Escola de Camins
Escola Tècnica Superior d'Enginyeria de Camins, Canals i Ports
UPC BARCELONATECH

SIMULACIÓ NUMÈRICA DE LAMINATS COMPOSTOS SOTMESOS A CÀRREGUES CÍCLIQUES

Treball realitzat per:
Marc Martinez Gomez

Dirigit per:
Xavier Martinez Garcia
Sergio Horacio Oller Martinez

Grau en:
Enginyeria Civil

Barcelona, 17/06/2014

Departament de Resistència de Materials i Estructures a l'Enginyeria

TREBALL FINAL DE GRAU

ABSTRACT

The present work has been focused on the numerical simulation of the composite materials in both linear and non linear regions. Several experimental tests provided by the literature have been simulated using the PLCD(CIMNE,2008) finite element code, a code developed by the RMEE (Department of Structures and Strength of Materials, UPC) and CIMNE(International Center for Numerical Methods in Engineering) which allows the representation of both material and kinematic nonlinearities.

The work starts with a brief introduction about the composite material, its heterogeneity and how the different components of the composite are able to interact. The research was focused on the mechanical failures which usually take place in a composite, the delamination process, as well as the effect of the cyclic loads related with the fatigue analysis. The next part of the work relies on the PLCD finite element code, how the numerical composite models are generated and the different implementations in the code. Special attention has been placed on describing the Serial/Parallel mixing theory and its scope to deal with the composite delamination numerical simulation. Equilibrium and conditions are imposed by considering an iso-strain behavior through the parallel fiber direction and an iso-stress behavior along the remainder directions. This condition, in addition to the fact that the effects of each component in the overall composite is taken into account by its volume content allow us to describe numerically the natural behavior found in a composite delaminated material.

An iterative learning process has been carried out in order to get the enough knowledge to deal with the simulation of the experimental tests provided by the literature. The first analysis has been made with the aim of gaining some experience regarding the mesh size required to properly analyze an structure, and the differences obtained with different meshes in terms of numerical results and computational costs.

Afterwards a new simulation has been conducted in order to learn how to simulate composite laminates in the linear and non-linear range, as well as to fully understand the performance of the serial/parallel mixing theory. Finally, the third case considered consisted in the non-linear simulation of a sandwich material. This analysis was intended to be conducted under static and cyclic loading. Unfortunately, the complexity of the problem has not allowed to reach the final stage of the project which has the cyclic analysis of the composite.

Despite being unable to reach the last goal of this project (cyclic analysis of a sandwich material), the simulations conducted and the results analyzed have allowed to obtain an excellent understanding on the mechanical performance of composite materials, the complexities associated with material non-linearities, and the requirements of non-linear finite element methods. This work presents all these simulations as well as the conclusions obtained from each one of them.

Hence, the SP mixing theory, in addition with an appropriate constitutive equation, has showed that the approach is perfectly capable to simulate the delamination process and predict the material non-linear behavior of the composite components.

