



HAL
open science

Nonlinear finite element analysis of composite bolted lap joints: experimental vs numerical tests

Benoît Montagne, Frederic Lachaud, Eric Paroissien, Dominique Martini,
Fabrice Congourdeau

► To cite this version:

Benoît Montagne, Frederic Lachaud, Eric Paroissien, Dominique Martini, Fabrice Congourdeau. Non-linear finite element analysis of composite bolted lap joints: experimental vs numerical tests. The 22nd International Conference on Composite Materials (ICCM22), Aug 2019, Melbourne, Australia. pp.1-11. hal-02881211

HAL Id: hal-02881211

<https://hal.science/hal-02881211>

Submitted on 25 Jun 2020

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



Open Archive Toulouse Archive Ouverte (OATAO)

OATAO is an open access repository that collects the work of some Toulouse researchers and makes it freely available over the web where possible.

This is an author's version published in: <https://oatao.univ-toulouse.fr/24592>

To cite this version :

Montagne, Benoît and Lachaud, Frédéric and Paroissien, Eric and Martini, Dominique and Congourdeau, Fabrice Nonlinear finite element analysis of composite bolted lap joints: experimental vs numerical tests. (2019) In: 22nd International Conference on Composite Materials (ICCM22), 11 August 2019 - 16 August 2019 (Melbourne, Australia).

Any correspondence concerning this service should be sent to the repository administrator:

tech-oatao@listes-diff.inp-toulouse.fr

NONLINEAR FINITE ELEMENT ANALYSIS OF COMPOSITE BOLTED LAP-JOINTS: EXPERIMENTAL VS. NUMERICAL TESTS

B. Montagne¹, F. Lachaud², E. Paroissien³, D. Martini⁴ and F. Congourdeau⁵

¹Institut Clément Ader (ICA), Université de Toulouse, ISAE-SUPAERO, INSA, IMT MINES ALBI, UTIII CNRS, 3 Rue Caroline Aigle 31400 Toulouse, France, benoit.montagne@isae.fr, <http://institut-clement-ader.org/>

²Institut Clément Ader (ICA), Université de Toulouse, ISAE-SUPAERO, INSA, IMT MINES ALBI, UTIII CNRS, 3 Rue Caroline Aigle 31400 Toulouse, France, frederic.lachaud@isae.fr, <http://institut-clement-ader.org/>

³Institut Clément Ader (ICA), Université de Toulouse, ISAE-SUPAERO, INSA, IMT MINES ALBI, UTIII CNRS, 3 Rue Caroline Aigle 31400 Toulouse, France, eric.paroissien@isae.fr, <http://institut-clement-ader.org/>

⁴DASSAULT Aviation, 78 Quai Marcel Dassault 92210 Saint-Cloud, France, dominique.martini@dassault-aviation.com, <https://www.dassault-aviation.com/fr/>

⁵DASSAULT Aviation, 78 Quai Marcel Dassault 92210 Saint-Cloud, France, fabrice.congourdeau@dassault-aviation.com, <https://www.dassault-aviation.com/fr/>

Keywords: Composite materials, Bolted joints, Finite element analysis, Damage model

ABSTRACT

Carbon fiber reinforced plastics (CFRP) are widely used in aircraft industry because of their high mechanical properties. However these properties considerably drop when composite materials are drilled and bolted. Thus it is important to master the composite bolted joints behavior to avoid the structural failure. Progressive damage behavior for composite materials developed at Institut Clément Ader (ICA) is implemented on finite element (FE) models to simulate the behavior of supported single lap joints and double lap joints specimens. An in-plane shell model is sufficient to accurately simulate double lap joint tests. However this kind of model does not allow for an accurate representation for a single lap joint test because of the out of plane phenomena occurring during the test. A model with shell behavior is then developed. Damaging scenarios are elaborated to identify the composite material degradations leading to the structural stiffness loss. Compressive then tensile fiber failure in the 0° plies are the main reasons of this stiffness loss.

1 INTRODUCTION

Carbon fiber reinforced plastics (CFRP) are considered as a mean to increase future aircraft performances. That is why these materials are more and more used by aircraft manufacturers. Even if composite materials have interesting mechanical properties, these properties considerably drop when these materials are drilled and bolted. Moreover civil aircrafts count thousands of fasteners. Thus it is necessary for the manufacturers to master the behavior of bolted joints in order to optimize them.

DASSAULT AVIATION and ICA lead a research project on composite bolted joints behavior. The goal of this project is to better understand the different failure modes [1] of these junctions as function of design parameters (edge distance, lay-up, etc.). Basing on advanced modelling tools for damaging behavior of these materials, robust simulations of failure behavior are developed. They are validated by comparing experimental tests results and numerical tests predictions.

First, this work focuses on an important experimental database analysis on composite bolted joints provided by DASSAULT AVIATION. This database contains double lap joint and supported single lap joint tests results. These joint configurations are considered as representative for bolted joints on aircraft structure. Design rules have been established [2] thanks to experimental observations but these tests have a high cost in time and resources. That is why numerical models are developed in order to

accurately represent those physical phenomena at a lower cost. The limits of standard models (assuming a linear behavior of the composite material) to predict the failure behaviors of this kind of joint have been observed [3]. These failure behaviors are then approached thanks to an advanced model developed at ICA. In this paper we will first present the nonlinear material law. It is used on an in-plane shell model developed using the FE code SAMCEF®. It permits to model double-lap joint (DLJ) tests. A second model is developed based on shell element behavior to simulate supported single lap joint (SSLJ) tests.

2 COMPOSITE NONLINEAR BEHAVIOR

The composite nonlinear law is a damage one initially developed by Matzenmiller, Lubliner and Taylor (MLT) [4]. It has been turned into a SAMCEF® user law at ICA [5]–[7]. This law works at the ply scale thanks to composite shell elements. Degradation of the material can be described with failure criteria corresponding to different failure modes. Damage variables $0 < d_i < 1$ are applied to the Young moduli E_{jk} of the composite material in the compliance matrix C_{dam} in (1).

$$C_{dam} = \begin{pmatrix} \frac{1}{E_{11}(1-d_1)} & -\frac{\nu_{12}}{E_{11}} & -\frac{\nu_{13}}{E_{11}} & 0 & 0 & 0 \\ -\frac{\nu_{12}}{E_{11}} & \frac{1}{E_{22}(1-d_2)(1-d_2^b)} & -\frac{\nu_{23}}{E_{22}} & 0 & 0 & 0 \\ -\nu_{13}/E_{11} & -\nu_{23}/E_{22} & \frac{1}{E_{33}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{12}(1-d_4)(1-d_4^b)} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{23}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{13}} \end{pmatrix} \quad (1)$$

These damage variables are calculated (2) based on variables ϕ_i which represent for the degradations associated with a given failure criterion.

$$\begin{pmatrix} d_1 \\ d_2 \\ d_4 \end{pmatrix} = \begin{pmatrix} 1 & 1 & 0 \\ 0 & 0 & 1 \\ 0 & 0 & 1 \end{pmatrix} \begin{pmatrix} \phi_1 \\ \phi_2 \\ \phi_4 \end{pmatrix} \quad (2)$$

An evolution law [8] is applied to the failure criteria in order to describe the damageable behavior of the material :

$$c_i = \max(\sqrt{f_i}, 1) \quad (3)$$

$$\phi_i = 1 - \exp\left(\frac{1 - c_i^{m_i}}{m_i}\right) \quad (c_i > 1) \quad (4)$$

Where m_i is a related parameter which drives the evolution of ϕ_i . The higher this parameter is the more brittle is the behavior.

Criteria are computed with the elastic stresses within the material. The failure modes considered here are: the fibers tensile failure (5) the fibers compressive failure (6) and the matrix failure (7 and 8):

$$f_1 = \left(\frac{\langle \sigma_{11} \rangle_+}{\sigma_{11}^{RT}}\right)^2 + \frac{\sigma_{12}^2}{\sigma_{12}^{RS}} \quad (5)$$

$$f_2 = \left(\frac{\langle -\sigma_{11} \rangle_+}{\sigma_{11}^{RC}}\right)^2 \quad (6)$$

$$f_4 = \left(\frac{\langle \sigma_{22}(1 - d_2^{t-1}) \rangle_+}{\sigma_{22}^{RT}(1 - d_2^{t-1})} \right)^2 + \left(\frac{\langle -\sigma_{22} \rangle_+}{\sigma_{22}^{RC}} \right)^2 + \left(\frac{\sigma_{12}(1 - d_4^{t-1})}{\sigma_{12}^R(1 - d_4^{t-1}) + \langle -\sigma_{22} \rangle_+ \tan(\phi)} \right)^2 \quad (7)$$

$$f_4^b = \left(\frac{\langle \sigma_{22}(1 - d_2^{t-1}) \rangle_+}{\sigma_{22}^{RT}} \right)^2 + \left(\frac{\langle -\sigma_{22} \rangle_+}{\sigma_{22}^{RC}} \right)^2 + \left(\frac{\sigma_{12}(1 - d_4^{t-1})}{\frac{\sigma_{12}^R}{k} + \langle -\sigma_{22} \rangle_+ \tan(\phi)} \right)^2 \quad (8)$$

Where:

- $\langle \cdot \rangle_+$ are the Macaulay brackets,
- R in the exponent indicate a failure stress : T for the tensile stress, C for the compressive one and S for the shear, identified experimentally,
- d_j^{t-1} is the damage variable at the previous step of the computation,
- $d_{j,i}^t$ is the damaging variable at the i^{th} iteration of the current step of the computation.

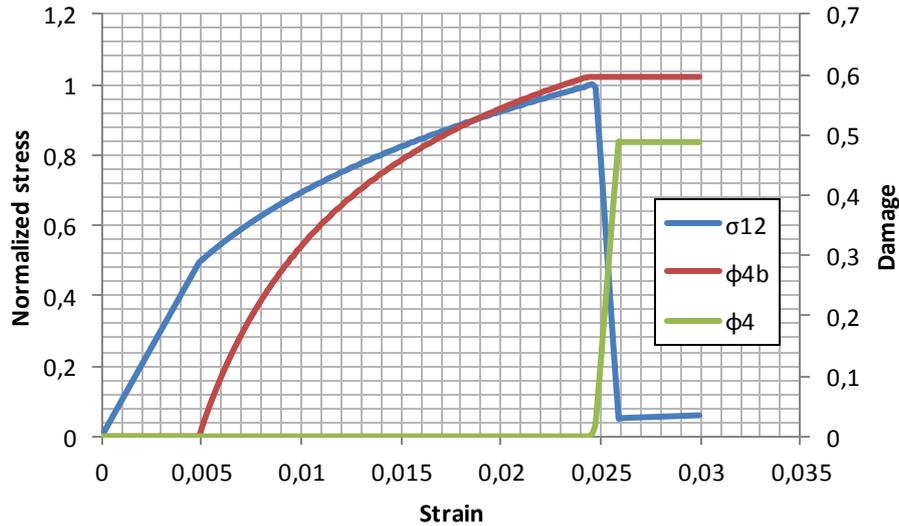


Figure 1: Nonlinear shear behavior

The shear nonlinear behavior of the composite material is taken into account thanks to a diffuse damage variable ϕ_4^b . The related criterion is presented in (8) and illustrated in Fig. 1. The coefficient k reduces the shear failure stress in order to progressively decrease the shear modulus until the shear stress reaches the failure stress. Then the second damage variable is activated and the failure happens. A delayed damage effect [9] is applied to limit the localization of the damage in the model and to suppress the mesh dependency. The different parameters are identified thanks to numerical vs experimental comparisons.

3 FINITE ELEMENT MODELS

3.1 Double lap joint tests modeling

An in-plane shell FE model (termed 2D FEM) is used to simulate tensile tests on DLJ specimens. Here the composite plate is jointed with two aluminum plates. The composite plate has a width of 5 times the diameter Φ of the bolt (here $\Phi=6\text{mm}$) and an end distance of 2.5Φ . It is meshed with stratified membrane elements. They allow for the modelling of a laminate with a layer by layer representation for a low computational cost. This choice of in-plane modelling has been made because

out of plane displacements are small in this kind of test. Because of its bending, the bolt axis tends to press unequally the hole in the composite plate. This phenomenon is neglected here. The mesh is refined in areas of high stress gradients, i.e. in the vicinity of the hole. A convergence study has been made on this model to validate the mesh but it is not presented here. The scheme (Fig. 2) presents the 2D FEM used and the boundary conditions applied on it.

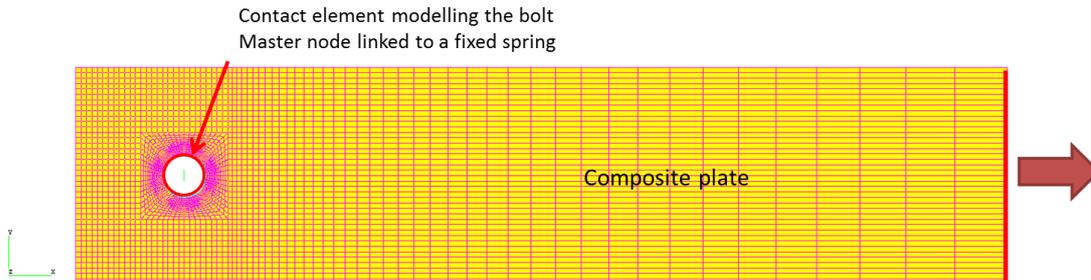


Figure 2: Mesh and boundary conditions for 2D FEM

The bolt is modeled thanks to contact elements acting as a rigid cylindrical surface. They are connected to a fixed spring. Its stiffness corresponds to the ratio between the force and displacement of a beam supported at its two extremities (Eq. 9).

$$f = \frac{FL^3}{48EI} \text{ and } K = \frac{F}{f} = \frac{48EI}{L^3} \quad (9)$$

Where E is the young modulus of the material of the beam, L its length and I its second moment of area. The simulation is lead with mecano: the nonlinear solver of SAMCEF. Geometrical nonlinearities are not taken into account here.

3.2 Supported single lap joint modeling

Tensile tests on SSLJ are modeled thanks to a shell finite element model (termed 2.5D FEM). The composite plate is jointed with an aluminum plate. The composite plate is meshed using laminated shell elements (Fig. 3). It has the same dimensions as the previous model (width of 5Φ and end distance of 2.5Φ). The aluminum plate is meshed using classical shell elements. An elastoplastic bilinear law (Fig. 4) is introduced for the aluminum plate. Finally the bolt is represented by contact elements simulating:

- the bolt in contact with the composite plate by a rigid cylindrical surface
- the bolt in contact with the aluminum plate by a rigid cylindrical surface
- the bolt head by a circular surface
- the nut by a circular surface

The master nodes of these contact element groups are linked by steel beam elements of circular section. The radius of this section is equal to the radius of the bolt. In the scheme below T are the nodal displacements and R are the nodal rotations.

