LOW VELOCITY IMPACT ANALYSIS OF LAMINATED FRP COMPOSITES

B. KRANTHI KUMAR [1]
M.Tech student, Department of Mechanical Engineering,
Godavari Institute of Engineering and Technology,
Rajahmundry, Andhrapradesh, India.
E-mail:Kranthikumar301@gmail.com

LAKSHMANA KISHORE.T [2]
Associate professor, Department of Mechanical Engineering,
Godavari Institute of Engineering and Technology,
Rajahmundry, Andhrapradesh, India.
tlkishore.giet@gmail.com

Abstract

Fiber reinforced composites have become increasingly important over the past few years and are now the first choice for fabricating structures where low weight in combination with high strength and stiffness are required. Fiber Reinforced Plastics (FRP) composites are in greatest commercial use. They have been extensively used in aerospace, automotive, marine and construction industries due to their inherent advantages over conventional metals. Failure modes of such laminated structures are also different than those of conventional metallic materials. Impact is one such great design limitation criteria involved in designing new composite products.

The present work is aimed at gaining an initial understanding of the impact behavior of fiberglass reinforced laminates with vinylester and is polyester resins. The purpose of this research is to characterize the damage done to fiberglass laminates subjected to low velocity, high mass impact. The effect of adding a protective layer of rubber to the laminates is also investigated.

Finite element models are created with ANSYS/LS-DYNA onlinear finite element software. These models are used to simulate the drop tower tests and extended to thicker laminates as well as different impact speeds and impactor mass.

These models are able to predict approximate stresses and strains induced in the laminates during the impact which are compared to the damage from the drop tower tests.

***Keywords*: FRP, composites, vinylester, polyester resins, stress and strain.**

1. INTRODUCTION

A Composite material is a material brought about by combining materials differing in composition or form on a macro scale for the purpose of obtaining specific characteristics and properties. The constituents retain their identity such that they can be physically identified and they exhibit an interface between one another. In addition, three other criteria are normally satisfied before we call a material as composite. Firstly, both the constituents have to be present in reasonable proportions. Secondly, the constituent phases should have different properties, such that the composite properties are noticeably different from the properties of the constituents. Lastly, a synthetic composite is usually produced by deliberately mixing and combining the constituents by various means. The constituent that is continuous and is often, but not always, present in the greater quantity in the composite is termed as the Matrix. The aim is to improve the properties of the matrix by incorporating another constituent. Generally used Matrix materials in composites are Polymers, Ceramics and Metallic Matrix. Polymers have low strength and Young’s moduli, ceramics are strong, stiff and brittle, and metals have intermediate strength and Young’s moduli, together with good ductility. The second constituent is known as the reinforcement, as it enhances the mechanical properties of the matrix. The diameter of the fibers varies from 0.1- 100 µm. Strength and stiffness properties of reinforced elements are generally 10- 1000 times higher than those of the matrix. At least one of the reinforcement is small, say <500μm and mechanical properties of composites are function of the shape, dimensions, orientation and quantity of the reinforcement.

2. Finite Element Analysis

Some of the first research on the modeling of impact damage resistance in Composite materials was on the subject of bird strikes. In 1981, R.E McCarty [1] used the nonlinear finite element software MAGNA to model birds striking the canopy of the F-16A and in 1982; M.S. Hirschbein [2] used NASTRAN to model the effects of bird strikes on turbine blades. Both of these authors claimed reasonable results.

There have been many damage criterions created for composite damage analysis in Livermore's LS-DYNA transient non-linear FEA software [3]. One of the earliest was established by Chang and Chang [4] in 1985. They created a damage model to predict damage in notched laminate composites under tensile loading. The model could be used to assess the type and the extent of the damage, evaluate residual strengths, and predict ultimate strengths. The damage was assessed with a proposed failure criteria and proposed property degradation model. They found excellent agreement between their predicted results and experimental data. The current composite damage model that is used by LS-DYNA is based on this model.

