component built in finite element. However, the contact problem between bodies must be evaluated before predicting the material behavior. Thus, the focus of this work is to investigate the capacity of contact algorithm from ANSYS/LS-DYNA in modeling impact on composite structures.

IMPACT PROBLEM

Considering *N* bodies in contact in time *t*. ${}^tS_C$ is the total area of contact for each body L, L= 1, ..., *N*. The principle of virtual works for *N* bodies in time *t* gives (Bathe) [6]:

$$\sum_{L=1}^{N} \left\{ \int_{{}^tV} {}^t\tau_{ij}\,\delta_t\varepsilon_{ij}\, d\,{}^tV \right\} = \sum_{L=1}^{N} \left\{ {}^tR_i \right\} + \sum_{L=1}^{N} \int_{{}^tS_C} \delta u_i^C\, {}^t f_i^C\, d\,{}^tS \qquad (1)$$

where: ${}^t f_i^C$ corresponds to contact forces components by unit of area in time t and ${}^t s_C$ corresponds to the surface where these forces act. The effect of the contact forces is included as a contribution to the tension forces applied. ${}^t\tau_{ij}$ are the cartesian components of the Cauchy' s stress tensor and the strain tensor is denoted by $\delta_t\varepsilon_{ij}$.

Assume that δu_i are the virtual displacements applied to the current time *t*. The forces acting on the body are given by Equation 2:

$${}^t\mathrm{R} = \int_{{}^tV} {}^t f_i^B \delta u_i d\,{}^tV + \int_{{}^tS_f} {}^t f_i^S \delta u_i^S d\,{}^tS - \rho_i H\delta\ddot{u} - \eta_i H\delta\dot{u} \qquad (2)$$

where: ${}^t f_i^B$ are body forces by unit of volume; ${}^t f_i^S$ are surface forces by unit of area and ${}^tS_f$ corresponds to the surface where the tension forces are applied. H corresponds to the shape function of the finite element used to mesh the body; ρ is the mass related property and η is the damping related property. The terms $\rho_i H\delta\ddot{u}$ and $\eta_i H\delta\dot{u}$ are more relevant when impact loads are present. It should be noted that the left side of equation (1), which represents the internal forces on the loaded body, are directly related to the material strain. This material strain is directly dependent on the plasticity and failure criteria adopted, which for composite materials are far more complex.

In general, the impact problem can be divided in two important problems. The first problem is involved by contact between a body called "*contactor*"(slave surface) and another body called "target"(master surface). The ANSYS/LS-DYNA program uses Penalty Method to solve contact problem [8]. Thus, each slave node is checked for penetration through the master surface. If the slave node does not penetrate nothing is done. If it does penetrate an interface

force is applied between the slave node and its contact point. The magnitude of this force is proportional to the amount of penetration.
The second problem involved is characterized by the non-linear material behavior during the analysis. The literature describes many different material models, but the engineer should be careful to choose the more appropriate to his application.

Finally, there are several numerical strategies to solve impact problems. Some are explicit (Central Differences Method) and others are implicit (Houbolt, Wilson and Newmark Methods). Each one has its advantages and difficulties (Bathe)[6](Owen and Hinton)[7]. In this work, it was used explicit integration (Central Differences Method) implemented in the commercial package ANSYS /LS-DYNA [8].

PROPOSED METHODOLOGY

The proposed methodology attempts to evaluate the impact behavior of components made from composite materials. This methodology is basically a numeric-experimental procedure, where finite element models are calibrated by experimental tests.