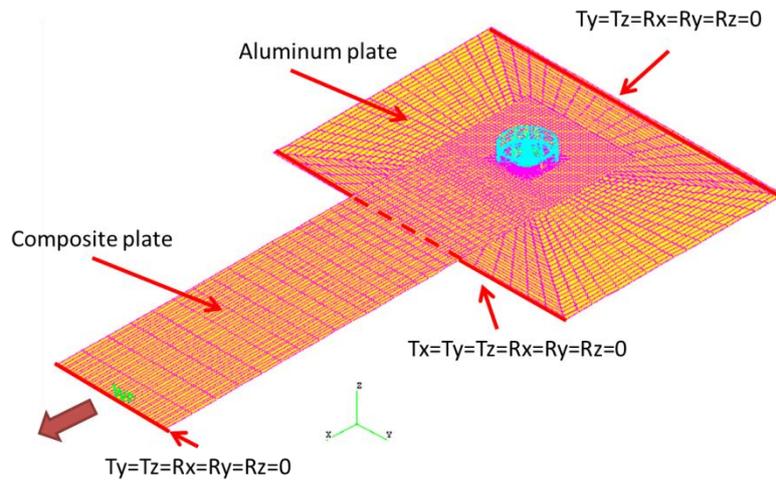


Figure 3: Mesh and boundary conditions for 2.5D FEM

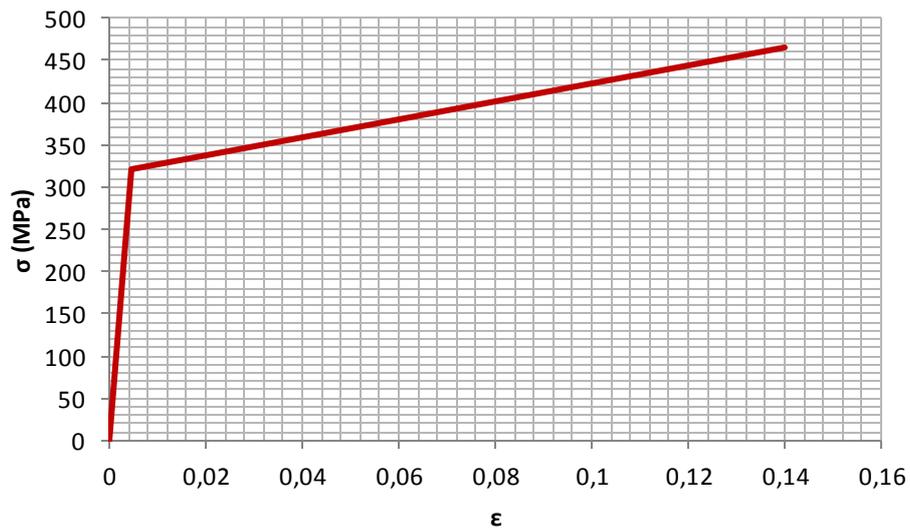


Figure 4: Elastoplastic behavior for the aluminum

Mecano is still the solver used and the geometrical nonlinearities are taken into account here.

4 RESULTS

The damageable behavior presented in this paper is applied to the composite plates of both models. The results obtained for DLJ and SSLJ tests are then presented.

4.1 Double lap joint

In the load vs displacement curves presented the load is measured by the sensor of the machine and the displacement by an extensometer as shown in the picture given in Fig. 5.