They identified the three main in plane failure modes as matrix cracking, fiber matrix shearing, and fiber breakage. The following failure criterion (equation 1) was used to predict matrix cracking:

$$\left(\frac{\sigma_Y}{Y_t}\right)^2 + \frac{\int_0^{\gamma_{XY}} \sigma_{XY} d\gamma_{XY}}{\int_0^{\gamma_{XY}^u} \sigma_{XY} d\gamma_{XY}} = e_m^2 \qquad (1)$$

Where $\boldsymbol{\sigma}_y$ is the transverse tensile stress
$\boldsymbol{\sigma}_{xy}$ is the shear stress in each layer.
Y_t is the transverse tensile strength,
$\boldsymbol{\gamma}^{\mathbf{u}}_{\mathbf{xy}}$ is the ultimate shear strain and
$\mathbf{e_m}$ is the failure strain of the matrix material.
By adding the shear stress-shear strain relationship:

$$\gamma_{XY} = \left(\frac{1}{G_{XY}}\right)\sigma_{XY} + \alpha\sigma_{XY}^3 \qquad (2)$$

where Sc is the ply shear strength.
For laminates with linear elastic behavior (a = 0),
As before, for linear elastic materials the equation can be reduced to:

$$\left(\frac{\sigma_x}{X_t}\right)^2 + \left(\frac{\sigma_{xy}}{S_c}\right)^2 = e_f^2 \qquad (3)$$

This failure criterion states that when the stresses $\boldsymbol{\sigma}_x$ and $\boldsymbol{\sigma}_{xy}$, in any one of the plies, satisfy one of the previous equations ($e_f = 1$), either fiber breakage or fiber-matrix shearing occurs in that layer.

Once damage has occurred the material looses strength in that area and the reduction is dependent on the mode of failure. For matrix cracking in a layer, the transverse modulus Ey and Poisson's ratio **γy** are reduced to zero. The longitudinal modulus $_{Ex}$ and the shear-stress strain relation of that layer are unchanged

When fiber breakage and/or fiber-matrix shearing are predicted, the extent of the property degradation depends on the size of the damage. For fiber failure, both Ey and E_x are reduced to zero. The longitudinal modulus Ex and the shear modulus G_{xy} degenerate according to the Weibull distribution:

$$\frac{E_x^d}{E_x} = \exp\left[1 - \left(\frac{A}{A_0}\right)^\beta\right] \qquad (4)$$

2.1 Elements Used

Due to advantages and disadvantages of one element over the other, as well as the parameters to be determined in view, two elements are used in the total analysis. They are Shell 163 element and Solid 164 element.

2.1.1 SHELL163 Element

SHELL163 is a 4-node element with both bending and membrane capabilities [5]. Both in-plane and normal loads are permitted. The element has 12 degrees of freedom at each node: translations, accelerations, and velocities in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. This element is used in explicit dynamic analyses only.

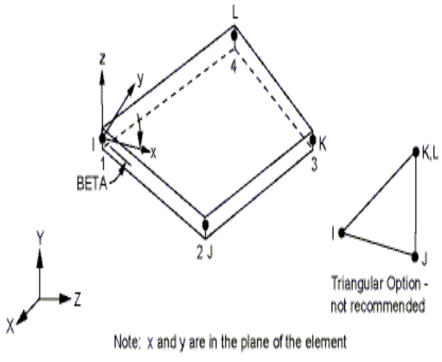

Fig. 1 Shell 163 Element.

2.1.2 SOLID164 Element

SOLID164 is used for the 3-D modeling of solid structures. The element is defined by eight nodes and nine degrees of freedom at each node: translations, velocities, and accelerations in the nodal x, y, and z directions. This element is used in explicit dynamic analyses only. This element cannot be used for models in which inter laminar stresses are to be predicted. This solid element uses an effective modulus through the thickness of the layer by averaging the elastic modulus of all the layers. This becomes a problem when the layers have drastically different moduli from layer to layer as in fiberglass laminates. This element will not take into effect the position in the laminate of the very high or very low modulus layers, which plays a key role in the bending of the laminate. It will also give inaccurate inter laminar stresses since the adjoining moduli are the key parameter for that calculation.

