MecSol2013
International Symposium on Solid Mechanics
April 18-19, 2013
Porto Alegre, Rio Grande do Sul, Brazil

# NUMERICAL AND EXPERIMENTAL ANALYSES OF LOW VELOCITY IMPACT ON THIN COMPOSITE

**Marcelo L. Ribeiro**
Department of Aeronautical Engineering
Engineering Scholl of São Carlos – University of São Paulo
malribei@usp.br

**Ricardo A. Angélico**
Department of Aeronautical Engineering
Engineering Scholl of São Carlos – University of São Paulo
ricardoangelico@gmail.com

**Ricardo de Medeiros**
Department of Aeronautical Engineering
Engineering Scholl of São Carlos – University of São Paulo
medeiros@sc.usp.br

**Gerson Marinucci**
Department of Aeronautical Engineering
Engineering Scholl of São Carlos – University of São Paulo
gerson@ctmsp.mar.mil.br

**Volnei Tita**
Department of Aeronautical Engineering
Engineering Scholl of São Carlos – University of São Paulo
voltita@sc.usp.br

**Summary.** *The dynamic behavior of composite laminates is very complex because there are many concurrent phenomena during composite laminate failure under impact load. Fiber breakage, delaminations, matrix cracking, plastic deformations due to contact and large displacements are some effects which should be considered when a structure made from composite material is impacted by a foreign object. Thus, the mechanical material behavior is simulated using a phenomenological model that considers five failure modes: two for fiber-failure (FF) and three for inter-fiber-failure (IFF). In FF modes, the lamina failures under longitudinal tension or compression. In IFF modes, the lamina failures under transverse tension (Mode A), or transverse compression (Mode B or C). Each failure mode has a failure criteria and an associated degradation function that decreases the engineering material properties, turning the process of analysis iterative. The material model is implemented in a sub-routine written in Fortran (UMAT and VUMAT) and used together the finite element package Abaqus for implicit and explicit numerical solution. First, the material parameters are identified and calibrated using case studies based on 3-point bending problem. Thus, finite element analyses are performed for a set of parameters through a Matlab program that controls the parameters variability and generate the results of interest which are then evaluated. After calibration procedure, the material model is used to predict the response of thin disks under impact loads. Finally, the numerical simulations are compared to experimental results, and limitations, as well as potential of material model implemented are discussed.*

**Keywords:** Composite Laminates, Material Model, Low Velocity Impact, Finite Element Analysis.

## 1 INTRODUCTION

During the last years, criteria of aircraft projects have been more and more rigorous for component development to absorb impact energy. Research on the development of structural components with high crashworthiness has been carried by the aeronautical industries. The project concept for structural components with high crashworthiness depends on the crash resistance concept described by Kindervater and Georgi [1]. The crash resistance concept is based on the energy absorption capacity and structural integrity. However, the dynamic behavior of composite laminates is very complex, because there are many concurrent phenomena during composite laminate failure under impact load. Fiber breakage, delaminations, matrix cracking, plastic deformations due to the contact and large displacements are some effects, which should be considered when a structure made from composite material is impacted by a foreign object. Therefore, it is very common to find current research works about this issue at the literature, for example: Sari et al. [2], Ribeiro et al. [3], Tita et al. [4].

In this work, each failure mode has a failure criterion and an associated degradation function, which decreases the engineering material properties, turning the process of analysis iterative (Progressive Failure Analyses – PFA). Thus, the material model proposal was implemented as Fortran sub-routines (UMAT and VUMAT) linked to finite element package Abaqus$^{TM}$ for implicit and explicit numerical solution. First, the material parameters were identified and calibrated using 3-point bending tests and implicit numerical simulations with UMAT sub-routine. Therefore, finite element analyses were performed for a set of parameters. A Matlab program was developed in order to control the parameters variability and the post-processing of the results.. After calibration procedure, the material model proposal is used to predict the response of thin disks under impact loads using explicit numerical simulations with VUMAT sub-routines. Finally, the numerical results are compared to experimental data and it is discussed the advantages and limitation of the material model proposal.

## 2 MATERIAL MODEL

The material model proposal in this work is a new combination of different failure criteria and degradation laws adopted by other researchers. Also there is a new method for identification and calibration of material model parameters.