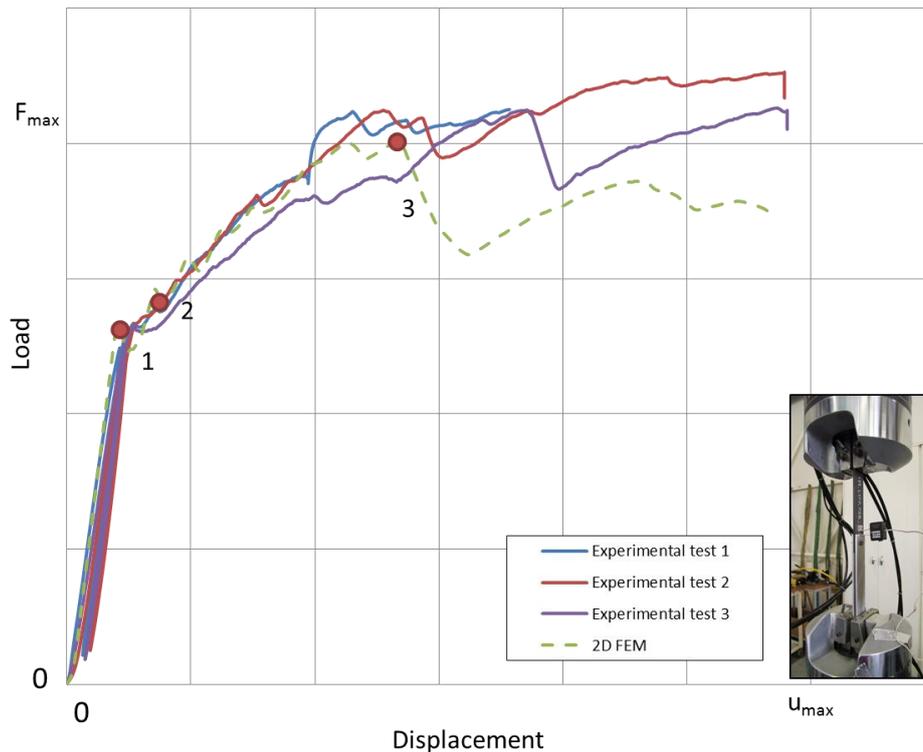


Figure 5: Experimental and numerical load-displacement curves for DLJ tests

The results are close between experimental and numerical ones until the point 3. Then the load decreases more in the model than during tests. This can be explained with the confinement of the damage between the aluminum plates. The hypothesis made here is that the aluminum plates limit the damage propagation in the composite plate during the test. It leads to an augmentation of the compressive yield stress of the composite plates and as a consequence a load drop less important [10]. Out of plane phenomena are not taken into account in the 2D FEM. That is why the gap between experimental and numerical results increases after the point 3. The matrix is damaging for very small displacements. This has no influence on the linear behavior of the structure. Compressive and tensile fibers damages are represented on Fig. 6 and Fig. 7. Plies oriented at -45° are not represented here because the damages are symmetrical with the ones of 45° plies. Fibers start to damage in compression at point 1. It appears in all the orientations simultaneously. This damage propagates in the laminate and tensile damages appear at point 2. Once again the different orientations damage initiate at the same time. The load drop at point 3 is due to the propagation of the tensile fiber damage in the 0° plies. Then 45° plies are damaging in the zone where 0° plies are broken because after that they are the only plies with -45° ensuring the tensile load transfer. Finally the different damages progress in the composite material and continually decrease the global stiffness of the joint.