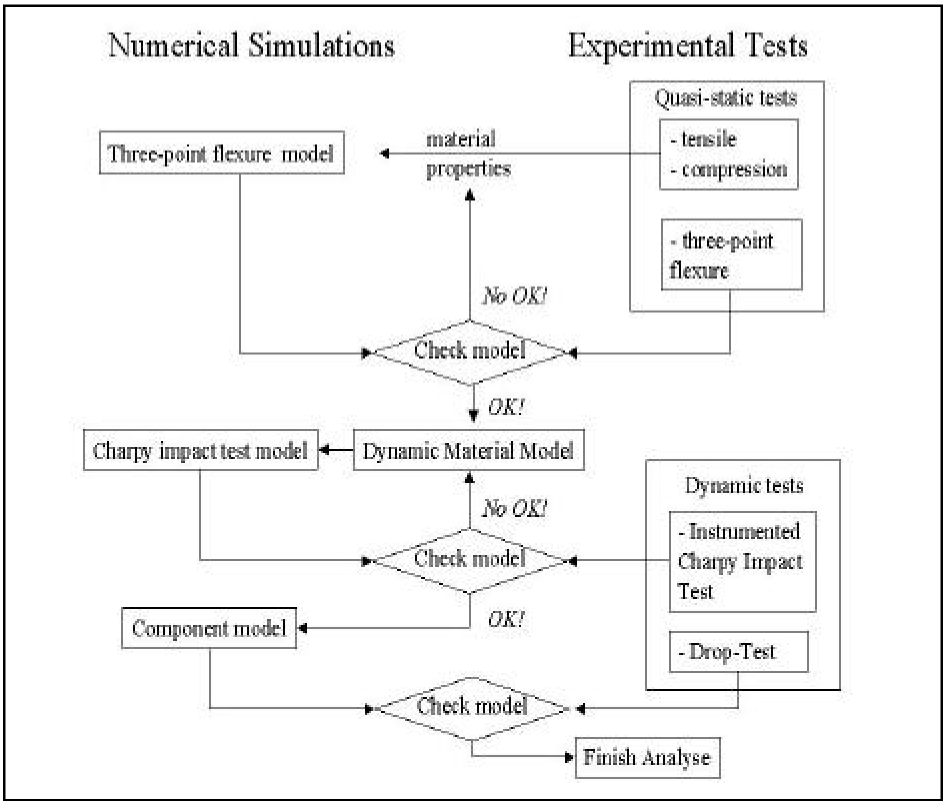


Fig. 1 Proposed Methodology

Quasi-static experiments are used to estimate material properties of unidirectional laminate in tensile, compression and three-points flexure tests. Later on, a finite element model of three-point flexure is created and material properties are set using data from tensile and compression tests. Then the simulation results are compared to the flexure tests. A new model is built to simulate Charpy impact test, considering the influence of strain rate. This model is calibrated by results from instrumented Charpy impact test. Finally, the finite element model of the component is built using the material model previously calibrated. A drop-test is used to validate this model. However, in order to apply this methodology it becomes necessary to evaluate the contact problem and material non-linearities with accuracy. A preliminary simulation of the contact problem is done by using a commercial finite element package and the results are compared to those from literature.

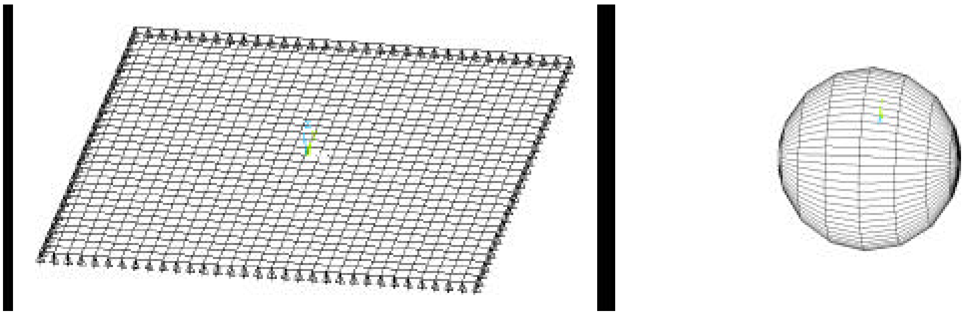
CASE STUDY

The contact algorithm (Central Differences Method), implemented in the commercial package ANSYS /LS-DYNA [8] , is tested using the problem of a laminated plate impacted by one steel sphere. The results obtained are compared with those from the literature (Sun and Chen)[9] (Christoforou Swanson)[10] (Yigit and Christoforou)[11].