TABLE OF CONTENTS

CHAPTER 1. INTRODUCTION TO COMPOSITES **5**

1.1 INTRODUCTION 5

1.2 COMPOSITE MATERIALS 6

1.2.1 FIBRES 6

1.2.2 MATRIX 7

CHAPTER 2. FATIGUE BEHAVIOUR AND MECHANICAL FAILURE IN COMPOSITE **8**

2.1 FATIGUE IN COMPOSITES 8

2.2 DEGRADATION CHARACTERIZACION 9

2.3 MECHANICAL FAILURE 10

2.4 DELAMINATION 11

CHAPTER 3.CALCULATION METHOD **13**

3.1 FINITE ELEMENT METHOD (FEM) 13

3.2 PLCD 14

3.3 PREPROCESS AND POSTPROCESS 15

3.4 COMPOSITE SIMULATION 16

3.4.1 SP MIXING THEORY 16

3.4.3 ALGORYTHM SCOPE. 17

CHAPTER 4. MESH ANALYSIS **20**

4.1 STEEL BEAM EXAMPLE 20

4.2 CROSS SECTION ANALYSIS 21

4.3 LONGITUDINAL ANALYSIS 22

4.4 NUMERICALS RESULTS IN TERMS OF STRESSES AND DISPLACEMENTS 23

4.5 QUADRATIC VS LINEAR AND CONCLUSIONS 25

CHAPTER 5. SIMULATION OF A DELAMINATION COMPOSITE BY USING THE SERIAL/PARALLEL MIXING THEORY **26**

5.1 AIM OF THE PROCEDURE 26

5.2 ISOTROPIC CONTINUUM DAMAGE FORMULATION 26

5.3 DEFINITION OF MATERIALS AND GEOMETRY 27

5.4 MESH DEFINITION 29

5.5 RESULTS 29

5.5.1 FORCE-DISPLACEMENT GRAPH 30

5.5.2 CRACK PROPAGATION 31

5.6 CONCLUSION 34

CHAPTER 6.COMPOSITE-SANDWICH FOAM-LAMINATED GLASS/EPOXY UNDER SOLICITATION STATIC AND FATIGUE 36

6.1 DESCRIPTION OF THE PROBLEM 36

6.2 FIRST MODEL. DESCRIPTION ACCORDING WITH THE INFORMATION PROVIDED........ 38

6.3 SECOND MODEL. ELASTIC MATERIAL TO AVOID THE LOCAL EFFECTS OF THE LOAD APPLIED 41

6.4 THIRD MODEL. LONGER AND MORE FLEXIBLE SPECIMEN. 46

6.5 CONCLUSIONS 49

CHAPTER 7. CONCLUSIONS AND FURTHER STUDIES 51

REFERENCES 54

CHAPTER 1. INTRODUCTION TO COMPOSITES

1.1 INTRODUCTION

Throughout human history, the materials used by the engineers have been several and always aimed to satisfy a necessity in the more efficient way. While in civil engineering the concrete has widely been the more important due to its fullness and economy, some other industrial sectors designed with other kind of lightweight materials which allow more specific designs.

Even though the materials were used separately from the beginning, people slowly realize that in order to solve some deficiencies, could be useful consider them together seeking to improve specific features. Materials with different properties were as a first time together, allowing getting such performance which seemed impossible to obtain before. Bit by bit, in a continuous process, a globalization phenomenon took place, inducing that some materials more proper for making little designs started to be considered as a solution at big scale. Is in this background where the composite materials started to make sense

Despite the fact that at the beginning the use of such materials was mainly leaded to the industry and aircraft sector, due to a decrease in their production costs they started to be used in all kind of products, specially automotive field, ship industry and finally, in the last years and as an specific manner, also in civil engineering as an alternative solution to concrete and steel.

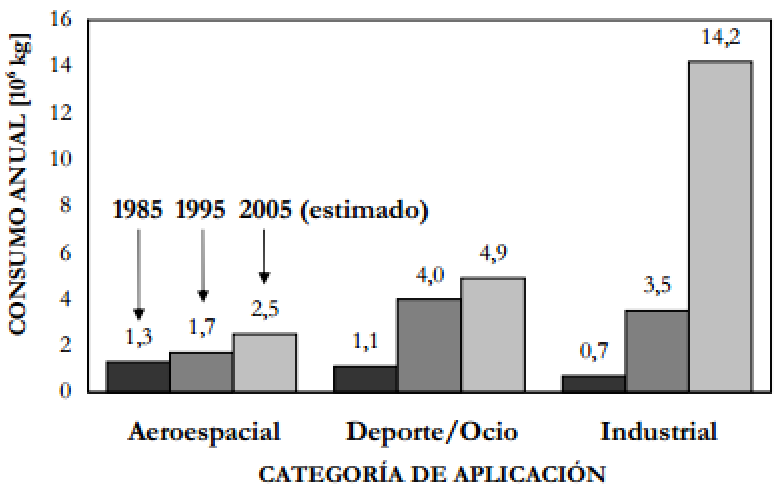

Fig. 1 Distribution of the application of the different composite materials between the different fields. Ref. 3

1.2 COMPOSITE MATERIALS

As a composite material we understand that material generated from the combination of others allowing getting specific features, both mechanical and structural which none of them have by their own. The reinforced concrete might be a clear example of composite material.