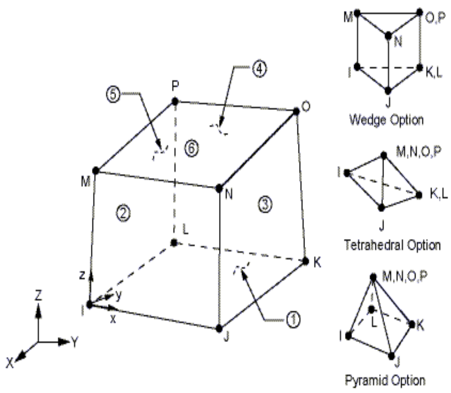

Fig.2 Solid 164 Element.

3 .Methodology :Statement of Problem

The problem is to characterize the damage occurred to fiberglass laminates subjected to low velocity, high mass impact. This subject is a crucial design question that appears frequently in the design of new composite products. Impact loads affect almost all commercial composite products whether by design or by accident. This investigation attempted to provide initial insight into impact damage done to the fiberglass laminates by performing several drop weight impact analyses with finite element models and predicted the behavior of the laminates under different impact situations. Further research is needed to evaluate the effects of impact damage on specific applications.

3.1 Non- Linear Analysis

The non- linearity arising from the nature of material is called ‘Material Non- linearity’. All non- linearities are solved by applying the loads slowly (dividing it into a number of small load increments). The model is assumed to behave linearly for each load increment, and the change in model shape is calculated at each increment. Stresses are updated from increment to increment, until the full applied load is reached.

4. Finite element analysis

Finite element analysis is used to gain information about the behavior of the composite laminates subjected to impact loading. Simple models are analyzed and more complex models are used to simulate additional parameters such as thick laminates and rubber protecting the composite. FEA is used to predict the stresses and strains induced in the laminate by a low velocity projectile. These stresses and strains can later be used to predict the life span of the composite under various loading conditions. ANSYS and ANSYS/LS-DYNA finite element codes are used for the simulations.

4.1 Static Model

Initially a static model is created to get an idea of the loads caused by a 1.25 m. diameter granite rock. Several simple shell models are made with pressures in various locations. The resulting stresses and strains are very low and well within the capability of the composite even without a rubber liner. From this it is decided that the impact is going to be the limiting factor in the design.

4.2. Results and Discussions

The block is placed at several distances from the shell and dropped onto it. The maximum deflection in the center of the shell converged to –0.3048 μm as the initial distance is decreased, table This final deflection compared to the static deflection represents an impact factor of 2.2, which is close to the 2.0 predicted by mechanics of materials.

Table 4.1 Convergence of deflection and stress

Distance from target (mm)	0.254	0.0254	0.00254	0.000254	0.0000254
Deflection (μm)	-0.11684	-0.4826	-0.03175	-0.03048	-0.03048
Stress Intensity (N/m2)	21.7	9.1	5.95	5.775	5.705

4.3 Drop Tower Model

The next step is to model the drop tower test in order to predict stress and strain information. Two separate ANSYS/ LS-DYNA models are created in order to compare the results of the two and pick one to use to model the tests. One target is created with shell 163 elements and one with solid 164 elements. The shell element has the advantage of fewer elements and degrees of freedom, which lowers the required computer time. The solid elements are advantageous due to their capability to capture the through thickness effects of the deflection and stress since more than one element is placed in the thickness direction.

4.3.1 Shell 163 Model

4.3.1.1 *Problem Description*

A 12.5 cm x 12.5 cm shell target whose thickness is 0.6 cm and with the laminate properties is used for the target and a sphere with 1.8 cm diameter is used to represent the round end of the impactor tup, figure.3. The shell is constrained in the Z direction around the edges to simulate the clamp device and also constrained in all directions along two sides to prevent rigid body motion.