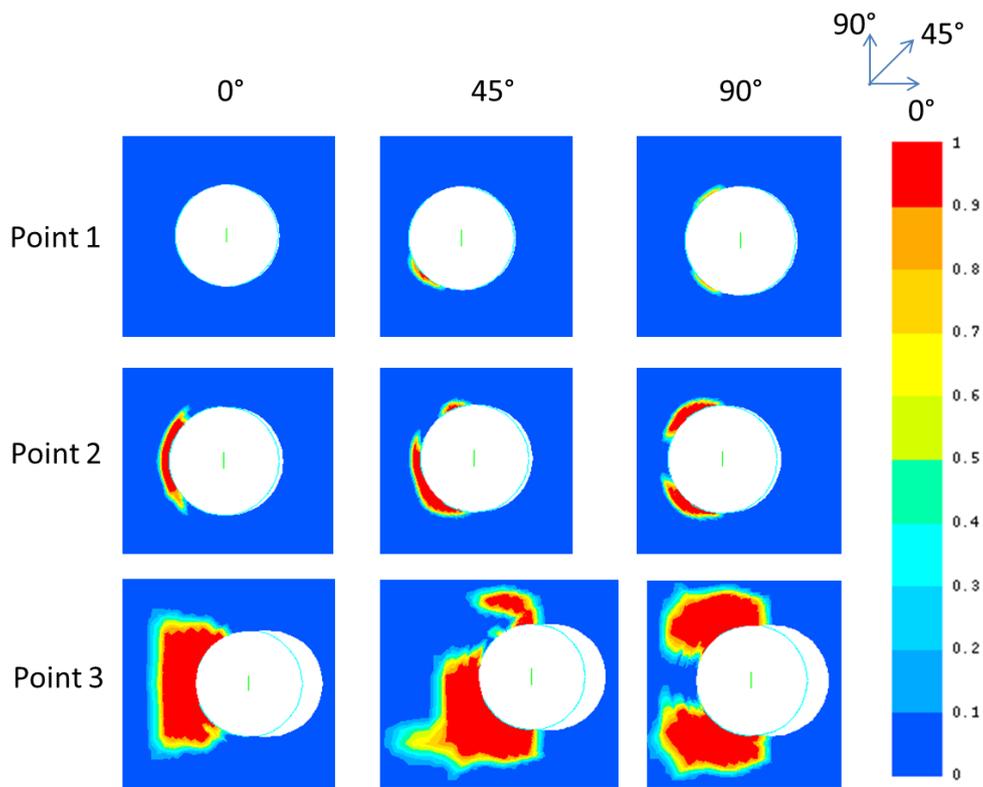


Figure 6: Damaging in compression of the fibers oriented at 0° 45° and 90°

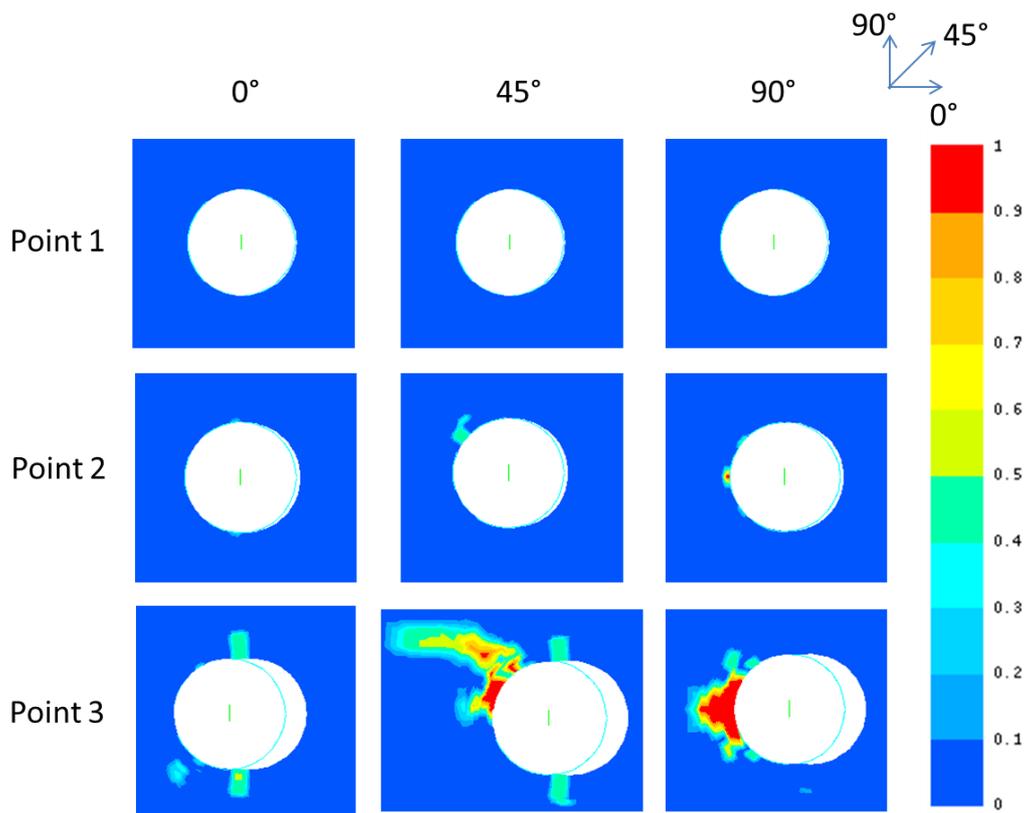


Figure 7: Damaging in tension of the fibers oriented at 0° 45° and 90°

4.2 Single lap joint

The load displacement curve obtained during a tensile test on SSLJ specimen is compared with the one taken from finite element analysis (Fig. 8). The initial stiffness of the 2.5D FEM is equal to the experimental one when it is computed after an unload-reload cycle at the beginning of the test. Because of convergence issues the 2.5D FEM is not able to give results after point 5. Despite the gap between initial stiffness the 2.5D FEM curve seems similar to the experimental one. A damage scenario could then be elaborated based on numerical results.

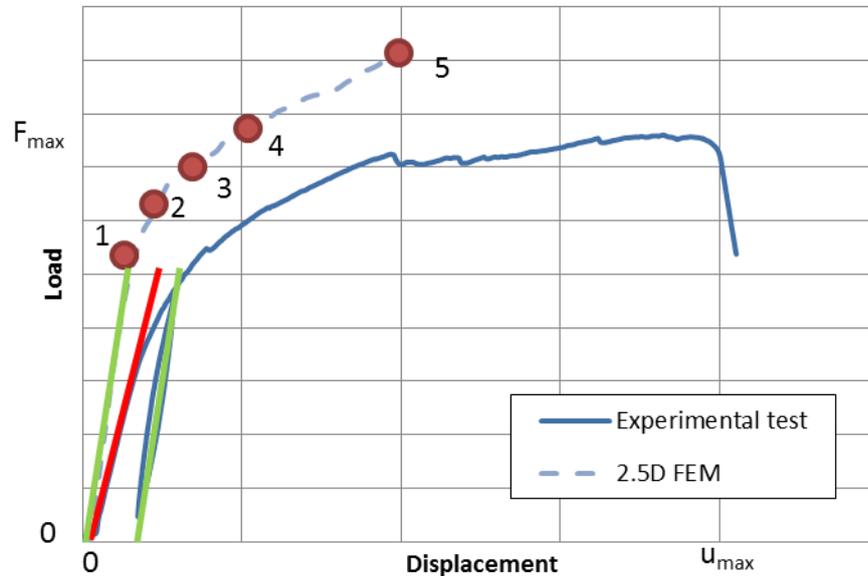


Figure 8: Experimental and numerical load-displacement curves for SSLJ tests

Matrix is damaging for a very small displacement in the finite element analysis. However the global behavior of the structure is still linear. Then aluminum starts to plasticize at point 1. This is the first joint stiffness loss. Damage in compression and in tension are presented Fig. 9 and Fig. 10. Even if the gap between the medium plane of the ply and the medium plane of the element is taken into account, plies of the same orientation behave very similarly. At point 2, damage of the fibers in compression appears simultaneously in the four orientations of the laminate. It results in a new joint stiffness loss. Then fibers of $\pm 45^\circ$ and 90° plies damage in tension at point 3 and stiffness drops again. Finally 0° plies damage due to high tensile stress at point 4. The composite material progressively fails, degrading continuously the joint stiffness until the end of the computation. At point 5 45° plies are failing with the same pattern as 0° ones. It is due to the tension failure of the plies oriented at 0° in the vicinity of the hole. They cannot transmit the load anymore thus the $\pm 45^\circ$ plies are shearing until their failure in this zone. This phenomenon happens probably because delamination is not taken into account here.