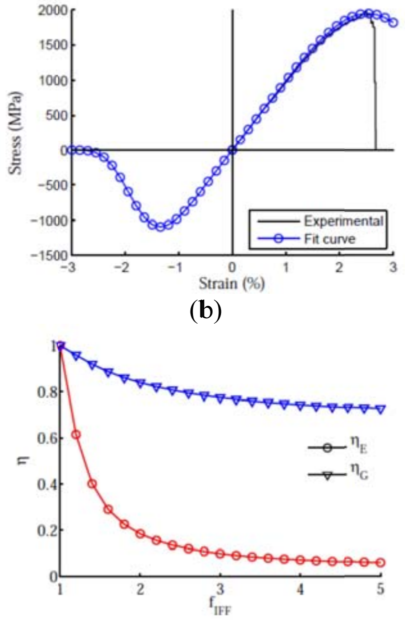
### 2.1 Mathematical Formulation

The stress state (in a generic point of the lamina) is evaluated by the constitutive equations, which are written for plane stress state. After that, the failure criteria use the value of the actuating stress and allowable in order to determine the occurrence of a failure or not. In fact, the failure occurs when the criterion provide a value greater than unit. The material model criteria for fiber failure (FF) are based on Hashin's Theory [5]. The material model criteria for inter-fiber failure (IFF)is based on Puck's Theory [6] [7]. Fiber failure criteria are evaluated for tensile loads (FF-T) and for compressive loads (FF-C). Inter-fiber failure criteria are evaluated for three failure modes: Mode-A (IFF-A), Mode-B (IFF-B) and Mode-C (IFF-C) (Fig.1). It is important to mention that the Puck's work nomenclature is used in this paper.

Once the element material point fails ($f_{FF} > 1$ or $f_{IFF} > 1$), the material properties are decreased according to the failure mode. if the FF is verified, then the properties associated to direction 2 are considered null and Young's modulus in direction 1 is decreased by the parameter $\omega$, considering the Matzenmiller's Theory ( [8]). If the IFF is verified, the mechanical material properties associated to direction 2 (matrix direction) are decreased by the parameter $\eta$. Moreover, Figure 1 resumes the failure criteria and degradation laws for the material model. The equations presented in Fig. 1(a). For convenience were hold the same Puck and Sührmann's notation [7].

| | | | |
|---|---|---|---|
| | **FAILURE CRITERIA** | | |
| **FF** | **FF-T**: Fiber failure under tensile stress ($\sigma_{11} > 0$) <br> $f_{FF} = \left(\frac{\sigma_{11}}{F_{1T}}\right)^2 + \left(\frac{\sigma_{12}}{F_{12}}\right)^2$ <br> **FF-C**: Fiber failure under compressive stress ($\sigma_{11} < 0$) <br> $f_{FF} = \left(\frac{\sigma_{11}}{F_{1C}}\right)^2$ | | |
| **IFF** | **IFF-A**: Inter-fiber failure - Mode A <br> $f_{IFF} = \sqrt{\left[\left(\frac{1}{R^t_\perp} - \frac{p^t_{\perp\parallel}}{R_{\perp\parallel}}\right) \cdot \sigma_{22}\right]^2) + \left(\frac{\sigma_{12}}{R_{\perp\parallel}}\right)^2} + \frac{p^t_{\perp\parallel}}{R_{\perp\parallel}}\sigma_{22}$ <br> **IFF-B**: Inter-fiber failure - Mode B ($\sigma_{22} < 0$ e $\lvert\sigma_{22}/\sigma_{12}\rvert \leq \lvert R^A_{\perp\perp}/\sigma_{12,c}\rvert$) <br> $f_{IFF} = \sqrt{\left(\frac{\sigma_{12}}{R_{\perp\parallel}}\right)^2 + \left(\frac{p^c_{\perp\parallel}}{R_{\perp\parallel}}\sigma_{22}\right)^2} + \frac{p^c_{\perp\parallel}}{R_{\perp\parallel}}\sigma_{22}$ <br> **IFF-C**: Inter-fiber failure - Mode C ($\sigma_{22} < 0$ e $\lvert\sigma_{22}/\sigma_{12}\rvert \geq \lvert R^A_{\perp\perp}/\sigma_{12,c}\rvert$) <br> $f_{IFF} = \left[\left(\frac{\sigma_{12}}{2(1+p^c_{\perp\perp})R_{\perp\parallel}}\right)^2 + \left(\frac{\sigma_{22}}{R^c_\perp}\right)^2\right]\frac{R^c_\perp}{-\sigma_{22}}$ | | |
| | **DEGRADATION LAWS** | | |
| | **Original Properties** | **Degraded properties** | |
| **FF** | $E_1$ <br> $E_2$ <br> $G_{12}$ <br> $\nu_{12}$ | $\overline{E_1} = (1-\omega)E_1$ <br> $\overline{E_2} = 0$ <br> $\overline{G_{12}} = 0$ <br> $\overline{\nu_{12}} = 0$ | $\omega = 1 - \exp\left[-\frac{1}{me}\left(\frac{\epsilon_{11}}{\epsilon_f}\right)^m\right]$ |
| **IFF** | $E_1$ <br> $E_2$ <br> $G_{12}$ <br> $\nu_{12}$ | $\overline{E_1} = E_1$ <br> $\overline{E_2} = \eta_E E_2$ <br> $\overline{G_{12}} = \eta_G G_{12}$ <br> $\overline{\nu_{12}} = \nu_{12}$ | $\eta = \frac{1-\eta_r}{1+c(f_{IFF}-1)^\xi} + \eta_r$ |

