EXPERIMENTAL AND FINITE ELEMENT ANALYSIS OF COMPOSITE LAMINATES ON FLEXURAL AND TRANSVERSE SHEAR LOADING

Volnei Tita, voltita@sc.usp.br
Department of Materials, Aeronautical and Automobilist Engineering - School Engineering of São Carlos - University of São Paulo
Av. Trabalhador São-carlense, 400 – São Carlos – SP – Brazil

Jonas de Carvalho, prjonas@sc.up.br
Department of Mechanical Engineering - School Engineering of São Carlos - University of São Paulo
Av. Trabalhador São-carlense, 400 – São Carlos – SP – Brazil

Dirk Vandepitte, Dirk.Vandepitte@mech.kuleuven.ac.be
Department of Mechanical Engineering - PMA Division - Katholieke Universiteit Leuven
Celestijnenlaan 300B - B-3001 Heverlee – Leuven – Belgium

Abstract. *During the last two decades, many researches about composite materials applied on aircraft structures have been developed, because these structures can have high stiffness and strength with low weight, which can guarantee the increase of the pay-load for airplanes without losing airworthiness. Many kinds of structures have been made using fiber reinforced plastic, because these materials can be designed, according to the loads. However, the mechanical behavior of composite laminates is very complex because there are many concurrently phenomena during composite laminate failure. Fiber breakage, delaminations, matrix cracking, plastic deformations at the polymer matrix and large displacements are some effects which should be considered when a structure made from composite material on flexural and transverse shear loading. Thus, an investigation of 3-point bending test for thin composite laminates of epoxy resin reinforced by carbon fiber is presented. The study shows the set-ups of equipments to make 3-point bending tests. Afterwards, the influence of stacking sequence was investigated using load-displacement graphics. Finally, Finite Element Analysis (FEA) were developed, using material models implemented by UMAT (User Material Subroutine) into software ABAQUS™, in order to simulate the failure mechanisms of the specimens during the 3-point bending tests. Numerical results show that the material model implemented can simulate the mechanical behavior of the composite laminate under flexural and transverse shear loads. Thus, the material model implemented can be used as a computational tool in order to help for developing of the aircraft composite structure.*

Keywords*: composite laminates, material model, 3-point bending test, finite element analysis*

1. INTRODUCTION

During the last two decades, many researches about composite materials applied on aircraft structures have been developed, because these structures can have high stiffness and strength with low weight, which can guarantee the increase of the pay-load for airplanes without losing airworthiness. Many kinds of structures have been made using fiber reinforced plastic, because these materials can be designed, according to the loads. However, the mechanical behavior of composite laminates is very complex because there are many concurrently phenomena during composite laminate failure. Fiber breakage, delaminations, matrix cracking, plastic deformations at the polymer matrix and large displacements are some effects which should be considered when a structure made from composite material on flexural and transverse shear loading. Therefore, it's very common to find research works about this issue at the literature, for example: Yamada and Sun (1978); Hashin (1979); Hashin (1980); Rowlands (1985); Chang and Chang (1987a, 1987b); Chang, Liu and Chang (1991); Chang and Lessard (1991); Shahid and Chang (1995); París (2001). Add to these works, a "World Wide Failure Exercise (WWFE)" was organized by Soden, Hinton e Kaddour (1998a, 1998b). In the WWFE, each research group applied failure theories in order to predict the strain and stress values of composite laminates under some load cases. The results and comments obtained by reseachers were published at Hinton, Kaddour and Soden (2002a, 2002b); Kaddour, Hinton and Soden (2004); Hinton, Kaddour and Soden (2004a, 2004b).

This work shows an investigation of 3-point bending test for thin composite laminates of epoxy resin reinforced by carbon fiber. Thus, Finite Element Analysis (FEA) were developed, using material models implemented by UMAT (User Material Subroutine) into software ABAQUS™ (Abaqus, 2002), in order to simulate the failure mechanisms of the specimens under the 3-point bending tests. Numerical results show that the material model implemented can simulate the mechanical behavior of the composite laminate under flexural and transverse shear loads. Afterwards, several issues are discussed:

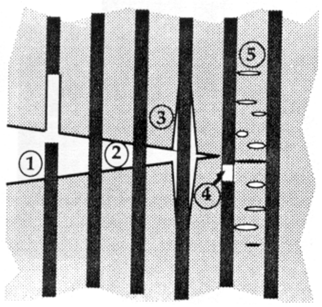
- Experimental results as force-displacement of the laminate, considering different stacking sequence;
- Material model implemented as UMAT into software ABAQUS™;
- Comparison between 3-point bending tests results and finite element model results, using the material model implemented by UMAT.

2. MECHANICAL BEHAVIOR OF COMPOSITE LAMINATES

Composite laminate structures were made from the stacking of plies, which contains a polymeric matrix reinforced by fibers. Therefore, composite laminate shows two failure modes:

1. Intra-ply failure mode: damages at fibers, polymeric matrix and/or interface between fibers and matrix(Fig.1(a));
2. Inter-ply failure mode: delaminations between plies (Fig.1(b)).