Within them, we'll find different phases. One known as fibre, from which the material will obtain the strength and stiffness properties and, on the other side, a phase known as matrix, whose purpose is acting like binding material, giving unity and consistency, allowing the fibres to transfer the efforts to each other. As a result of a material which so many different features, we will have a complex material characterised by its heterogeneous and anisotropic behaviour. However, the combination of different matrix and several fibres together will allow obtaining a material with specific features which are actually fitted to any sought solution. Besides, in a composite material we have different contributions from the different phases and different configurations, therefore, the final attribute of a laminate will depend on its components , as well as the fibres 's configuration and orientation.

Among its characteristics, the ones more valued are both high stiffness/weight and strength/weight ratios, really important when we are looking for solutions.

1.2.1 FIBRES

The fibres are the responsible for giving to the composite its structural properties, as well as stiffness and strength in the longitudinal direction. Among the main used we find fibreglass, carbon fibres and synthetic fibres like the Aramida

Material	Densidad	Módulo elástico	Resistencia tracción	Alarga-miento	Coef. Poisson	Módulo específico	Resist. específica
	ρ, [g/cm³]	E,[GPa]	S_t [MPa]	[%]	ν	[E/ρ]	[S_t/ρ]
Fibra vidrio							
E-Glass	2,54	72,4	3450	4,8	0,20	28,5	1,36
S- Glass	2,49	86,9	4300	5,0	0,22	34,9	1,73
Fibra carbono							
AS-1 (*)	1.80	228,0	3100	1.32	—	126,7	1,72
AS-4 (*)	1,80	248,0	4070	1.65	0,20	137,8	2,26
IM-7 (*)	1,78	301,0	5310	1,81	0,20	169,1	2,98
P-100 (‡)	2.15	758,0	2410	0,32	0,20	352,5	1,12
T-40 (‡)	1.81	290,0	5650	1,80	—	160,2	3,12
T-300 (‡)	1.76	231,0	3650	1,40	0.20	131,3	2,07
Fibra boro	2.70	393,0	3100	0,8	0,20	145,6	1,15
Fibra aramida					—		
Kevlar 49 (†)	1.45	131,0	3620	2,8	0,35	90,3	2,50
Kevlar 149(†)	1.47	179,0	3450	1,9	—	121,8	2,35

Table 1 Most common fibres used as a component of a composite material, responsible for the structural properties

1.2.2 MATRIX

The matrix acts like blinding material, allowing to work together the different fibres. Normally, as the more used, we find lightweight metals and polymers, or also resins. We must be aware of those last have certain limitations when they are used under high temperatures, like loosing of resistance.

Material	Densidad ρ, [g/cm^3]	Módulo elástico E,[GPa]	Resistencia tracción S_t [MPa]	Alarga-miento [%]	Coef. Poisson ν	Módulo específico $[E/\rho]$	Resist. específica $[S_t/\rho]$
Termoplásticos							
PEEK (*)	1,30 – 1,32	3,24	100,0	50	0.4	2,5	76,3
PPS (‡)	1,36	3,3	82,7	4	—	2,4	60,8
Termoestables							
Poliéster	1,1 – 1,4	2,1 – 3,4	34,5 – 103,5	1 – 5	—	2,2	55,2
Epoxy	1,38	4,6	58,6	—	0,36	5,5	74,0
Poliamida	1.46	3,5	103,0	—	0,35	2,4	70,5

Table 2 Different Matrix used as a component of a composite material, acting like a blinding material

Thus, the composites materials allow us developing some news solutions in which we are seeking to strengthen the resistance of the composite along different directions, where the fibres will be placed. Even thought the response of the composite to efforts applied through those directions will be really good, along the other directions the strength will be defined by the matrix material. So for a specific problem under a specific load state, we must handle which directions are important to reinforce, as well as ensuring that the matrix and the fibres are working in consistence together.