(a)




(b)

(c)

**Figure 1: (a) Failure model summary; (b) Tensile and compression Matzenmiller's parameters adjustment; (c) inter-fiber failure degradation parameters.**

Figure 1(b) shows the Matzenmiller fiber failure degradation parameter ω adjustment. For the $[0]_{10}$ uniaxial tensile test, the parameters *e=9.5* and *m=3.8* showed a good correlation between the experimental test and simulation. Due to a lack of experimental test results for compression, the parameters must lead to ultimate compression load of 930 MPa [9]. Thus, the paremeters value of *e=16.0* and *m=4.7* have shown to perform the best fit for the compression part of the curve. Figure 1(c) shows how the IFF parameter,η, evolves considering normal and shear stresses.

## 3 FINITE ELEMENT MODELS

### 3.1 Three-point bending finite element model

Four node homogeneous fully integrated linear elements (Abaqus$^{TM}$ S4 shell elements) were used to simulate the three point bending test. Also, for each layer, it was used three integration point throught the thickness.

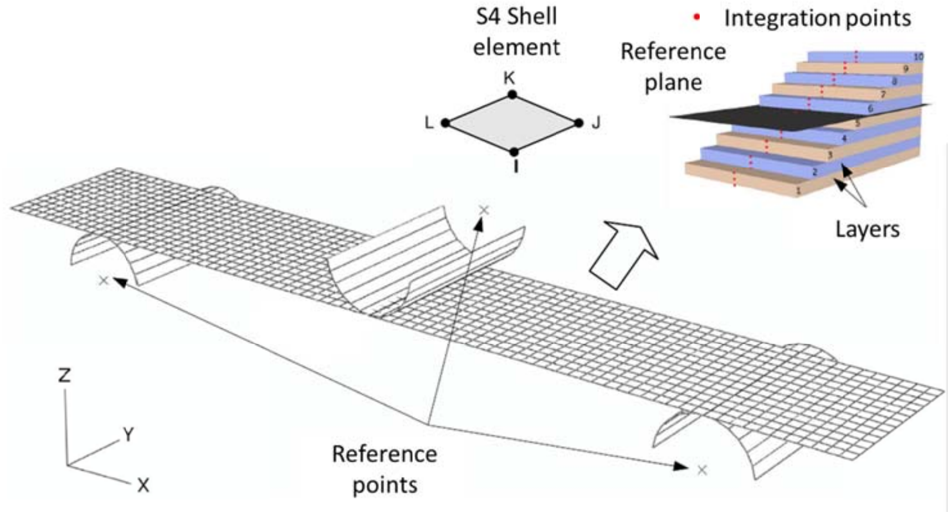



**Figure 2: Finite element model mesh, boundary conditions and element integration points**

For boundary conditions, all the degress of freedom of the two external rigidy surfaces was restricted at the reference points and, a prescribed displacement of 8 mm in z direction was applied at the reference point of the middle rigidy surface. Hard contact law is used as to simulate the normal interactions between the laminate shell and rigidy surfaces of the 3-point bending device. The applied step size consists on an initial increment of 1% and a maximum of 3% of the total prescribed displacement.

It is very important to mention that this model was used to calibrate the parameters of the material model shown by the Figure 1(a), mainly the parameters (*m* and *e*) related to the degradation of the properties due to FF (fiber failure). Table 1 shows the material eleastic properties and the strength values

**Table 1:** Elastic properties and strength values [4].

| **Elastic properties** | | **Strength values** | |
|---|---|---|---|
| $E_{11}$ | 127 GPa | $X_T$ | 1400 MPa |
| $E_{22}= E_{33}$ | 10 GPa | $X_C$ | 930 MPa |
| $G_{12}= G_{13}$ | 5.4 GPa | $Y_T$ | 47 MPa |
| $G_{23}$ | 3.05 GPa | $Y_C$ | 130 MPa |
| $\upsilon_{12}= \upsilon_{13}$ | 0.34 | $Z_T$ | 62.3 MPa |
| $\upsilon_{23}$ | 0.306 | $S_{12}= S_{13}$ | 53 MPa |
| | | $S_{23}$ | 89 MPa |

### 3.2 Impact finite element model

It was modeled the whole coupon fixture in order to have more realistic boundary conditions (Figure 3).

For the steel fixture discs, it was used 8 nodes linear hexahedron elements and all the displacements degrees of freedom of the lower disc were restricted. And, 8 nodes linear brick elements for small discs were used to simulate the screw of the device. A torque of 50 Nm was applied on the screws during the experimental impact tests. Thus, those small disks had $U_x$ and $U_y$ degrees of freedom restricted and, a small displacement of minus 0.157mm was applied for $U_z$ to simulate the screw compression forces on the fixture steel disks (due to the torque).

The normal interaction between the composite shell and fixtures was simulated using a hard contact law and, penalty contact algorithm with 0.1 friction coefficient was used to simulate the tangential interaction. The same interaction properties were applied for the interactions between the impactor and composite shell.

The mass of 1.2 Kg, which corresponds to the total of impact mass, was simulated applying this value of mass at the impactor reference point, as well as initial velocity of 3.130m/s for the impactor. The impactor was modeled with R3D4 4 node discrete rigid bilinear quadrilateral element.

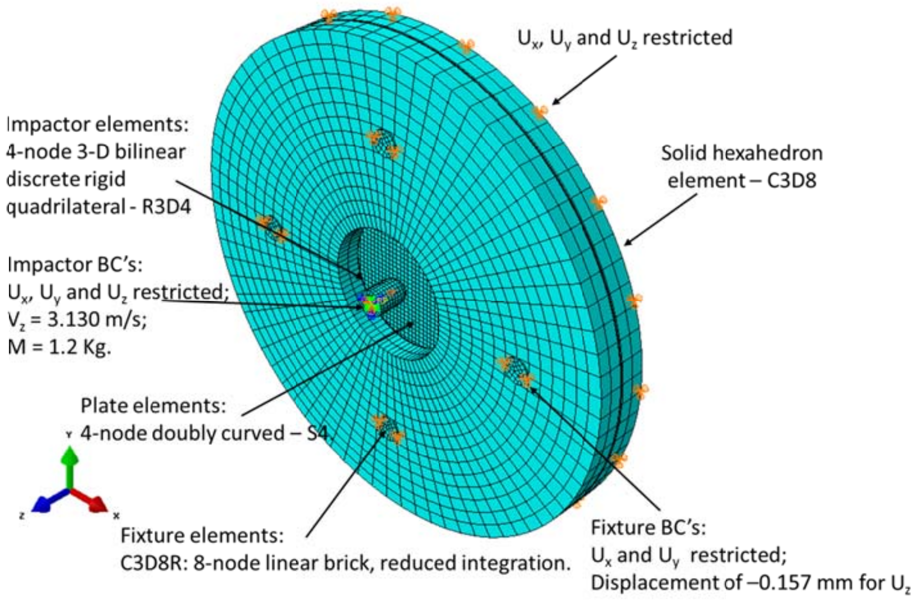
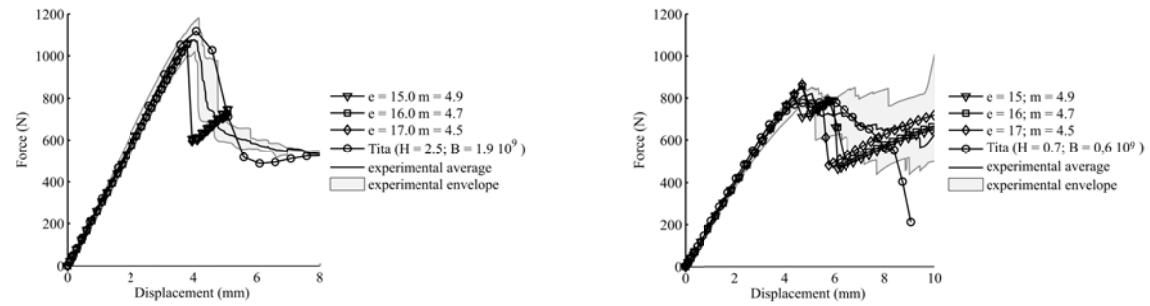



**Figure 3: Finite element model: mesh, boundary conditions and initial condition**

## 4 RESULTS

### 4.1 Results for three-point bending

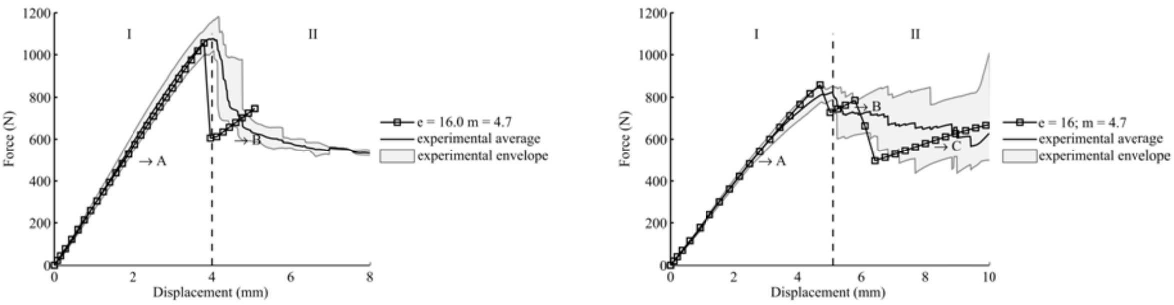
Three-point bending tests of $[0]_{10}$ and $[0/90/0/90/0]_s$ laminates were used not only to evaluate the material model proposal (implemented via UMAT), but also to calibrate the material model parameters for degradation of material properties due to FF. Figure 4 (a) shows the parameters sensibility investigation for unidirecitonal $[0^o]_{10}$ laminate and, Figure 4 (b) for cross-ply $[0/90/0/90/0]_s$ laminates. The shaded area in the figures is the experimental test envelop. Also, it possible to verify the numerical results provided by another material model [4].




(a) (b)

**Figure 4: Parameters calibration: (a) unidirectional 0° - $[0^o]_{10}$; (b) cross-ply laminate - $[0/90/0/90/0]_s$.**

In the Figure 5, it is shown a comparison between numerical results provided by the material model proposal, considering *e equal 16.0* and *m equal 4.7.* It is important to highlight that these values were used for the impact numerical analyses, after to perform the calibration process as discussed earlier.




(a) (b)

**Figure 5: Three-point bending tests and numerical simulations: (a) $[0^o]_{10}$; (b) $[0/90/0/90/0]_s$.**

## 4.2 Results for impact analyses

Figure 6 shows force-time graphics for composite plates with stacking sequence $[0/90/0/90/0]_s$, under 5.91 J impact energy. For the experimental data, there is a region with oscillations of high frequency (from 0 to 1.2 ms). From 1.8 ms, there are some small oscillations, which show damage process. Due to the stacking sequence $[0/90/0/90/0]_s$, the main failure mechanisms are the matrix rupture and delaminations, which reduces the global stiffness of the structure. Therefore, there is no abrupt drop of the impact force value after the progressive damage process initiation. The finite element simulations via ABAQUS™ using the damage model proposal (implemented via VUMAT) were capable to capture the initial oscillations of the impact (from 0 till 0.5 ms). For the other side, the numerical simulations via ABAQUS™ with no damage were also capable to capture those initial oscillations. However, the maximum peak force is closer to the experimental maximum value when is used the proposed damage model. The duration of the impact event is around 4.1 ms for the experimental, 4.2 ms for the damage model and 3.9 ms for the undamaged model (Table 2).

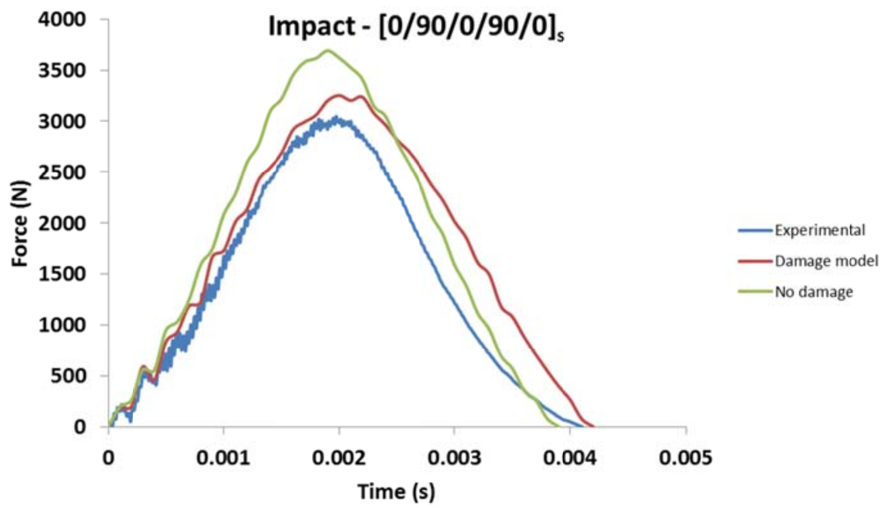



**Figure 6: Force vs. Time impact experimental and numerical results for [0/90/0/90/0]s.**

**Table 2: Results for [0/90/0/90/0]s.**

| | Max Force (N) | Error %* | Time (ms) | Error %* |
|---|---|---|---|---|
| Experimental | 3048 | - | 4.1 | - |
| Damage model | 3252 | 6.7 | 4.2 | 2.4 |
| No damage | 3691 | 21.1 | 3.9 | 4.9 |

$$* \, Error = \left| (Experimental - Simulations) / Experimental \right|$$

Figure 7 shows force-time graphics for composite plates with stacking sequence $[+45/-45/+45/0/90]_s$, under 5.91 J impact energy. The quasi-isotropic laminate behaves rather similar to cross-ply laminates. For the experimental data, there is a region with oscillations of high frequency (from 0 to 1.2 ms). From 1.8 ms, there are some small oscillations, which show damage process. As for the previous laminate, the main failure mechanisms are the matrix rupture and delaminations, which reduces the global stiffness of the structure. Therefore, there is no abrupt drop of the impact force value after the progressive damage process initiation. The finite element simulations using the proposed damage model were capable to capture the initial oscillations of the impact (from 0 till 0.2 ms). For the other side, the numerical simulations with no damage were also capable to capture those initial oscillations. For this laminate, the maximum peak force provided by the proposed material model as well as estimated by the model with no damage are not so closer to the experimental maximum value. The duration of the impact event is around 4.0 ms for the experimental, 3.8 ms for the damage model and 3.8 ms for the undamaged model( Table 3).

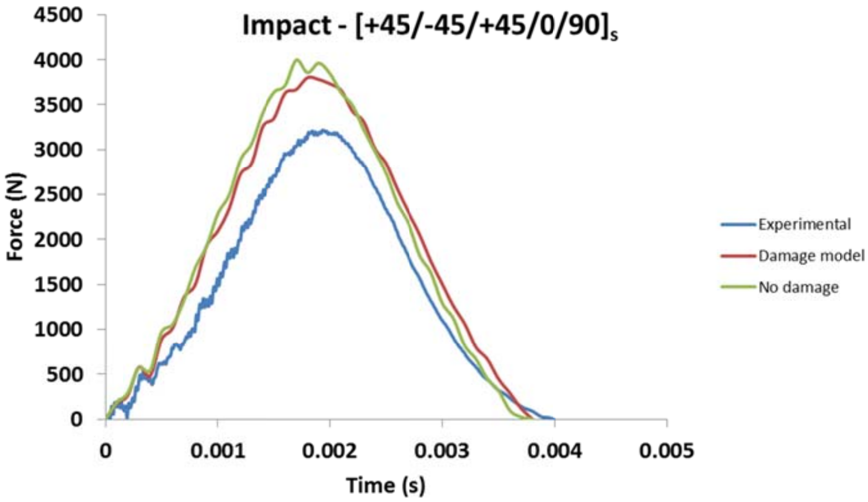



**Figure 7: Force vs. Time impact experimental and numerical results for [+45/-45/+45/0/90]$_s$ laminate.**

**Table 3: Results for [+45/-45/+45/0/90]s.**

| | Max Force (N) | Error %* | Time (ms) | Error %* |
|---|---|---|---|---|
| Experimental | 3218 | - | 4.0 | - |
| Damage model | 3800 | 18.1 | 3.8 | 4.9 |
| No damage | 3995 | 24.1 | 3.8 | 4.9 |

$* \, Error = \left|(Experimental - Simulations)/Experimental\right|$

Finally, Figure 8 shows force-time graphics for composite plates with stacking sequence $[0^o]_{10}$ under 5.91 J impact energy. The experimental data shows high frequency oscillations in the beginning of the impact event, as well as some lower frequency oscillations (from 0 ms till 0.5 ms), which the finite element model was not capable to simulate.

Despite the others lay-up showed in this work, the $0^o$ lay-up shows an abrupt drop on the force around 1.8 ms. For this laminate, mechanisms failure are more severe than for the previous cases. For example, fiber breakage absorb much more energy than matrix damage [10], thus the force peak is lower and the impact duration is bigger for this laminate. The finite element model results are presented in Figure 8 and the Table 4 shows the comparison between the experimental and numerical results.

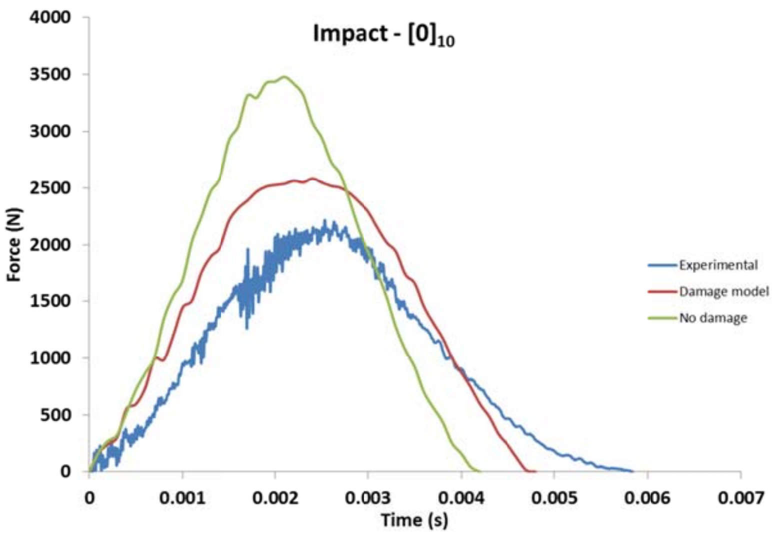



**Figure 8: Force vs. Time impact experimental and numerical results for $[0^o]_{10}$ laminate.**

**Table 4: Results for $[0^o]_{10}$.**

| | Max Force (N) | Error %* | Time (ms) | Error %* |
|---|---|---|---|---|
| Experimental | 2217 | - | 5.8 | - |
| Damage model | 2583 | 16.5 | 4.8 | 17.8 |
| No damage | 3478 | 56.9 | 4.2 | 28.1 |

$* \, Error = \left|(Experimental - Simulations)/Experimental\right|$

Based on the results, it is possible to observe that regarding the impact event on cross-ply laminate, the proposed material model was able to capture the small force oscillations in the beginning of the impact and to predict with a reasonable accuracy the duration of the event. For quasi-isotropic laminates, the proposed material model showed not so good performance to capture the maximum force (difference of 18.1% between experimental data and numerical s results). However, the total time error is still reasonable (4.9%). For unidirectional laminates, the model performs the worst predictions for the overall impact time with error around 17.8% and for maximum force with error of 16.5%.