The case study is based on a steel sphere with diameter equal to 12.7 mm and mass equal to 8.40 g which impacts a composite square plate at $3.00 ms^{-1}$. The plate has an edge equal to 200mm and is simply-supported. The stacking sequence is symmetric $[0/90/0/90/0]_s$ with the following elastic properties:

$$E_{11} = 141.2GPa;\ E_{22} = 9.72GPa;\ G_{12} = 5.53GPa;$$
$$G_{23} = 3.74GPa;\ \nu_{12} = 0.30;\ \rho = 1536\ Kgm^{-3}.$$

There were used 2048 shell elements with Hughes-Liu formulation [8] to model the plate and 1024 solid elements to model sphere (Fig.2), yielding to a full model with 3148 nodes. It was adopted elastic behavior of the composite plate with orthotropic material properties and rigid behavior of the steel sphere. The analysis was performed considering 300 µs as total impact time and it was divided in 150 steps of 2 µs each .



Composite Plate Steel Sphere

Fig. 2 Finite element models

It was considered non-linear contact where the plate was taken as "contactor" (slave surface) and sphere the "target" (master surface).

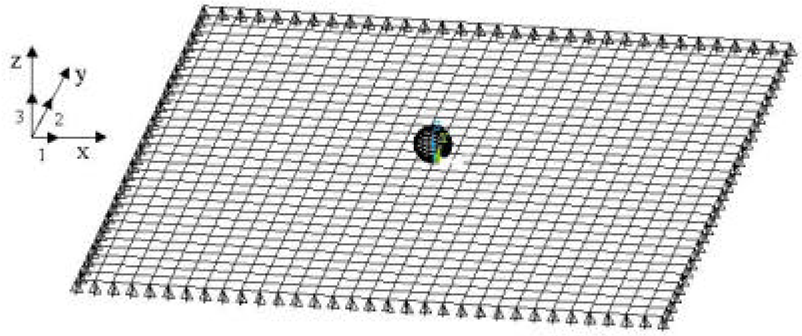


Fig. 3 Contact Model

Figure 4 shows results obtained from simulations where only the non-linearity of contact is considered, using the explicit approach. As comparison, some other results from literature are included [9,10]. The comparison indicates that the present result gives a reasonable match with the finite element calculation of Sun and Chen [9], and also with the results given by Wu [15], who also used the finite element method. However, the present results do not give a reasonable match with the analytical calculation of Christoforou and Swanson [11], who used

Fourier series expansion combined with Laplace transformation techniques. However, it should be considered that analyses they performed were not exactly the same as here simulated since different material properties were used. The displacement results of the plate is shown in Fig.5.

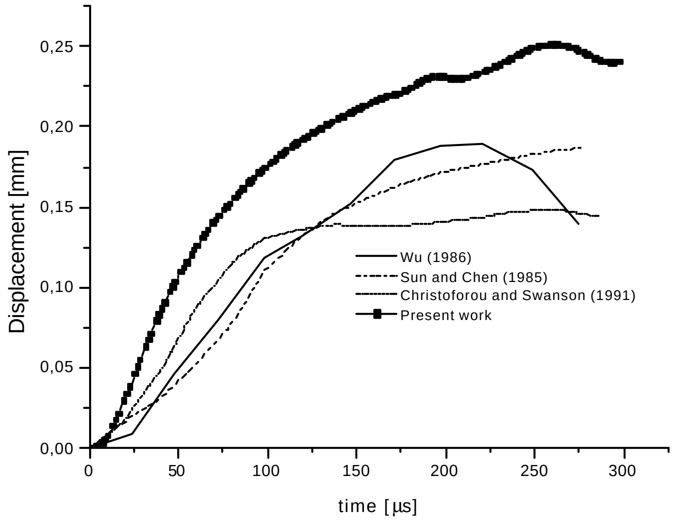


Fig. 4 Deflection of plate center

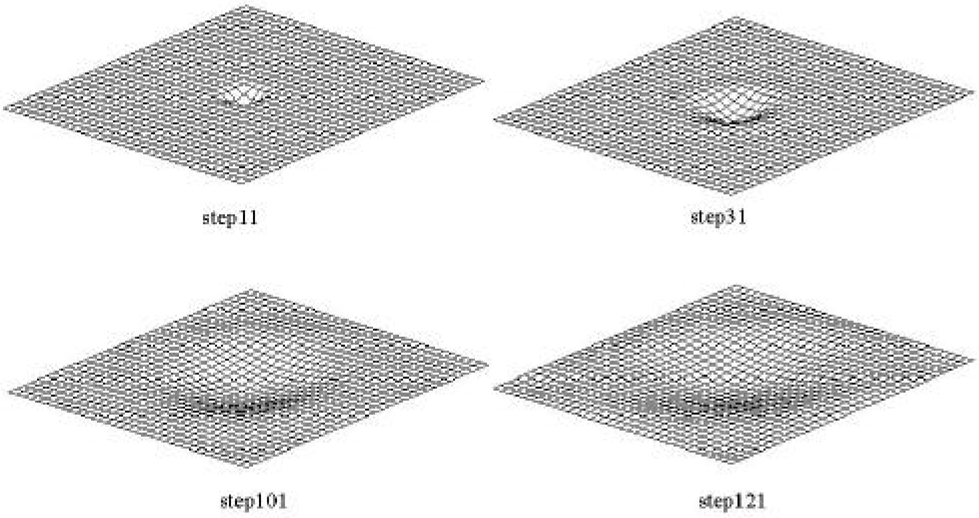


Fig. 5 Calculated displacement response for composite plate

Figures 4 and 6 indicate that the impact response is ‘dynamic’, i.e., is a combination of local and structural effects. The zero portion at Fig. 6 indicates loss-of-contact of the sphere with the plate. Sphere loses contact not because it bounces back upon impact as usually thought, but because the plate moves away faster than the sphere can follow. The sphere bounces back only at the end of the impact cycle when the sphere regains velocity in the opposite direction, provided the plate is not fractured or permanently deformed (Marur and Nair)[13].

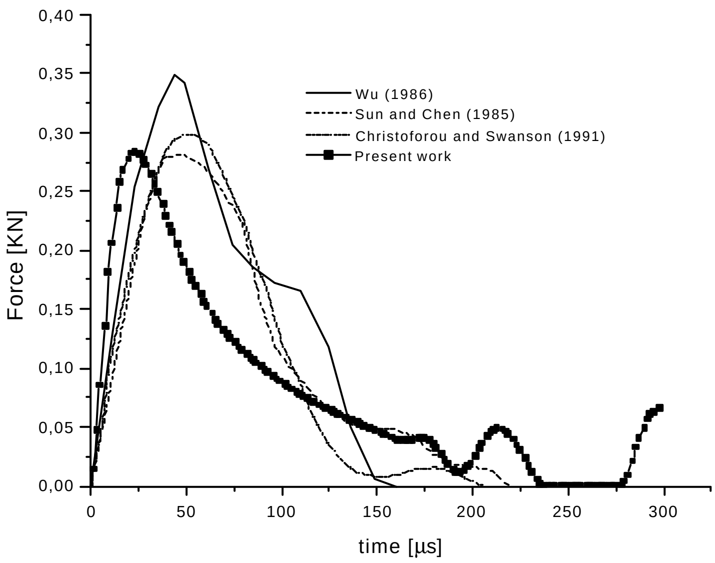


Fig. 6 Contact Force

The total energy in the system is divided between the kinetic and strain energies of the plate and sphere in a time-varying manner. The sphere is modeled as rigid body so its strain energy is zero (Fig. 7). Damping effects are not considered, thus the total system energy is constant and equal to the initial kinetic energy of the sphere. The kinetic energy of the sphere decreases rapidly as the sphere slows during contact with the plate. The other side, the strain and kinetic energy in the plate increase rapidly during the contact and remain constant after contact has ended (Trowbridge, Grady and Aiello) [12].