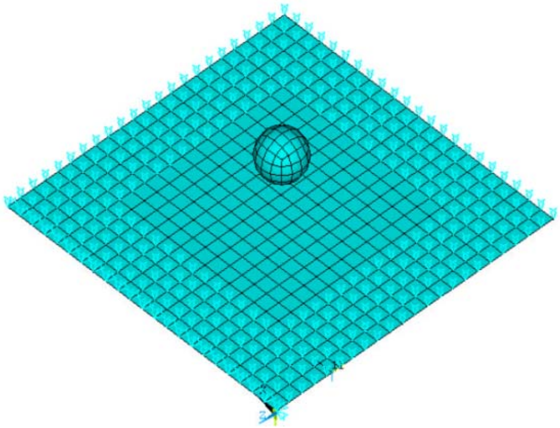

Fig.3 Meshed model of Square shell target

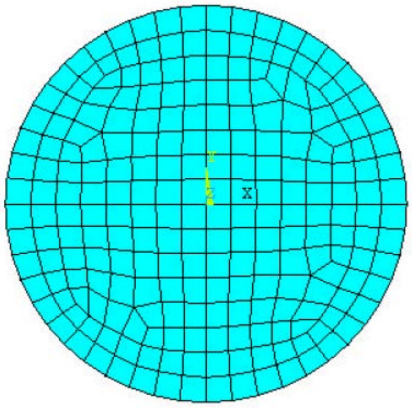

Fig.4 Cylindrical target with quadrilateral mesh with boundary conditions specified.

4.3.1.2 *Problems with Target Plate Shape*

The above model used very similar boundary conditions as the test jig in the drop weight tower. However, these boundary conditions gave some strange results in the clamped region due to the bending and membrane actions of the shells. In order to remove the possibility of these effects and also to lower the number of degrees of freedom, a new model is created. This model consisted of a 7.5 cm diameter flat cylindrical shell with the z displacements and out-of-plane rotations fixed around the outer edge.

4.3.1.3 *Problems with Target Plate Mesh*

A uniform mesh is often difficult with a cylindrical plate. If a triangular mesh is used, singularities are created at the center of the circle where the elements internal angles are very small. ANSYS also does not recommend the triangular mesh option with shell 163 elements therefore the quadrilateral mesh is used for all models. With the quadrilateral mesh the center elements have a slightly different shape than the elements on the outer rings of the circle; however, no irregularities are noticed with this mesh, figure.4.shown above

4.3.1.4 *Results and Discussions*

In order to verify the accuracy of the model, the impact force and the total energy was compared with the experimental results. Both were within 15% of the experimental results. Figure.5 shows the impact force predicted along with the force history from the experimental results with an impact velocity of 2 m/s.

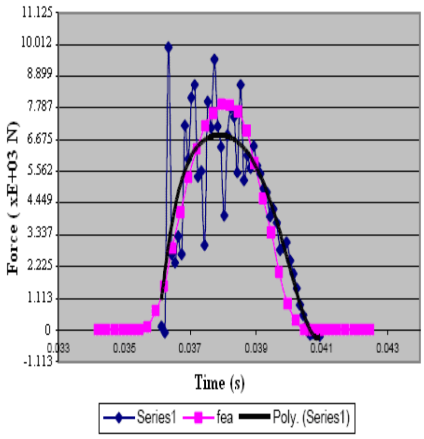

Fig.5 Force history and prdicted FEA

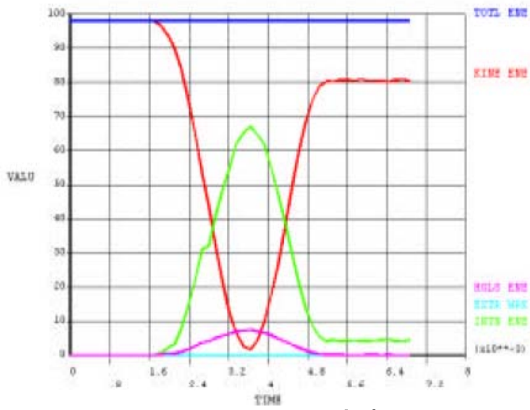

Fig .6 Energy- History Graphs for Impactor Velocity of 2 m/s shell 163 element.