The intra-ply damage (Fig.1(a)) at fibers is shown by mechanism 4 that is the fiber rupture. However, the fiber failure mode depends on the type of loading, because, compression loads can induce micro-buckling, but, tensile loads can induce rupture of fibers. The intra-ply damage at the matrix depends on the ductility of the polymer, as well as on the in-service temperature. Thus, the polymeric matrix can present a fragile or a plastic behavior (mechanism 5). Other intra-ply failure mechanisms are shown by Fig. 1. The mechanism 1 is called “Pull-Out” and occurs when the interface between fiber and matrix is weak. So, the fiber is pulled out of the matrix after the debonding mechanism (mechanism 3) occurs. If the interface between fiber and matrix is strong, the fiber isn’t pulled out of the matrix and the mechanism 2 called “Fiber Bridging” occurs.



(a)


(b)

Figure 1. Composite failure mechanisms: (a) intra-ply (Anderson, 1995); (b) inter-ply failures (delamination)

The inter-ply failure called delamination (Fig.1(b)) occurs, in many cases, after intra-ply damages, i.e., the evolution of intra-ply damages propagates the delaminations, because the regions damaged at the ply propagates when the load increases and the cracks at two adjacent plies (with different orientation angle) join for creating a discrete failure between them. At that moment, the interlaminar shear increases strongly and the delamination process initiates. This failure mechanisms is very common to occur under flexural and transversal shear loading.

3. MATERIAL AND METHODS

American Society for Testing and Materials (ASTM) standards were followed to manufacture the specimens and to realize the experimental tests. It’s important to note that the fabrication of the specimens and the experimental tests were executed at Leuven Composites Processing Centre (LCPC) of Katholieke Universiteit Leuven (Belgium). Afterthat, a material model was proposed only to simulate the intra-ply damage mechanisms, using laminated shell elements and plane stress state. That material model was implemented as a UMAT subroutine into software ABAQUS^{TM} (Abaqus, 2002).

3.1. Laminate manufacturing

American Society for Testing and Materials (ASTM) standards were followed to manufacture the specimens and to realize the experimental tests. The prepreg M10 (epoxy resin reinforced by carbon fiber unidirectional) from Hexcel Hexcel^{TM} was used for manufacturing of the composite plates. It’s important to note that the fabrication of the specimens and the experimental tests were executed at Leuven Composites Processing Centre (LCPC) of Katholieke Universiteit Leuven (Belgium).

After the hand-lay-up process used to stack the plies, the composite plate was put in the auto-clave with vacuum system set to -0.8 Bar (-0.08 MPa). According to Hexcel^{TM}, the complete cure cycle for M10 occurs when this material is processed at 120^{o}C, under a pressure with range from 0.3 to 5.0 Bar during 60 minutes (Fig.2). After the curing process, the composite plates were cut on rectangular shapes, using diamond saw in order to guarantee the tolerances specified by standards.

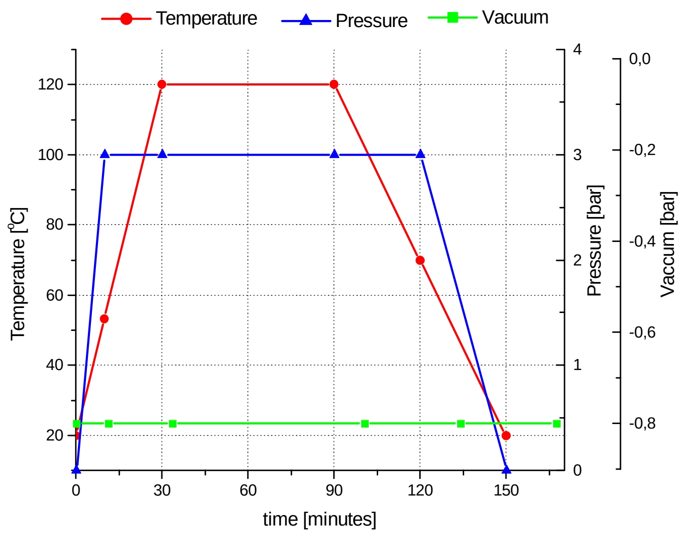


Figure 2. Cure cycle

3.2. Three-point bending tests

The 3-point bending tests were realized in order to evaluate the capability of material model proposed to predict the mechanical behavior of composite laminate. The experimental tests were developed following the specifications by the ASTM D790 and using a universal machine test instaled at Leuven Composites Processing Centre (LCPC) of Katholieke Universiteit Leuven (Belgium). Table 1 shows the values for dimensions of specimen, as well as, the stacking sequences and the speed of the test (V). It's very important to note that the relation span/thickness is 32, where span is the distance between the supports of the flexural device.

Table 1. Specifications for 3-point bending tests (L = length; W = width; TH = thickness)

Stacking Sequence	**L [mm]**	**W [mm]**	**TH [mm]**	**V [mm/min]**
$[0]_{10}$	80	25	1.8	2.8
$[0/90/0/90/0]_s$	80	25	1.8	2.8
$[+45/-45/+45/0/90]_s$	80	25	1.8	2.8

The experimental tests were realized using displacement control. Thus, the normal displacement imposed at the center line of the specimen and the force measured in the same position were captured in order to obtain force-displacement graphs

3.3. Material model

The material model proposed simulates only the intra-ply damage mechanisms, using laminated shell elements and plane stress state. Afterthat, it will be developed a more complex material model in order to simulate the intra-ply and inter-ply damage mechanisms, using laminated solid elements and 3-D stress state. The material model was implemented as a UMAT subroutine into software ABAQUS™ (Abaqus, 2002).

Table 2 shows the material model used finite element analysis with shell elements, where a Failure Criteria associated to a Type of Failure constitutes the Material Model. Table 3 shows an explicit definition for all symbols and the material data used during the finite element analysis.

Table 2. Material Model I for Shell Element

Failure Criteria	**Type of Failure**	**Degradation Law**
$\left(\frac{\sigma_1}{X_C}\right)^2 = e_F^2 \begin{cases} e_F > 1 \rightarrow \text{damage} \\ e_F \leq 1 \rightarrow \text{no damage} \end{cases}$	Fiber Compression	$E_{11}^{df} = E_{11} \exp\left[-\left(\frac{A}{A_o}\right)H\right] - B\bar{\varepsilon}$ $G_{12}^{df} = G_{12} \exp\left[-\left(\frac{A}{A_o}\right)H\right] - B\bar{\varepsilon}$ $E_{22}^{df} \rightarrow 0$ $\nu_{12}^{df} \rightarrow 0$
$\left(\frac{\sigma_1}{X_T}\right)^2 + \frac{\left(2\sigma_{12}^2/G_{12}\right)+\left(3\alpha\sigma_{12}^4\right)}{\left(2S_{12}^2/G_{12}\right)+\left(3\alpha S_{12}^4\right)} = e_F^2$, i.e., $\left(\frac{\sigma_1}{X_T}\right)^2 + \frac{F_1}{F_2} = e_F^2 \begin{cases} e_F > 1 \rightarrow \text{damage} \\ e_F \leq 1 \rightarrow \text{no damage} \end{cases}$	Fiber Tension	
$\left(\frac{\sigma_2}{2S_{23}}\right)^2 + \left[\left(\frac{Y_C}{2S_{23}}\right)^2 - 1\right]\frac{\sigma_2}{Y_C} + \frac{F_1}{F_2} = e_M^2 \begin{cases} e_M > 1 \rightarrow \text{damage} \\ e_M \leq 1 \rightarrow \text{no damage} \end{cases}$	Matrix Compression	$E_{11}^{dm} \rightarrow E_{11}$ $E_{22}^{dm} \rightarrow 0$ $\nu_{12}^{dm} \rightarrow 0$
$\left(\frac{\sigma_2}{Y_T}\right)^2 + \frac{F_1}{F_2} = e_M^2 \begin{cases} e_M > 1 \rightarrow \text{damage} \\ e_M \leq 1 \rightarrow \text{no damage} \end{cases}$	Matrix Tension	

The Failure Criteria for the material model is based on works published by: Yamada and Sun (1978); Hashin (1979); Hashin (1980); Chang and Chang (1987a); Chang and Chang (1987b).

According to the failure (fiber or matrix failure), there is a Degradation Law in order to reduce the material properties of the failure ply based on the works published by: Chang and Chang (1987a); Chang and Chang (1987b); Chang, Liu and Chang (1991); Chang and Lessard (1991); Shahid and Chang (1995). For the matrix and fiber failure, E_{22} and ν_{12} are reduced to zero. However, E_{11} is not reduced, considering the matrix failure, but this property and G_{12} are reduced for fiber failure, based on an exponential decaying:

$$P^{df} = P \exp\left[-\left(\frac{A}{A_o}\right)H\right] - B\bar{\varepsilon} \quad (1)$$

Where P is the property to be reduced, A is the area of the damage zone; A_o is the area of the interaction zone of the fiber failure that was obtained from Chang and Chang (1987a, 1987b) and from Shahid and Chang (1995). The parameter H controls the level of the property degradation. Finally, the parameter B adjusts the degradation process according to the second invariant of deviatoric strain tensor ($\bar{\varepsilon}$), because the composite failure mechanisms influenced by the second invariant of deviatoric strain tensor was presented previously by Gosse (2001) and Tay et al (2005).

Table 3. Definition for the symbols of material model and the values for the finite element analysis

Symbol	**Definition**	**Values for finite element analysis**
σ_1	component stress acting at fiber direction	-
σ_2	component stress acting at normal to the fiber	-
σ_{12}	shear stress acting at the ply plane	-
X_T	strength value parallel to the fiber under tension [MPa]	1400
X_C	strength value parallel to the fiber under compression [MPa]	930
Y_T	strength value normal to the fiber under tension [MPa]	47
Y_C	strength value normal to the fiber under compression [MPa]	130
S_{12}	strength value for shear at the ply plane [MPa]	53
S_{23}	strength value for shear transverse to the ply plane [MPa]	89
e_F	failure index for the fiber	-
e_M	failure index for the matrix	-
E_{11}	Young Modulus at fiber direction [GPa]	127
E_{22}	Young Modulus at normal to the fiber [GPa]	10
G_{12}	Shear Modulus at the ply plane [GPa]	5.4
ν_{12}	Poisson’s ratio	0.34
df	Index for Modulus degradeted after fiber damage	-
dm	Index for Modulus degradeted ater matrix damage	-

The Figure 3 shows the E_{11} degradation law for Chang and Chang (1987a, 1987b) and for present work, where β is the Weibull distribution parameter. The parameter H is similar to parameter β, but the degradation phenomenon is more slightly for equation 1. Thus, it is not verified that the Young modulus reduces suddenly, which is very reasonable for flexural quasi-static tests.

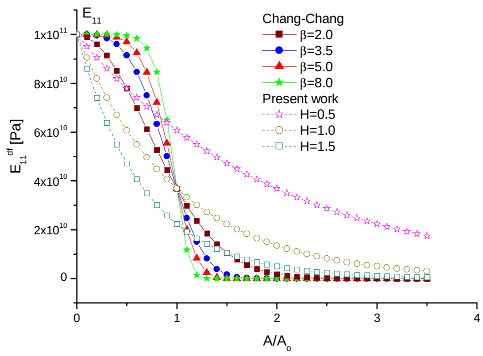


Figure 3. E_{11} degradation laws

However, G_{12} degradation law also is updated according to the non-linear behavior of the shear stress at plane 1-2. Thus, $\overline{G}_{12}$ is the updated shear modulus, which is calculated using the tangent of the curve shear stress-shear strain:

$$\overline{G}_{12} = \frac{1}{\dfrac{1}{G_{12}} + 3\alpha\sigma_{12}^{2}} \tag{2}$$

Where α represents the value provided by the stress tensor component of the fourth order showed by Hahn and Tsai (1973).

3.4. Finite element analysis

The Finite Element Analysis (FEA), using UMAT subroutines, were realized to evaluate the performance of material model to predict the composite structure behavior. Numerical results as force-displacement graphs were compared to experimental results from experimental tests.

Material model were implemented by UMAT subroutine into software ABAQUS™, because some failure criteria adopted for these material models were used by other researchers, but, the combination of the failure criteria is a new investigation and the degradation law proposed at equation 1 is a new investigation, too.

The stacking sequences investigated are shown at table 1, in order to evaluate the influence of the fiber orientation on the composite structure behavior, and, to evaluate the material models performance for predicting the composite structure behavior.

For the finite element analysis were created using shell elements (S4) with 4 nodes and 4 integration points (Abaqus, 2002) in order to simulate the plane stress at the laminate. The line of nodes, which represents the support of the flexural device, were fixed the diplacements at direction 3 and rotations at direction 1 to simulate the supports (Fig.4). The force applied by the machine test was simulated as a displacement imposed on the nodes at the center line of the laminate at direction 3. Therefore, the loading phase was simulated, i.e., the nodes were displaced from the initial position to the final position (maximum displacement verified at the experimental test).

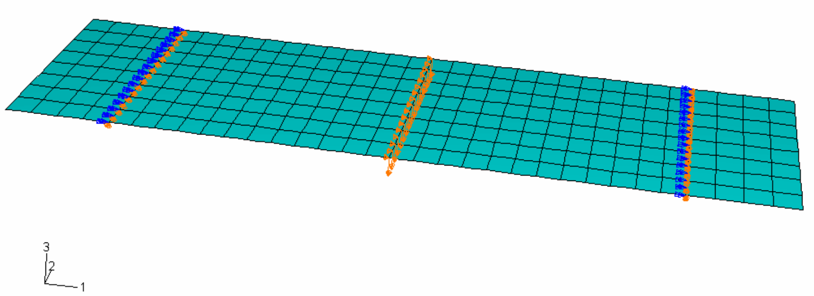


Figure 4. Finite element model.

Figure 5 shows that the software ABAQUS^{TM} manages all calculation procedures, and the user can recover components of the stress tensor for each iteration i. After that, the stress tensor for each integration point is read by the UMAT subroutine in order to be used in a Failure Criteria. If failure doesn't occur then the updated stress tensor is equal to the original stress tensor. Thus, this tensor comes back to the principal calculus procedure in order to form the internal forces vector, which will be compared to the external forces vector. The result of this comparison will produce residues (R) that will be compared to a specified tolerance. If R is higher than the tolerance, it will be necessary a new iteration.

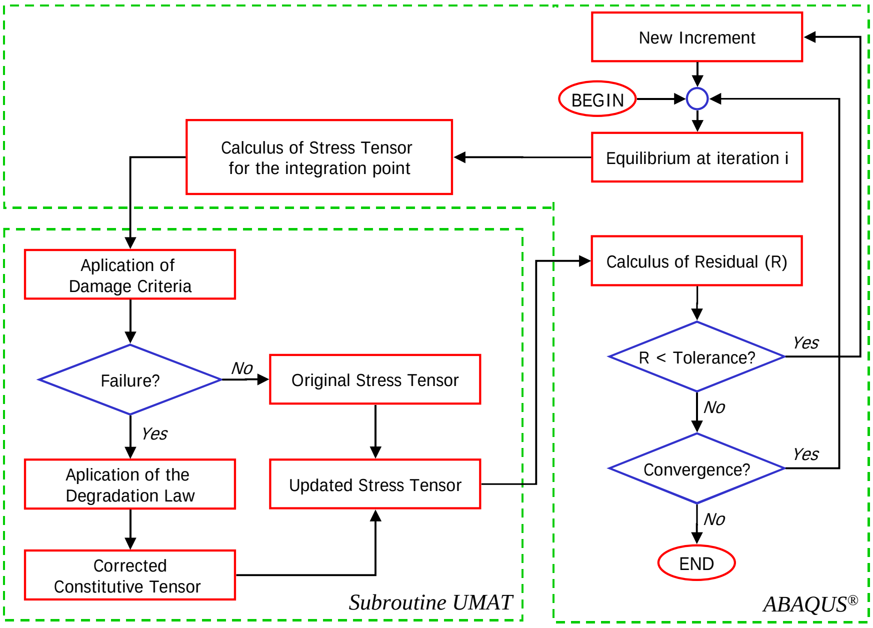


Figure 5. UMAT subroutine and software ABAQUS^{TM}.

However, if failure occurs then a Degradation Law is applied on the elastic properties of the ply, according to the type of failure (Table 2 and 3). Therefore, the constitutive tensor must be corrected and the stress tensor will be updated before to return to software ABAQUS^{TM}. Then, R is calculated with the internal and external forces vector in order to be compared to the thereshold that is set to indicate convergence. If there is convergence, then a new increment of load occurs, otherwise, the software ABAQUS^{TM} checks the convergence. During the iterations for only one load step, it is expected that R decrease. If R increases, then the solution produces divergence, and the procedure stops. Frequently, the divergence occurs, because the material properties have high degradation, and the structure doesn't have enough stiffness to support the loads imposed. Other critical parameter, which can cause divergence in nonlinear Finite Element Analysis, is the time step applied on the simulations. For the initial time increment, it was used 0.05, but this value would be modified as required, because the automatic time stepping scheme was set on. For the time period of the step, it was used 15. For the minimum and maximum time increment allowed, they were chosen 0.0001 and 0.5, respectively. Finally, for the maximum number of increments in a step, it was chosen 300.

4. RESULTS AND DISCUSSIONS

The finite element results are shown, comparing the force-displacement graphs between numerical and experimental results. The stacking sequences investigated are shown at table 1, in order to evaluate the material model performance for predicting the composite laminate behavior on flexural and transverse shear loading.

4.1. Results for laminate $[0]_{10}$

Figure 6(a) shows that the first ply failure occurs close to 1.1 kN for the experimental test. It was verified during the tests that the fibers outer of laminate broke under tension. Thus, the stiffness of the structure was reduced strongly. Afterthat, over to 4.0 mm of deflection, the specimen presents new other failures, like matrix cracking and fiber breakage close to the support of the device flexural which reduced sligthly the stiffness of the structure.

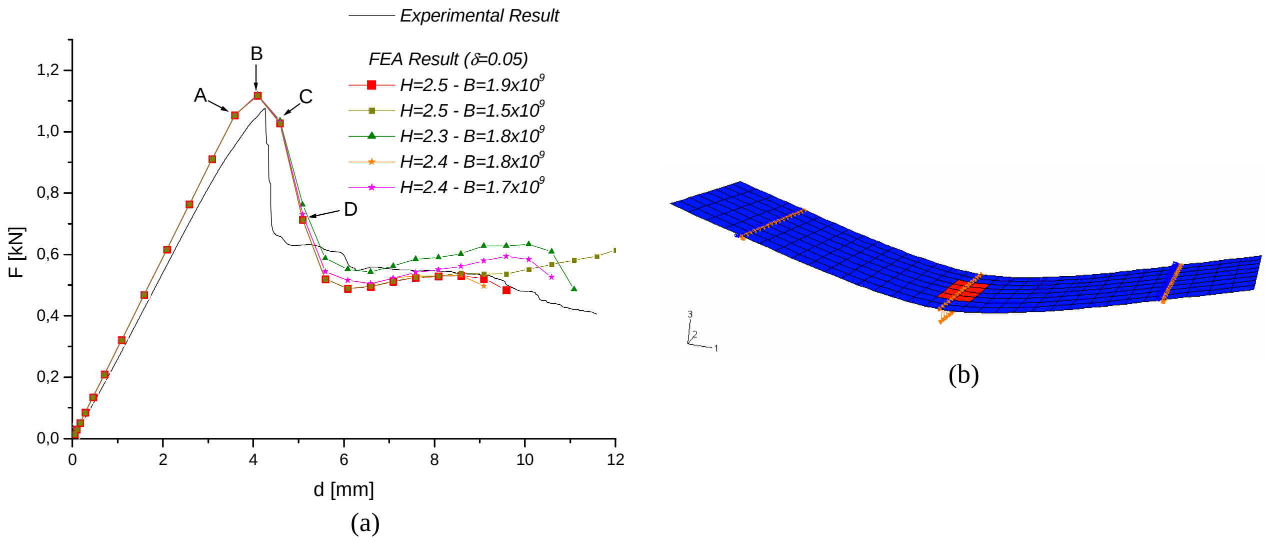


Figure 6. (a) Numerical and experimental results for laminate $[0]_{10}$; (b) First failure elements

Figure 6(a) shows that the first ply failure occurs between 1.0 kN and 1.1 kN for the numerical analysis. It was verified during the FEA that the first failure elements are in the center line of the specimen and the plies are outer of the laminate under tension (Figure 6(b)). Therefore, the failure criteria proposed for the present material model can simulate the first ply failure for this laminate. Afterthat, the mechanical properties are reduced according to the degradation law. For this case, they were investigated some values for the H and B parameters. It was verified for the first (point A), second (point B) and third (point C) failures that the parameters didn't influence the results. Besides, it was verified that after third failure (point C), the numerical results didn't fit to the experimental results, because the stiffness reduction is not so strong. After fourth failure (point D), the numerical result is sensible to the H and B parameters. After some finite element analysis, it was concluded that the H parameter should be between 2.3 and 2.5 and the B parameter should be between $1.5x10^9$ and $1.9x10^9$.

Althought the mechanical behavior of this laminate during the danification process was simulated, using the parameters H and B set like described, the finite element model was investigated for different number of increments (Figure 7). It was verified that the number of the increments can change the numerical results. It's interesting to note for 35 and 41 increments, the first ply failure occurs before than for 29 increments. Besides the numerical curve for higher increments is more smoth than numerical curve for 29 increments, because equilibrium iteration occurs more time.

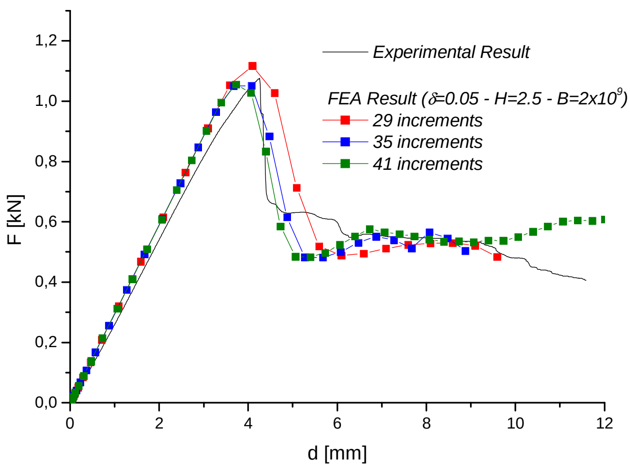


Figure 7. Test of increment numbers for laminate $[0]_{10}$

4.2. Results for laminate $[0/90/0/90/0]_s$

Figure 8 shows that the first ply failure occurs close to 0.7 kN for the experimental test (point A). It was verified during the tests that the matrix crackings occur. Thus, the stiffness of the structure was reduced slightly up to 0.8 kN (point B). In that moment, first delaminations occur between the plies $0^o/90^o$ which were close to the device flexural, which applies force, i.e., the plies under compression over the neutral line. Afterthat, it was verified that the stiffness of the structure was reduced strongly. However, from 5.4 mm to 6.6 mm (point C) of deflection, the force level is almost constant, because the delaminated area only increases. At the 6.6 mm of deflection (point C), fiber breakage occurs at the plies outer laminate under tension, and the stiffness of the structure was reduced strongly, again. At 7.7 mm (point D), new delaminations and fiber breakage occur, causing a stiffness reduction of the laminate which supports the loadings up to 10 mm, when the structure show the catastrophic failure.

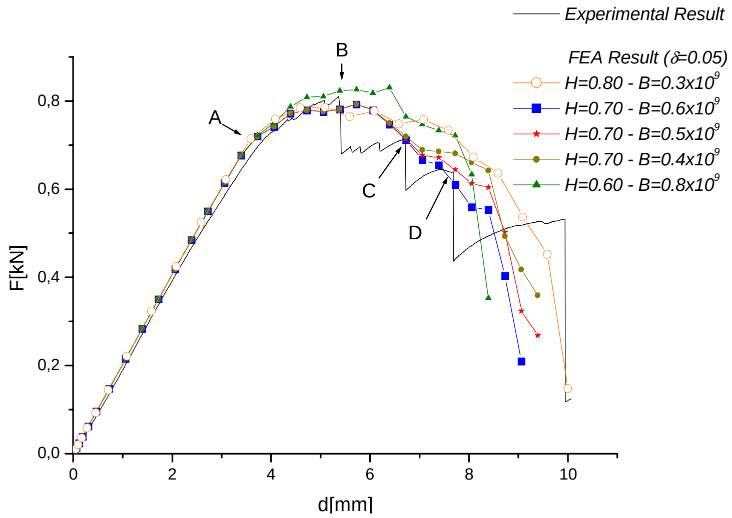


Figure 8. Numerical and experimental results for laminate $[0/90/0/90/0]_s$

It's very important to note that the delaminations were developed between the plies $0^o/90^o$. This phenomenon can occur due to the difference between stiffness of the adjacent plies with different angle ply which promotes the interlaminar shear, because each ply has a specific strain mode. The interlaminar shear avoids that the plies separate, moving the plies together. However, if the stress acting on the laminate is higher than the interlaminar strength, the adjacent plies separate, providing delamination.

Figure 8 shows that the first ply failure occurs close to 0.7 kN for the numerical analysis. It was verified during the FEA that the first failure elements are in the center line of the specimen, because matrix failures are simulated. Afterthat, the degradation process is activated and the numerical stiffness reduction is very close to the experimental stiffness reduction. Therefore, the failure criteria proposed for the present material model can simulate the first ply failure and the initial degradation process for this laminate. The mechanical properties are reduced according to the degradation law. For this case, they were investigated some values for the H and B parameters. It was verified for the first failures (point A) that the parameters didn't influence the results. Besides, it was verified that after second failure (point B), the numerical results are influenced only by B parameter. Finaly, after third failure (point C), the numerical results are influenced by H and B parameters. After some finite element analysis, it was concluded that the H parameter should be between 0.6 and 0.8 and the B parameter should be between $0.3x10^9$ and $0.8x10^9$.

It's very important to note that after second failure (point B), the numerical results didn't fit to the experimental results, because the stiffness reduction is very strong due to delamination and this material model proposed can't simulate this phenomenon. The same comment can be applied after third failure (point C) and fourth failure (point D). However, the material model proposed can provide force-displacement graphs similar to the experimental curve, except the abrupt reduction of the force level due to the delaminations.

4.3. Results for laminate $[+45/-45/+45/0/90]_s$

Figure 9 shows that the laminate has a non-linear behavior caused by the shear stress in the ply plane due to plies $+/-45^o$. After 8.5 mm of deflection, the stiffness of the laminate increases due to a combination of phenomena: large displacement (non-linear geometric behavior); increase of the friction between specimen and supports; the fibers oriented at $+/-45^o$ tends to allign to the tension stress direction at the ply. Finally, at 0.61 kN (point A), delaminations occur between the plies $0^o/90^o$ and between the plies $45^o/0^o$, causing a strong stiffness reduction. Afterthat, the structure show the catastrophic failure.

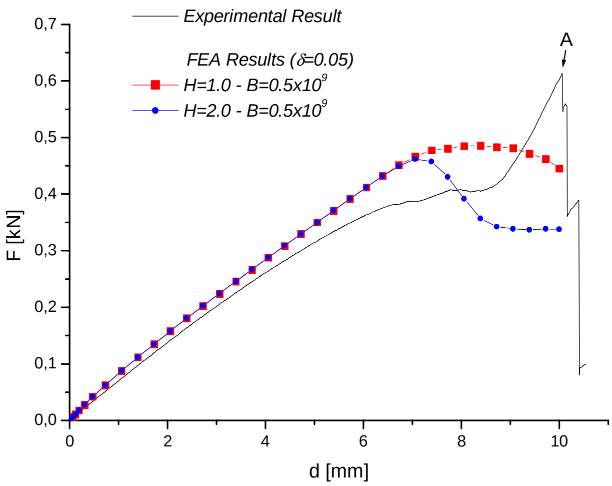


Figure 9. Numerical and experimental results for laminate $[+45/-45/+45/0/90]_s$

Figure 9 shows that the material model can simulate the laminate behavior up to 5.0 mm of specimen deflection. After some finite element analysis, it was concluded that the H parameter should be between 1.0 and 2.0 and the B parameter should be around $0.5x10^9$.

However, after 5.0 mm, it was verified that the model can't simulate the non-linear behavior of the laminate. Therefore, the numerical results didn't fit to the experimental results, because the stiffness increase is very strong due to a combination of phenomena and this finite element model can't predict the friction between the specimen and the supports. Besides, this material model proposed can't simulate the allignment of fibers oriented at $+/-45^o$ in the tension stress direction at the ply. Finally, the delaminations were not simulated too.

5. CONCLUSIONS

The material model proposed can simulate the intra-ply failure the damage evolution for composite laminate on flexural and transverse shear loading, if the parameter H and B are adjusted according to the experimental results. However, the material model proposed can't represent the delaminations, because any failure criteria and material degradation law were defined for this model. Thus, for some regions of the force-displacement graphs, the numerical result doesn't converge to the experimental result. Therefore, a new material model have been proposed by the authors in order to simulate the delaminations. Besides, the finite element model can be improved in order to simulate the contact phenomenon between specimen and supports.

Finally, the material model implemented can be used as a computational tool in order to help for developing of the aircraft composite structure.

6. ACKNOWLEDGEMENTS

The authors would like to thank Dr. Ir. Mieke Lossie and technitian Paul Van Cauwenbergh (Mechanical Engineering Department of Katholieke Universiteit Leuven) for the technical support during the realization of this work. The authors would like to thank Professor Ignaas Verpoest (Material Engineering Department of Katholieke Universiteit Leuven - Belgium) for allowing the realization of this work at LCPC (Leuven Composites Processing Centre). The authors would like to thank FAPESP for the finnantial funds to realize this work.

7. REFERENCES

Abaqus, 2002, "Standard User's Manual: Theory", Pawtucket: Hibbitt, Karlsson & Sorensen, USA.

Anderson, T.L.,1995, "Fracture Mechanics - Fundamentals and Applications", Ed. CRC Press, New York, USA, 688p.

American Society for Testing and Materials, 1996, "D 790–96a: Standard Test Method for Flexural Properties of Unreinforced and Reinforced Plastics and Electrical Insulating Materials", CD-Rom , Philadelphia, USA.

Chang, F.K. and Chang, K.Y, 1987a, "Post-failure Analysis of Bolted Composite Joints in Tension or Shear-out Mode Failure", Journal of Composite Materials, Vol. 21, No. 9, pp. 809-833.

Chang, F.K. and Chang, K.Y., 1987b, "A Progressive Damage Model for Laminated Composites Containing Stress Concentrations", Journal of Composite Materials, Vol. 21, No. 9, pp. 834-855.

Chang, K.Y., Liu, S. and Chang, F.K.,1991, "Damage Tolerance of Laminated Composites Containing an Open Hole and Subjected to Tensile Loadings", Journal of Composite Materials, Vol. 25, No. 3, pp. 274-301.

Chang, F.K. and Lessard, L.B., 1991, "Damage Tolerance of Laminated Composites Containing an Open Hole and Subjected to Compressive Loadings. Part I – Analysis", Journal of Composite Materials, Vol. 25, No. 1, pp. 2-43.

Goose, J.H., 2001, “Strain Invariant Failure Criteria for Polymers in Composite Materials”, Proceeding of the 42nd AIAA Conference, Seatle, USA.
Hahn, H.T. and Tsai, S.W, 1973, “Nonlinear Elastic Behavior of Unidirectional Composite Laminae”, Journal of Composite Materials, Vol. 7, pp. 102-118.
Hashin Z., 1979, “Analysis of Properties of Fiber Composites with Anisotropic Constituents”, Journal of Applied Mechanics, Vol. 46, No. 3, pp. 543-550.
Hashin, Z., 1980, “Failure Criteria for Unidirectional Fiber Composites”, Journal of Applied Mechanics , Vol. 47, No. 2, pp: 329-334.
Hinton, M.J.; Kaddour, A.S. and Soden, P.D., 2002a, “Evaluation of Failure Prediction in Composites Laminates: Background to “Part B” of the Exercise”, Composites Science and Technology, Vol. 62, No. 12-13, pp.1481-1488.
Hinton, M.J.; Kaddour, A.S. and Soden, P.D., 2002b, “A Comparison of the Predictive Capabilities of Current Failure Theories for Composite Laminates, Judged Against Experimental Evidence”, Composites Science and Technology, Vol. 62, No. 12-13, pp.1725-1797.
Hinton, M.J.; Kaddour, A.S. and Soden, P.D., 2004a, “Evaluation of Failure Prediction in Composites Laminates: Background to “Part C” of the Exercise”, Composites Science and Technology, Vol. 64, No. 3-4, pp.321-327.
Hinton, M.J.; Kaddour, A.S. and Soden, P.D., 2004b, “A Further Assessment of the Predictive Capabilities of Current Failure Theories for Composite Laminates: Comparison with Experimental Evidence”, Composites Science and Technology, Vol.64, No. 3-4, pp. 549-588.
Kaddour, A.S.; Hinton, M.J. and Soden, P.D., 2004, “A Comparison of the Predictive Capabilities of Current Failure Theories for Composite Laminates: Additional Contributions”, Composites Science and Technology, Vol.64, No.3-4, pp.449-476.
París, F., 2001, “A Study of Failure Criteria of Fibrous Composite Materials”, NASA Report n.CR-2001-210661/March 2001.
Rowlands, R.E., 1985, “Strength (Failure) Theories and Their Experimental Correlation”. In: Sih, G.C. and Skudra, A.M., 1985, “Failure Mechanics of Composite”, Ed. North-Holland, Amsterdam, Chapter 2, pp.71-127.
Shahid, I. and Chang, FK., 1995, “An Accumulative Damage Model for Tensile and Shear Failures of Laminated Composite Plates”, Journal of Composite Materials, Vol. 29, No. 7, pp. 926-981.
Soden, P.D.; Hinton, M.J.; Kaddour, A.S., 1998a, “Lamina Properties, Lay-up Configurations and Loading Conditions for a Range of Fibre-Reinforced Composite Laminates”, Composites Science and Technology, Vol.58, No.7, pp.1011-1022.
Soden, P.D.; Hinton, M.J.; Kaddour, A.S., 1998b, “A Comparison of the Predictive Capabilities of Current Failure Theories for Composite Laminates”, Composites Science and Technology. Vol.58, No.7, pp.1225-1254.
Tay, T.E; Tan, S.H.N.; Tan, V.B.C.; Goose, J.H., 2005, “Damage Progression by the Finite Element-Failure Method (EFM) and Strain Invariant Failure Theory (SIFT)”, Composites Science and Technology, Vol.65, pp.935-944.
Yamada, S.E. and Sun, C.T., 1978, “Analysis of Laminate Strength and Its Distribution”, Journal of Composite Materials, Vol.12, pp. 275-284.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.