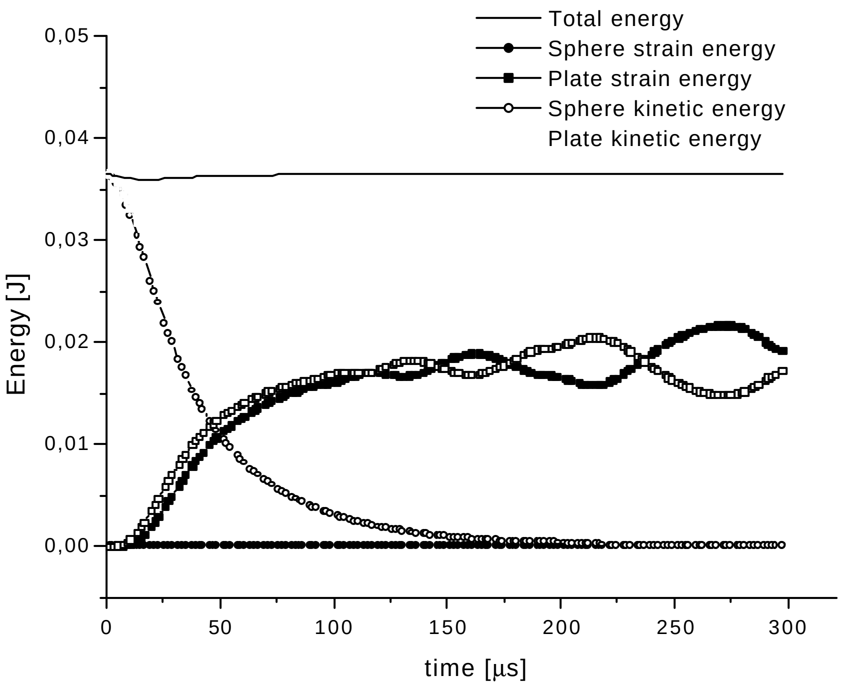


Fig. 7 Energies

Both strain and kinetic energies of the plate show close values until the compressive stress wave generated by the impact reaches the far end of the plate. A tensile stress wave is generated when the compressive pulse reflects from the stress free boundary (Trowbridge, Grady and Aiello) [12]. For example, the superposition of the incident and reflected pulses momentarily leaves the plate stress-free. As example, Fig. 8 shows that close to 230 µs the generalized normal stress in direction x decreases to zero.