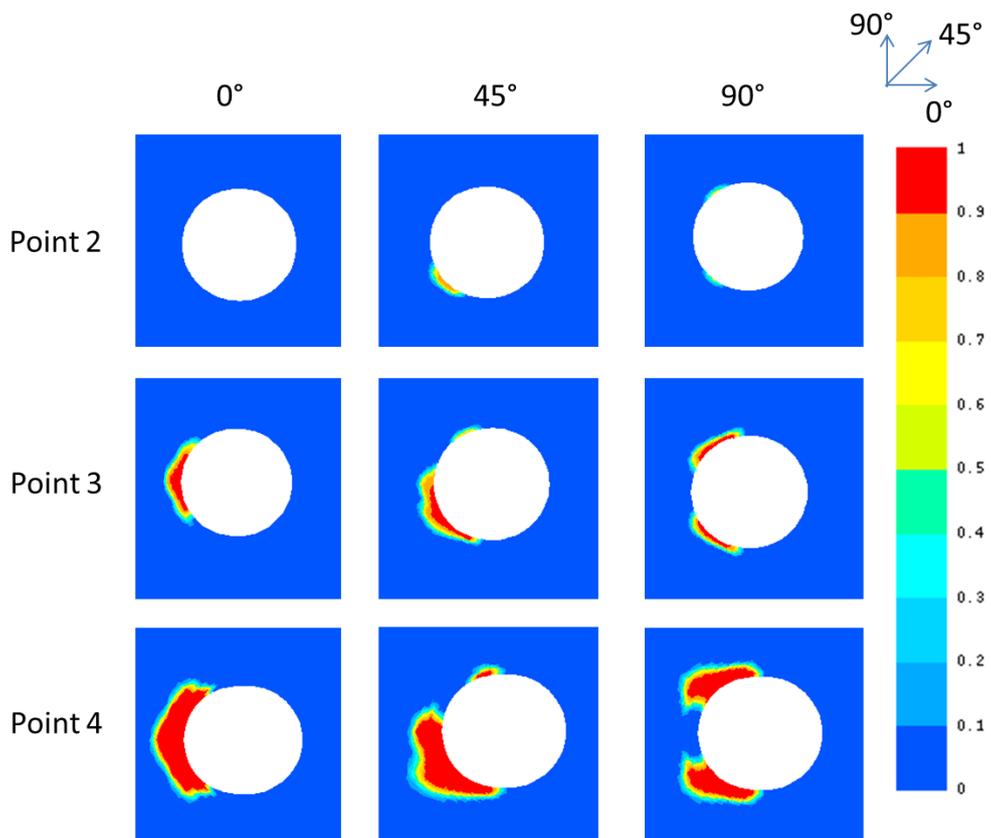


Figure 9: Fibers damaging due to compression in 2.5D FEM

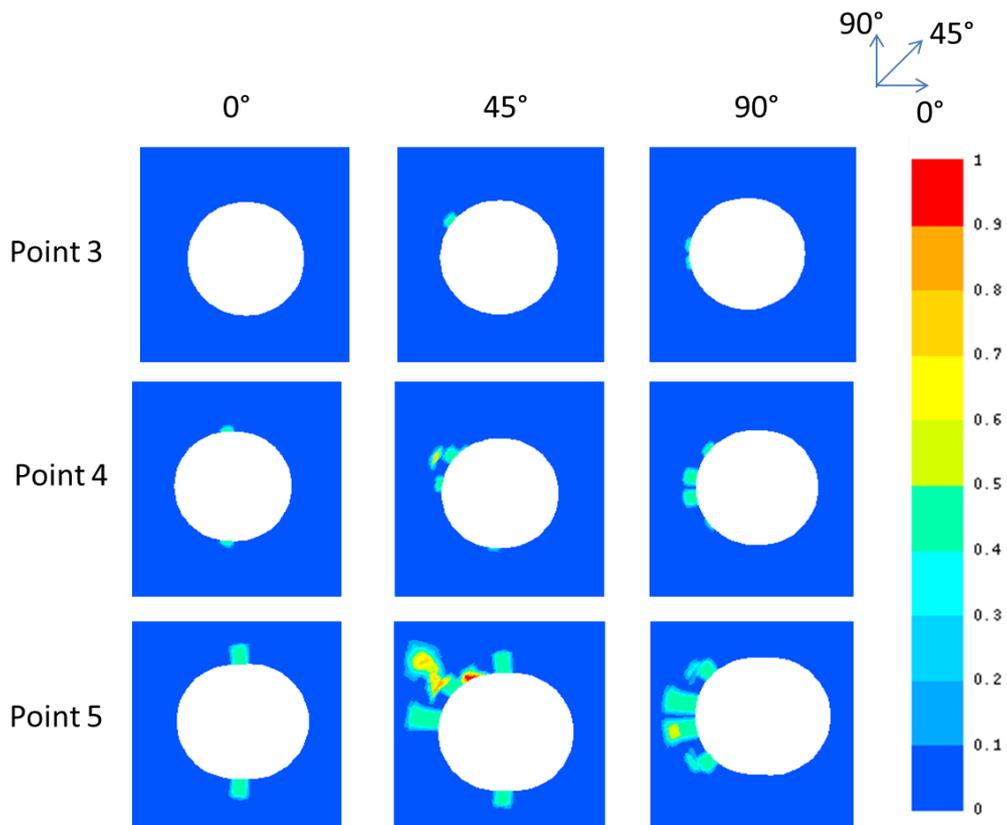


Figure 10: Fibers damage due to tension in 2.5D FEM

5 CONCLUSIONS

FE models have been developed to simulate double lap joint and supported single lap joint tests. Nonlinear behavior law has been implemented on SAMCEF in order to take into account the damage of the composite material. Different failure modes have been studied. The first stiffness loss is due to fibers failure under compression in the four directions of the laminate for both tests. This phenomenon appears simultaneously in all plies according to the FE model. Then this damage progress from element to element behind the fastener. After that other elements damage due to high tensile stresses. It appears in all plies at the same time once again. This modelling based on composite shell elements permits to simulate with a good accuracy tests on composite bolted joint. An in-plane shell model is a good way to model DLJ specimens. Using the nonlinear behavior of composite materials it permits to correctly predict the yield strength of the structure. Then the damaging scenario still needs to be validated thanks to additional experimental tests. Concerning SSLJ even if the behavior of the model is quite similar to the experimental one it is not accurate enough to correctly predict the failure of the joint. The stiffness of the 2.5D FEM is higher than the experimental one, probably because of the shell modelling. Here the contact between the plates and the fastener is concentrated on two circular edges whereas it happens on a surface in the experimental test. Convergence troubles still persist and improvements need to be done. In this model only intralaminar phenomena are taken into account. However important delaminations are observed on the failed specimens. Including this interlaminar behavior in a 3D model could be a way to improve the simulation and elaborate better damaging scenarios. Moreover a 3D model is a better way to model the contact between the plates and the bolt. But this gain of accuracy would be done at the cost of an increase of computation duration. Finally interrupted experimental tests on DLJ and SLJ will be lead in order to validate or not the scenarios elaborated in this paper.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the French DGAC for financially supporting this study.