In dynamic finite element analysis, the hourglass energy is a large concern. A representative energy plot from ANSYS/ LS-DYNA is shown in Figure 6.

4.3.2 Solid 164 Model

4.3.2.1 *Problem Description*

A 7.5 cm diameter solid cylinder is used for the composite target and a 1.75 cm diameter sphere is used for the steel impactor, figure.7. The solid elements are constrained with all translations fixed around the edges to simulate the clamped boundary conditions in the drop tower tests. The spherical solid is given an initial velocity (2 m/sec) in the direction of the target as in the other models. The material properties used for this model are shown in the tables listed below table 4.2

Table 4.2 Material Properties used for Composite Laminates.

Distance from target (mm)	0.254	0.0254	0.00254	0.000254	0.0000254
Deflection (μm)	-0.11684	-0.4826	-0.03175	-0.03048	-0.03048
Stress Intensity (N/m2)	21.7	9.1	5.95	5.775	5.705

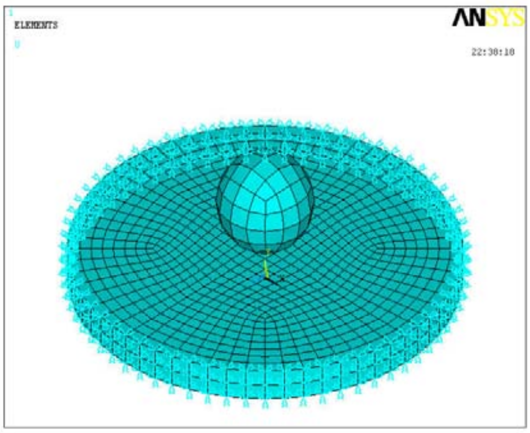

Fig.7 Finite Element Model of the Impactor

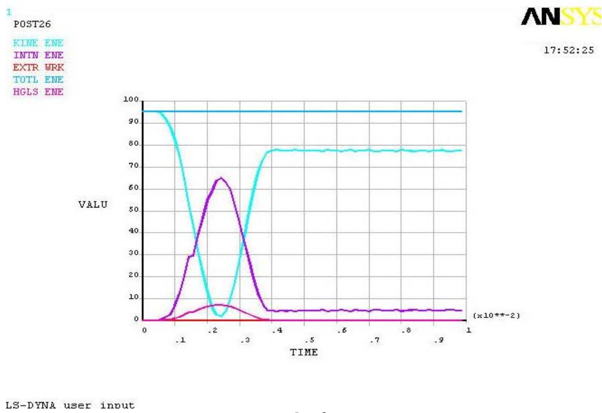

Fig.8 Energy- History Graphs for Impactor Velocity of 2 m/s using Solid 164 element

4.4.1 Drop Tower Tests Simulations :Test 1 – Vinylester Resin

4.4.1.1 *Description*

For this drop tower test, the impact mass is 5.5 kg and the impactor velocity just before impact is 3.81 m/s. In the FEA model, the density of the sphere is adjusted to achieve the same mass as the impactor. The target is modeled with solid 164 elements with a thickness of 6 mm. A linear orthotropic material model is used for the composite laminate and linear isotropic properties for the steel impactor. These properties are listed in table 4.3

Table 4.3 Material properties used for composite laminate.