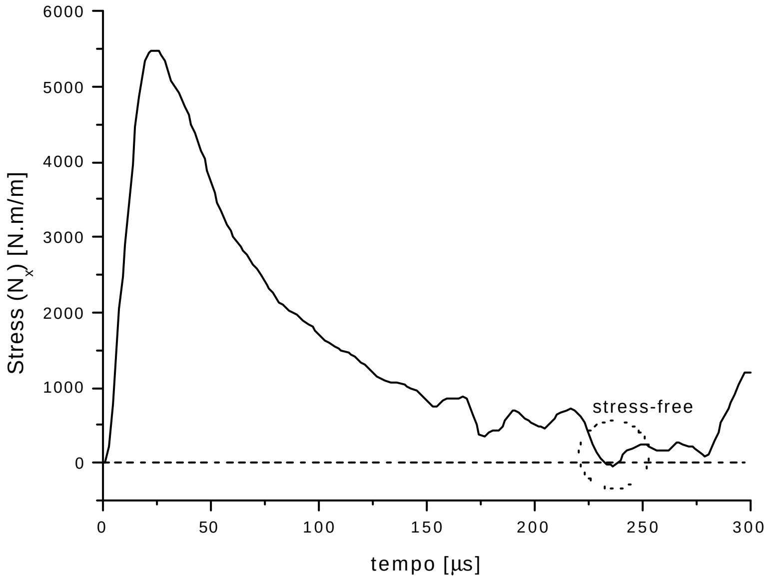


Fig. 8 Generalized stress in plate center

CONCLUSION

The contact algorithm (Central Differences Method), implemented in the commercial package ANSYS /LS-DYNA [8] , was used to define the force-indentation relationship for impact problem between composite plate and steel sphere. The results show good agreement with those from literature. Thus, this contact algorithm can be used when implementing the proposed methodology. Future work will address the non-linear material effects such as plasticity and damage in quasi-static and dynamic analyses.

ACKNOWLEDGEMENTS

This work is part of a project to evaluate the absorption energy capacity of composites structures loaded with impact forces and it is supported by FAPESP (Research Foundation of S. Paulo State – Brazil).

The authors would also like to thank the CAD/CAE Laboratory (University of S. Paulo) for the support in the use of the package ANSYS/LS-DYNA.

REFERENCE

1. Kindervater, C.M. and Georgi, H., “Composite strength and energy absorption as an aspect of structural crash resistence”, *Proc. of Structural Crashworthiness and Failure*, 14-16 April, 1993, Liverpool, UK.
2. Haug, E. and De Rouvray, A., “Crash response of composite structures”, *Proc. of Structural Crashworthiness and Failure*, 14-16 April, 1993, Liverpool, UK.
3. Pérez Galán, J.L., Climent, H. and Le Page, F., “Non-linear response of metallic and composite eronautical fuselage structures under crash loads and comparison with full scale test”, *Proc. of ECCOMAS*, 11-14 September, 2000, Barcelona, Spain.
4. Vicente, J.L.S., Béltran, F. and Martínez, F., “Simulation of impact on composite fuselage structures”, *Proc. of ECCOMAS*, 11-14 September, 2000, Barcelona, Spain.

5. Kindervater, C.M., Johnson, A.F., Kohlgrüber, D., Lützenburger, M. and Pentecôte, N. "Crash and impact simulation of aircraft structures-hybrid and FE based approaches", *Proc. of ECCOMAS*, 11-14 September, 2000, Barcelona, Spain.
6. Bathe, K-J. *Finite Element Procedures*. Prentice-Hall: London, 1996.
7. Owen, D.R.J. and Hinton, E. *Finite Elements in Plasticity: Theory and Practice*. Pineridge Press Limited: Swansea, 1986.
8. Ansys/Ls-Dyna Theoretical Manual –Ansys, Inc., 1998.
9. Sun, C.T. and Chen, J.K. "On the impact of initially stressed composites laminates", *J. of Composite Materials*, 1985, Vol. 19.
10. Christoforou, A.P. and Swanson, S.R. "Analysis of impact response in composite plates", *Int. J. of Solids Structures*, 1991, Vol. 27, No. 2.
11. Yigit, A.S. and Christoforou, A.P. "Impact dynamics of composites beams", *Composite Structures*, 1995, Vol. 32.
12. Trowbridge, D.A., Grady, J.E. and Aiello, R.A "Low velocity impact analysis with NASTRAN", *Computers & Structures*, 1991, Vol. 40, No. 4.
13. Marur, P.R. and Nair, P.S. "Two degrees of freedom modeling of precracked beams under impact", *Engineering Fracture Mechanics*,1996, Vol. 53, No.3.
14. Chun, L. and Lam, K.Y. "Behavior of uniform anisotropic beams of rectangular section under transverse impact of a mass", Shock and Vibration, 1997, Vol 4, No. 2.
15. Wu, H.T. "Impact damage of composites", Ph.D. Dissertation, Department of Aeronautics and Astronautics, Stanford University, 1986.