## 5 CONCLUSIONS

The material model proposal is a new model because consists on a combination of different failure criteria and damage evolution law. The parameters associated to the degradation of the material properties due to fiber failure were deeply investigated. It can be concluded that the investigation of these parameters is fundamental to understand how it affects the structural behavior, as well as the relevance of one parameter in relation to the other. Also, it can be concluded that the calibration of the parameters related to the composite material model needs to be consider for different stacking sequences.

Regardless the difficulties related to three point bending tests (matrix crushing under the load applier), the material model performed good numerical predictions for this quasi-static case. However, the simulation for an impact event represents a bigger challenge for a material model. As it can be observed, the proposed material model could predict very well the behavior of cross-ply laminates, determining with accuracy the maximum force peak and the impact duration. On the other hand, the model performance for the others lay-up was not so good with error bigger than 16% for force peak. This can be partially explained through the calibration process, which is better performed for cross-ply laminate. Another reason for the poor model accuracy is due to lack of delamination criteria. The simulation of this phenomenon is essential for laminates that presents mainly delaminations during impact events.

## 6 ACKNOWLEGMENTS

The authors are grateful for the support from CTM (Navy Technological Centre – Brazil), São Paulo Research Foundation (FAPESP process number: 2009/00544-5), National Council of Research (CNPq process number: 208137/2012-2), AFOSR and US-Army (Grant/Contract Number: FA9550-10-1-0011). The authors also would like to thank Prof. Reginaldo Teixeira Coelho (Engineering School of São Carlos – University of São Paulo) for the ABAQUS license.

## REFERENCES

[1] C. M. kindervater and H. Georgi, "Composite strength and energy absorption as an aspect of structural crash resistence," *Structural crashworthiness and failure*, pp. 189-235, 1993.

[2] Murat Sari, Ramazan Karakuzu, Mehmet Emin Deniz, and Bulent Murat Icten, "Residual failure

pressures and fatigue life of filament-wound composite pipes subjected to lateral impact," *Journal of Composite Materials*, vol. 46, no. 15, pp. 1787-1794, 2012.

[3] Marcelo Leite Ribeiro, Volnei Tita, and Dirk Vandepitte, "A new damage model for composite laminates," *Composite Structures*, vol. 94, no. 2, pp. 635-642, 2012.

[4] Volnei Tita, Jonas de Carvalho, and Dirk Vandepitte, "Failure analysis of low velocity impact on thin composite laminates: Experimental and numerical approaches," *Composite Structures*, vol. 83, no. 4, pp. 413-428, 2008.

[5] Z. Hashin, "Failure Criteria for Unidirectional Fiber Composites," *Journal of Applied Mechanics*, vol. 47, no. 2, pp. 329-334, 1980.

[6] A. Puck and H. Schurmann, "FAILURE ANALYSIS OF FRP LAMINATES BY MEANS OF PHYSICALLY BASED PHENOMENOLOGICAL MODELS," *Composites Science and Technology*, vol. 58, no. 7, pp. 1045-1067, 1998, 0266-3538.

[7] A. Puck and H. Schurmann, "Failure analysis of FRP laminates by means of physically based phenomenological models," *Composites Science and Technology*, vol. 62, no. 12-13, pp. 1633-1662, 2002.

[8] A. Matzenmiller, J. Lubliner, and R. L. Taylor, "A constitutive model for anisotropic damage in fiber-composites," *Mechanics of Materials*, vol. 20, no. 2, pp. 125-152, 1995.

[9] Volnei Tita, "Contribuição ao estudo de danos e falhas progressivas em estruturas de material compósito polimérico," Ph.D. dissertation 2003 (*in Portuguese*).

[10] Serge Abrate, "Modeling of impacts on composite structures," *Composite Structures*, vol. 51, no. 2, pp. 129-138, 2001.