REFERENCES

- [1] Hart-Smith, "Mechanically-fastened joints for advanced composites," *Fibrous Compos. Struct. Des.*, pp. 543–574, 1980.
- [2] L. J. Hart-Smith, "Design and analysis of bolted and riveted joints," in *Recent Advances in Structural Joints and Repairs for Composite Materials*, 2003, pp. 211–254.
- [3] B. Montagne, F. Lachaud, E. Paroissien, and D. Martini, "Failure analysis of composite bolted joints by an experimental and a numerical approach," presented at the 18th European Conference on Composite Materials (ECCM18), Athens, Greece, 2018.
- [4] A. Matzenmiller, J. Lubliner, and R. Taylor, "A constitutive model for anisotropic damage in fiber-composites," *Mech. Mater.*, no. 20, pp. 125–152, 1995.
- [5] M. Ilyas, C. Espinosa, F. Lachaud, L. Michel, and M. Salaün, "Modelling aeronautical composite laminates behaviour under impact using a saturation damage and delamination continuous material model," *Key Eng. Mater.*, vol. 452–453, no. 2011, pp. 369–372.
- [6] G. Gohorianu, "Composite bolted joints behaviour : effects of hole machining defects," presented at the Joining Plastics, London, UK, 2006, pp. 65–77.
- [7] C. Espinosa, F. Lachaud, L. Michel, and R. Piquet, "Impact on composite aircraft fuselages. Modeling strategies for predicting the residual strength.," *Rev. Compos. Matér. Avancés*, vol. 24/3, pp. 271–318, 2014.
- [8] J. Xiao, B. Gama, and J. Gillespie, "Progressive damage and delamination in plain weave S-2 glass/SC-15 composites under quasi static punch shear loading," *Compos. Struct.*, no. 78, pp. 182–196, 2007.
- [9] O. Allix and J.-F. Deü, "Delayed damage modelling for fracture prediction of laminated composites under dynamic loading," *Eng. Trans.*, vol. 45, no. 1, pp. 29–46, 1997.
- [10] T. . Collings, "The strength of bolted joints in multi directional cfrp laminates," *Composites*, vol. 8, no. 1, pp. 43–55, Jan. 1977.