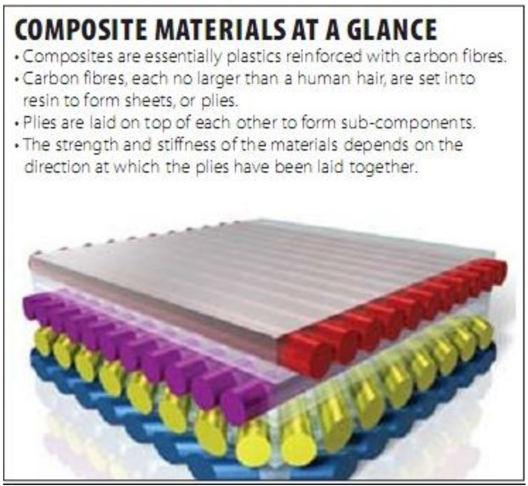

Fig. 2 Description of a composite material reinforced with carbon fibres along different directions. Ref 6.

CHAPTER 2. FATIGUE BEHAVIOUR AND MECHANICAL FAILURE IN COMPOSITES

2.1 FATIGUE IN COMPOSITES

As in whatever material, many of the composite elements are under cyclic solicitation during its service life. These loads, variable along time, induce a decrease in the material's mechanical properties which could mean the collapse of the structure at some load levels pretty lower than these cases where the structure is under static steady loads. This behavior is known as fatigue behavior of the material and it's obviously really important to study.

Within composites, elements with high resistance along its fibres direction, the fatigue is one of the main causes which could mean the fail of the material. It is more, a high percentage of the fails registered in experimental tests has been due to this phenomenon. That is the reason why in this work we are going to try to simulate the behavior of such material when it's under this kind of solicitation.

When a laminate is under this kind of cyclic loads state, two mainly main events take place : On the one hand, the failure might take place when a specific number of cycles is reached, all this with some load values smaller than in the static case. On the other hand, what could happen is the sudden decrease of the material stiffness along these cycles which might anyway induce the failure as well. Like we said before, the composites are heterogeneous materials. This same missing of homogeneity usually generates some inner stresses states as complex that are a significant obstacle to reproduce the consequence of the fatigue in composites and, moreover, it's even more difficult being able to study them by using a numerical model which can actually reproduce this behavior.

The answer of a composite under cyclic loads not only exclusively depends on the level of stresses reached, but rather also on the cyclic tension. As a general way, it is thought that the more important parameters to define a fatigue state are the maximum and minimum stresses values(σ_{max} and σ_{min}) and their ratio known as reversal index R:

$$R = \frac{\sigma max}{\sigma \min} \tag{1}$$

Another way to identify a cyclic stress is by the mean stress σ_{mean} and alternate stress σ_{alt}

$$\mathrm{\sigma mean} = \frac{1}{2} * \sigma max + \frac{1}{2} * \sigma min \tag{2}$$

$$\mathrm{\sigma min} = \frac{1}{2} * \sigma max - \frac{1}{2} * \sigma min \tag{3}$$

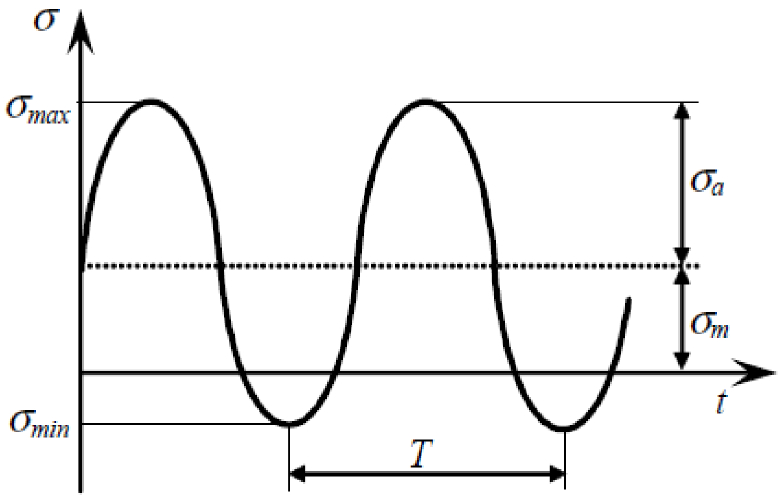

Fig. 3 Characterization of a cyclic tension. