International Journal for Ignited Minds (IJIMIINDS)

Study of Low-Velocity Impact Induced Damage on a Composite Drive Shaft

Thimmaiah A G[a], Sujith N S[b], Subbaiah K S[c] & Kaveriappa M B[d]

[a,b]*Asst. Prof, Dept of Mechanical Engineering, Coorg Institute of Technology, Karnataka, India.*
[c,d] *Asst. Prof, Dept of Mechanical Engineering, Coorg Institute of Technology, Karnataka, India.*

ABSTRACT

Composite materials have the very high strength to weight ratio so, they are now being widely used in aerospace and automobile structures but, as with all engineering materials, it is important to fully understand their properties and behavior before it is included in the structures. An important consideration when designing composite structures is its vulnerability to damage caused by impact loading. Even under low-velocity impact conditions, composites are susceptible to internal damage caused by transverse loads. Unlike metallic structures, material damage for composites can be hidden within the material and show no form of external damage. For the numerical work conducted in this study, the finite element software ANSYS/LSDYNA has been employed to composite drive shaft HM Carbon/epoxy for automotive applications. The outcome of the results shows that the comparisons for displacement, velocity and contact force are seen that the numerical results provided by ANSYS/LS-DYNA are in good agreement with theoretical values.

Keywords *– Composites, Low-Velocity Impact, Drive Shaft, Carbon/Epoxy, ANSYS/LSDYNA.*

1. INTRODUCTION

Composite materials can be made to efficiently meet the design requirements of strength, stiffness, and other parameters easily compared to conventional materials. These are also well known to possess very high stiffness to weight ratios. Thus, components made of composite materials would be of significant interest for design improvements in areas where weight reduction is essential without sacrificing the strength of the material, such as aerospace and automotive applications. While composites have been known for various applications in the automotive industry, especially in brake systems, pump housings, and fuel storage tanks. Composite drive shaft applications have received new impetus during the last two decades. It is known that energy conservation is one of the most important objectives in vehicle design. The weight reduction is one of the most effective measures to obtain the result.

Impact generally results in internal cracking and delamination for lower energies level, while high impact energies cause penetration and excessive local shear damage [1]. The problem is generally complex, involving the structural response of the impacted object, the effect of subsequent stress states on damages, and finally the effect of this damage on the strength and stiffness of the structure.

Various other parameters which define the morphology of the impact process include impactor velocity, geometric constraints applied to the system, impactor shape, and dimensions of the impacted structure. Therefore studies of these parameters are important in understanding the impact process and the damage caused by them in the composite structures. Although a significant amount of work is being done to characterize damage for flat plate composite laminates, there has been little work on characterizing and modeling of low-velocity impact effects on the curved structures such as tubes, vessels, and hollow shafts. Due to their curved profile, these structures possess natural boundary constraint.

Damage propagation on the curved laminates was studied analytically by Ganapathy and Rao [2]. For their finite element analysis, they used four node quadrilateral elements for determining the damage on the cylindrical panels due to low-velocity impact. The progressive failure analysis due to impact was carried out by considering the fiber breakage and matrix cracking modes of composite failure. Hwang and Sun [3] used an iterative three-dimensional finite element method for failure analysis of laminated composite. Unlike the isotropic, homogeneous materials such as the metals, the yield and the post yield theories for structural behavior after the onset of the permanent deformation were experimentally verified. Because composite laminates are heterogeneous, the interaction of the matrix and fiber upon the failure of the lamina or laminate is significant [4]. For the numerical work conducted in this study, the finite element software ANSYS/LSDYNA has been employed to composite drive shafts of HM Carbon/epoxy for automotive applications subjected to low-velocity impact.

2. PROBLEM FORMULATION

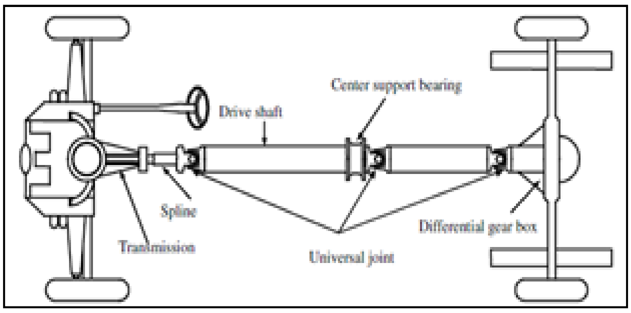


Fig 1: Conventional Two-Piece Steel Driveshaft [5].

The study of the impact and impact-induced damage in composite laminates has been approached by various researchers through experimental and analytical techniques. The experimental methods are essential for direct observation; they are expensive for parametric studies. In Analytical methods it is very difficult to analyze the impact damage. The finite-element method, on the other hand, is suited for simulating the physical phenomena such as damage initiation and propagation through the use of appropriate failure criteria.

The finite element analysis is generally based on the use of the modified Hertz contact law from static indentation tests on surfaces which are the plane for impact analysis. The analysis of impact and the prediction of impact-induced damage by means of the finite-element method can be categorized into the following steps:

(a) To investigate the contact force between impacting mass and the shell as a function of time.

(b) To Apply the contact force and find the transient dynamic response of the impacted structure as a function of position and time (displacements, strains, stresses, etc.) which depends on the mass and velocity of the impactor and the laminate characteristics (geometry, boundary conditions, ply arrangement, elastic properties, etc.).

(c) Predicting damage in the composite shaft using appropriate failure criteria.

2.1 Solution for the Contact Force

The impact force P and contact deformation α relation for impact of two bodies of revolution are given by [7] the Hertz law.

$$P = n\alpha^{3/2} \tag{1}$$

Where, n is the contact stiffness parameter, which depends on material and geometrical properties of the plate and the impactor. The expression for n, for an isotropic impactor and transversely isotropic composite plate, is given by

$$n = \frac{4\sqrt{R1}}{3\pi(K_1 + K_2)} \tag{2}$$

The contact deformation $\alpha = \frac{E_c}{1/2M_1 v_0^2}$

Where $K_1 = \frac{1-v_i^2}{\pi E_i}$

$$K_2 = \frac{\sqrt{A_{22}}\left[\left(\sqrt{A_{11}A_{22}} + G_{zr}\right)^2 - \left(A_{12} + G_{zr}\right)^2\right]^{1/2}}{2\pi\sqrt{G_{zr}}\left(A_{11}A_{22} - A_{12}^2\right)}$$

$$A_{11} = E_z\left(1 - v_r\right)\beta, \; A_{22} = \frac{E_r\beta\left(1 - v_{zr}^2\delta\right)}{\left(1 + v_r\right)}$$

$$A_{12} = E_r v_{zr} \beta, \; \beta = \frac{1}{1 - v_r - 2 v_{zr}{}^2 \delta}$$

$$\delta = \frac{E_r}{E_z}$$

The constants Ei and v_i are, respectively, Young's modulus and Poisson's ratio of the impactor. The constants E, G, and v are, respectively, Young's modulus, shear modulus, and Poisson's ratio of the laminates, while the subscripts r and z refer to radial and thickness directions, respectively. The contact energy E_c is then the integral of the product of the impact force and contact deformation:

$$E_c = \frac{2}{5} \frac{P^{5/3}}{n^{2/3}} \tag{3}$$

The reactive force P from the plate can be resolved into two components

$$P = P_{bs} + P_m \tag{4}$$

Where, P_{bs} is the force associated with bending and shear deformations and the P_m is the force associated with membrane deformation. Using the force-deflection relation, the force P is written as

$$P = K_{bs} w + K_m w^3 \tag{5}$$

Where, $K_{bs} = \frac{(K_b K_s)}{(K_b + K_s)}$ is the effective stiffness due to bending and shear.

The constants K_b, K_s, and K_m are bending, shear, and membrane stiffness, respectively of the target. Expressions for K_b and K_m for the four plate boundary conditions are given by:

$$K_b = \frac{4\pi E_r h^3}{3\left(1 - v_r{}^2\right) a^2}, \quad K_m = \frac{191 \pi E_r h}{648 a^2}$$

The shear stiffness K_s is given by:

$$K_s = \frac{4\pi G_{zr} h}{3} \left(\frac{E_r}{E_r - 4 v_{rz} G_{zr}} \right) \left(\frac{1}{4/3 + \log a / a_c} \right) \tag{6}$$

The contact radius a_c is the radius of contact between the impactor and the target, which depends on the force P, and which is expressed as

$$a_c = \left[\frac{3\pi}{4} P \left(K_1 + K_2 \right) R_1 \right]^{1/3} \tag{7}$$

The impact force P is initially unknown, hence an initial value of a_c = h/2 was used in equation (6) for the estimation of P. The bending-shear energy E_{bs}, and membrane energy E_m of the target, are given by:

$$E_{bs} = \frac{1}{2} K_{bs} w^2 \quad \text{and} \quad E_m = \frac{1}{4} K_m w^4 \tag{8}$$

Substituting equations (3) and (8) in equation (1), and then simplifying using equation (5), the energy-balance equation becomes

$$M_i v_0{}^2 = K_{bs} w^2 + \frac{K_m w^4}{2} + \frac{4}{5} \left[\frac{\left(K_{bs} w + K_m w^3 \right)^5}{n^2} \right]^{1/3} \tag{9}$$

The deflection w is calculated using the Newton- Raphson numerical technique. The inverse procedure of calculating the impact velocity for a chosen value of w can also be followed. The impact force P is then calculated by substituting the value of w into equation (5).

3. DEVELOPMENT OF THE FINITE ELEMENT MODEL

The finite element method is a numerical procedure that can be used to obtain solutions to a large class of engineering problems including stress analysis in dynamic conditions. The finite element is performed in three steps, namely, pre-processing, solving and post-processing. The pre-processing involves geometric modeling of the required structures or importing the CAD developed geometry to the finite element window. An appropriate meshing is applied to the modeled geometry. Material properties are assigned to the elements and boundary constraints are applied to the nodes of the element. Step two, solving, involves the processing of the geometric data and generating the output file. The third step, post-processing, involves studying the results through stress/strain and force graphs. In this study, MSC/PATRAN serves both as pre-processor and post-processor. MSC/PATRAN is interactive 3-D modeling software, used for modeling of many of the engineering components. Because of its extensive customization capabilities and simple menu-driven user interfaces, it is widely used as a powerful tool for modeling. LS-Dyna serves as a solver and LS-POST as post-processor. For simulation purposes, an LS-Dyna 970 code was used. LS-Dyna is a general purpose, non-linear explicit and implicit finite element program used to analyze the non-linear dynamic response. It has a fully automatic definition of contact areas and a large library of constitutive material models and failure models

The finite element model of the HM Carbon/epoxy composite drive shaft and impactor was developed in MSC/PATRAN. Only one quadrant of the impact system was modeled due to the symmetry. The composite drive shaft is 90 mm in diameter and 1250 mm in length and 6 to 8mm of thickness [5]. One time impact is only considered for all simulations. Fig 2 illustrates the model in MSC/PATRAN. The impactor is one-half inch diameter solid steel cylinder with a hemispherical tip. The boundary condition of the experimental test on carbon tube was also simulated in the finite element model. Proper material properties, boundary conditions, and loadings are assigned to the finite element model, similar to the experimental test setup.

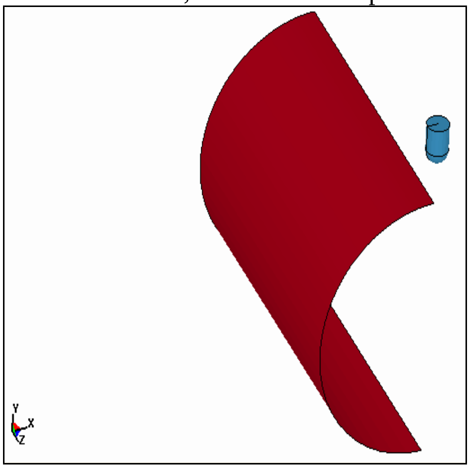
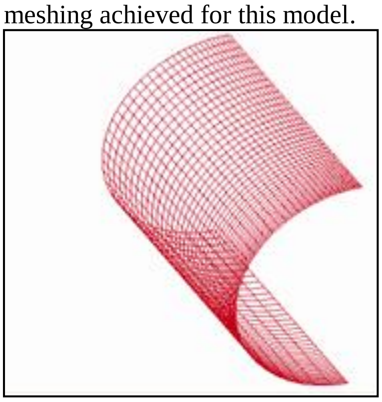


Fig 2: Model of Impact System.

3.1 Modeling Approach

Due to the geometric symmetry of the tube, only a one-quarter model was made. Symmetry boundary conditions were imposed on the tube such as x-y boundary plane were imposed by restricting the z displacement and x and z rotations, while at the symmetric y-z plane, x displacement and y and z rotations were restricted. The shell elements were located at the mid-plane of each wall. The lay-up sequence for the composite shaft was [30/-30/90/90/30/-30/90/90] [5] with reference direction coinciding with the axis of the shaft and the last 90° lamina is the outermost ply

Fig 3 illustrates the meshing achieved for this model.



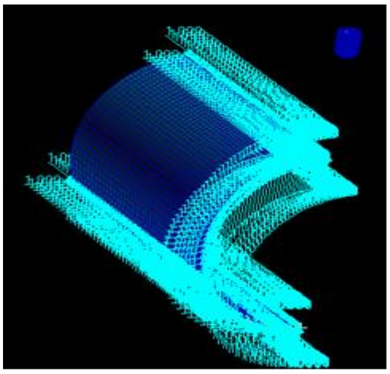


Fig 3: Meshing of Composite Drive Shaft. Fig 4: Boundary Conditions on the Composite Drive Shaft.

3.2 Material Models

The material properties of the steel impactor and carbon/epoxy tube were taken from the work of Yen and Cassin [3]. The material model selected for the steel impactor was considered to be a rigid body. The material model selected for the composite tube, which represents an orthotropic material that behaves linearly through damage parameters. Representative properties of the carbon/epoxy shaft and steel material were obtained from the published paper and are listed in Table 1.

Property	HM Carbon/Epoxy
$E_{11}(Gpa)$	190
$E_{22}(Gpa)$	7.7
$G_{12}(Gpa)$	4.2
υ_{12}	0.3
$\sigma_t^1 = \sigma_c^1 (Mpa)$	870
$\sigma_t^2 = \sigma_c^2 (Mpa)$	540
$\tau_{12}(Mpa)$	30
$\rho(kg/m^3)$	1600
V_f	0.6

Table 1: Mechanical Properties for each Lamina of the Laminate [5].

4. RESULTS AND DISCUSSIONS

Once the model was made in MSC/PATRAN and the KEY file was edited, the model was run on a UNIX machine for solving for the given loading and boundary constraints. The results were studied in post-solver LS-POST and were compared with the results in the literature and good correlation was found between the simulation and results of Yen and Cassin [3]. Fig 5, illustrates the damage caused to the composite shaft due to an impactor with velocity V= 47.94in/sec (121.76 cm/sec). The region in red shows maximum stress conditions at time t= 0.024sec.

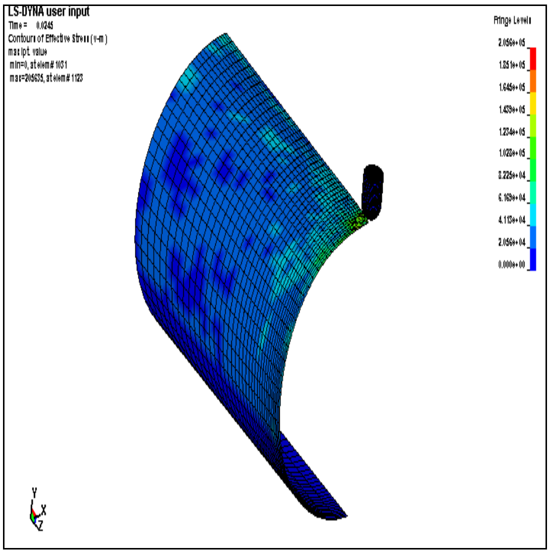


Fig 5: Stress Plots for HS Carbon/Epoxy Shaft with Lay-Up Configuration.

For the validation of the proposed FEM model, the FEM simulation results were correlated with experimental results of Yen and Cassin [3] for force vs. time. In Fig 6, a plot of impact force (lb) vs. time is shown for both FEM simulation and experimental results.

Since there is an initiation of the fiber damage mode, the impact force also decreases at that time as can be seen from Fig 6. Due to continuous loading beyond that point, there is a continuous progression of the damage to the fibers through the thickness of the structure which increases with impact load, until time t= 8.5msec (approx.) as seen from Fig 6, where there is a permanent damage caused to the structure and thus reduction in the impact force.

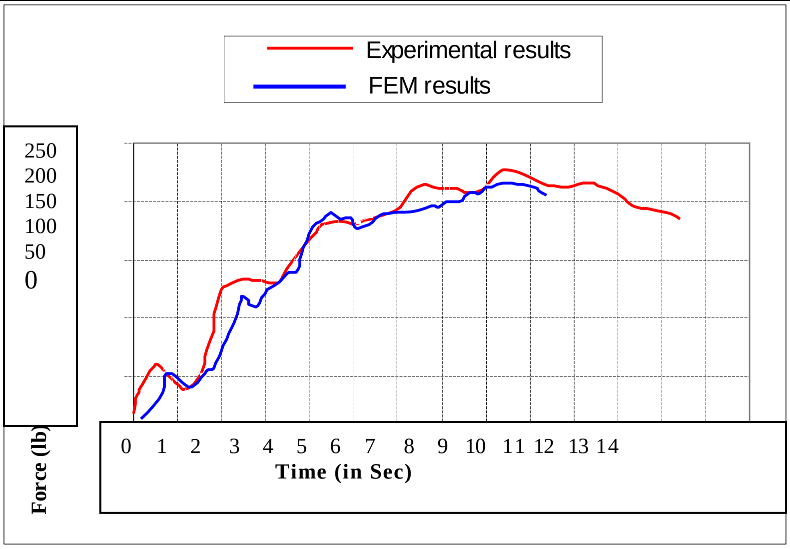


Fig 6: Comparison of LS-Dyna and Experimental Results.

Test	Peak Impact Load(kg)
FEM Simulation	96
Experimental Test	102

Table 3: Comparison of Fem and Exptl Peak Force for V = 47.04 In/Sec (119.48 Cm/Sec).

5. CONCLUSION

Finite element codes can be used effectively to simulate the low-velocity impact scenario on composite hollow shaft structures with closely predicting the failure in them. An FE model of a 90mm diameter carbon/epoxy composite drive shaft and impactor was successfully developed to analyze the structural behavior due to low-velocity impact. The difference between the FEM and experimental results for peak impact force was less than 12% and validated the model for additional parametric studies.

Also, the study of parameters such as different lay-up sequences, different impactor material properties, different boundary conditions on the composite shaft, the angle difference between the consecutive laminas and impactor with different velocities was done, which affects the impact damage process. Different parameters discussed affect the damage process significantly.

REFERENCES

1. S.Abrate, Impact *on Composite Structures*, Cambridge University Press, 1998.
2. Ganapathy Sand Rao, "Interlaminar Stresses in Laminated Composite Plates, Cylindrical/Spherical Shell Panels Damaged by Low-Velocity Impact", Proceedings of the 1997 9th International Conference on Composite Structures, Vol. 38, 1997, pp. 157-168.
3. C.T.Sun, "*On the Impact of Initially Stressed Composite Laminates*", Journal *of Composite Materials*, Vol. 19, 1985, pp. 490-504.
4. Sierakowski, Robert L, S.K.C, *Dynamic Loading and Characterization of Fiber- Reinforced Composites*, John Wiley & Sons INC., 1997.
5. T.Rangaswamy, Vijayarangan S, "*Optimal sizing and stacking sequence of composite drive shafts*", *Materials science*, Vol. 11, No. pp 133-139., 2005.
6. Yen, Chian-Fong, T.C, "*Progressive Failure of Thin Walled Composite Tubes under Low Energy Impact*", Structures, *Structural Dynamics & Materials Conference*, pp. 363-371, 1998.
7. K.N.Shiva Kumar et al, "*Prediction of Impact force and duration due to low velocity impact on circular composite laminates*", *Journal of Applied Mechanics*, Vol 52, pp 671-680, 1985.
8. LS-DYNA Keyword *User's Manual,* Ver. 2003, Livermore Software Technology Cooperation ,April 2003
9. Krishnamurthy et al, "*A parametric study of the impact response and damage of laminated cylindrical composite shells*", *Composites science and Technology*, Vol 61, pp. 1655-1669, 2001.
10. Shiuh-Chuan Her and Yu-Cheng Liang, "*The finite element analysis of composite laminates and shell structures subjected to Low-velocity impact*", *Composite Structures*, Vol 66, pp 277-285, 2004.