Distance from target (mm)	0.254	0.0254	0.00254	0.000254	0.0000254
Deflection (μm)	-0.11684	-0.4826	-0.03175	-0.03048	-0.03048
Stress Intensity (N/m2)	21.7	9.1	5.95	5.775	5.705

These material properties are determined with the aid of the shareware program *The Laminator* [6]. The individual material properties for each layer are given in Table 4.4,4.5,4.6. These layer properties are inputted into the program along with the layer thickness and stacking sequence to determine the apparent material properties.

Table 4.4 Material properties for each layer.

	E_1	E_2	G_{12}	V_{12}
Roving (E-glass)	1.00×10^7	2.00×10^6	2.00×10^6	0.2
Mat material (isotropic)	1.00×10^6	1.00×10^6	4.00×10^5	0.28
Matrix (vinylester)	5.00×10^5	5.00×10^5	1.00×10^5	0.3

Table 4.5 Lay-up of Isopolyester laminate.

Layer	**Thickness (mm)**
317.5 mm Nexus	0
311.15 mm – 2 oz. OC	0.419
Resin	0.482
90 – 113 yield roving	0.787
Resin	0.482
311.15 mm – 2 oz. OC	0.419
Resin	0.482
90 – 113 yield roving	0.406
Resin	0.482
311.15 mm – 2 oz. OC	0.419
342.9 mm Nexus	0
Total	4.378

4.4.1.2 *Results and Discussions*

The impact force is first compared to the experimental data. From a curve fit of the experimental acceleration data, the maximum impact force is found to be approximately 7116.8 N. Five samples are tested with the same impact velocity and the maximum impact force is almost identical in all samples. Figure.9 shows a comparison between the experimental data and the predicted impact force verses time. The higher predicted impact force can be attributed to the model not including the energy release due to the damage (delamination and matrix cracking).

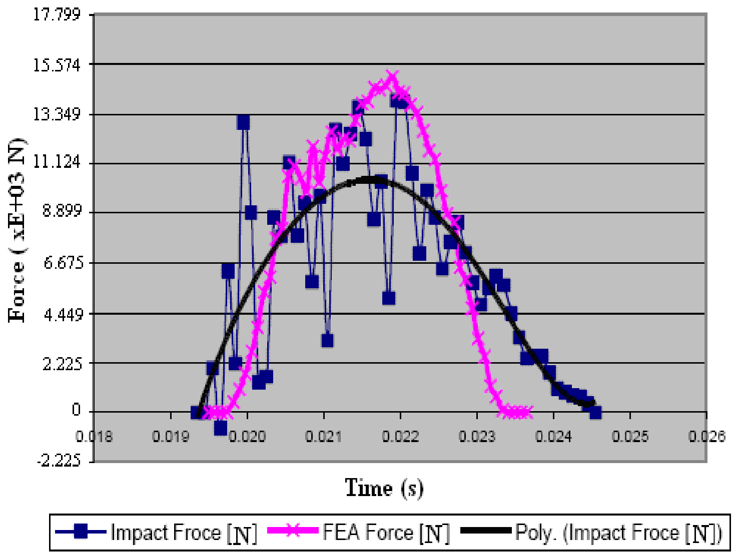

Fig.9 Force history and FEA force for an impactor velocity of 3.81 m/s.

4.4.2 Test 2 – Vinylester Resin

4.4.2.1 *Description*

The second test consisted of the same material with an impactor velocity of 2 m/s. Since the impact velocity is much lower, the impact force is also reduced. Figure.10 shows the predicted and actual force data for this test.

Table 4.6 Lay-up of vinylester laminate.

Layer	Thickness(mm)
342.9 mm Nexus	0
311.15 mm – 2 oz. OC	0.381
Resin	0.482
84 – 113 yield roving	0.787
Resin	0.482
311.15 mm – 2 oz. OC	0.381
Resin	0.482
42 – 113 yield roving	0.406
Resin	0.482
311.15 mm – 2 oz. OC	0.381
Resin	0.482
84 – 113 yield roving	0.787
Resin	0.482
311.15 mm – 2 oz. OC	0.381
342.9 mm Nexus	0
Total	6.396

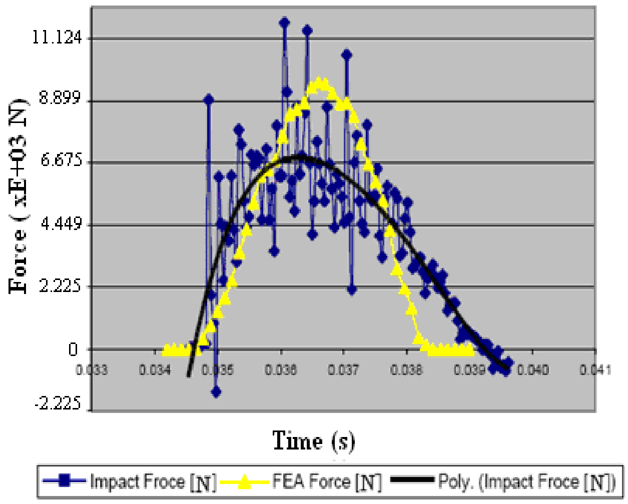

Fig.10 Force history and FEA force for impactor velocity of 2 m/s.

4.4.2.2 *Results and Discussions*

The predicted time span that the impactor is in contact with the target did not change with different impactor speed in the ANSYS models. The contact times used in the tests are 5.3 ms. for the first test (3.81. m/s) and 4.4 ms. for the second test (2 m/s). The FEA model predicted a contact time of 3.7 ms. for all impactor speeds.

4.4.2.3 *Comparisons*

Three tests are performed with the vinylester resin and impact speeds of 2, 3, and 4 m/s respectively. These three analyses are compared in order to examine the effect of velocity on the specimens; all other parameters are kept the same. Table 4.7 is a summary of these three tests.

Table 4.7 Comparison between test with impact speeds of 2, 3, and 4 m/

Impact Velocity (m/s)	**Predicted maximum in-plane stress on top (N/m^2)**	**Predicted maximum in-plane stress on bottom (N/m^2)**	**Backside crack length (m)**
1	300.3 $x10^6$	228.1 $x10^6$	0.0152
2	477.4 $x10^6$	292.6 $x10^6$	0.0533
3	651.7 $x10^6$	312.8 $x10^6$	0.0610

Based on comparisons of the predicted stress patterns on the bottom of the FEA models and the crack length, the cracks extend to the (175 – 245) $x10^6$ N/m^2 range on all three of these models. Further work could determine a stress range that initiates and continue matrix cracking and de-bonding.

4.4.2.4 *Stress and Strain*

The maximum predicted stress in the z direction for these three impactor speeds is shown in Figure.11

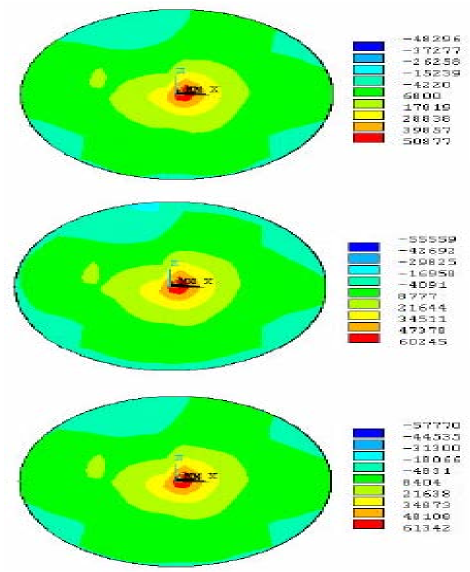

Fig.11 Maximum in-plane stress for three different impactor speeds
(*viz.* 2, 3, and 4 m/s).

As before, the predicted stress jumped quickly from the 2 to the 3 m/s case then leveled off as the speed went to 4 m/s. The stress contour patterns were almost identical in each case with only the values of the stress regions changing. The strain in the X direction (maximum in-plane) also showed nearly identical contour patterns for all three cases. These strains are shown in Figure.12.

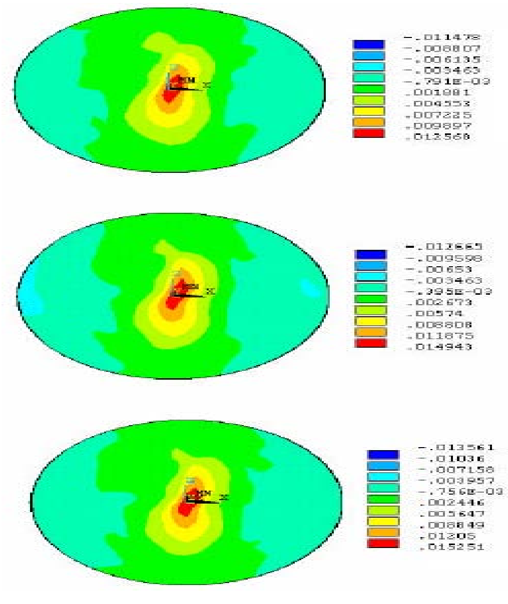

Fig .12 Maximum in-plane strains for three different impactor speeds
(*viz.* 2, 3, and 4 m/s)

The "wavy" pattern in the high strain area is very interesting. This pattern is also seen in the external backside crack on some of the test samples. This is most likely due to the anisotropic nature of the laminate.

5. Conculsion

The purpose of this research is to examine the impact response of fiberglass reinforced laminates. Several drop weight tower impact analyses are performed on 12.5cm x 12.5 cm FRP plates with resins including vinylester and iso polyester. These test simulated high mass (5.5 kg) and low velocity (2-4 m/s) impact. ANSYS/ LS-DYNA non-linear finite element analysis program is used to predict the stresses in the laminates during the impact. These stresses are then compared to the damage from the drop tests. The critical stress range for impact damage is in the range of (175-245) x10^6 N/m^2, which is very close to the static ultimate strength of the Duraspan laminates.

References

[1] R.E. McCarthy, "*Finite Element Analysis of a bird-Resistant Monolithic Stretched Acrylic Canopy Design*
[2] *for the F-16A Aircraft*." AIAA Aircraft Systems and technology Mtg. Dayton, Ohio, Aug. 11-13 1981.
[3] Hirschbein, M.S., "*Bird Impact Analysis Package for Turbine Fan Blades*." 23rd Structures,
[4] Structural Dynamics, and Materials Conference, New Orleans, LA, May 10-12 1992.
[5] LS-DYNA, Livermore Software Technology Corporation, 7374 Las Positas Rd., Livermore, CA 94550.
[6] Chang, F.K., and K. Y. Chang, "*A Progressive Damage Model for Laminated Composites Containing*
[7] *Stress Concentrations.*" J. Composite Mater. 21 (1997): pp. 834-855.
[8] Ansys Theory Manual, "*Elements Reference"*.
[9] *The Laminator* – Classical Analysis of Composite Laminates, http://www.thelaminator.net
[10] Ansys Theory Manual, "*Elements Reference"*.
[11] Cantwell, W.J., P. T. Curtis, and J. Morton, "*Low Velocity Impact Damage Tolerance in CFRP*
[12] *Laminates Containing Woven and Non-Woven Layer.*" Composites. 14 (1983): pp. 301-305.
[13] Lui, D. "*Impact Induced Delamination – A View of Bending Stiffness Mismatching."*
[14] Journal of Composite Materials. . 22
[15] Hou, J. P., N. Petrinic, and C. Ruiz, "*A Delamination Criterion for Laminated Composites Under*
[16] *Low- Velocity Impact.*" Composite Science and Technology. 61 (2001): pp. 2069-2074.
[17] Cantwell, W.J., and J. Morton, "*The Impact Resistance of Composite Materials – a Review*"
[18] Composites. 22 (1991): pp. 347-362.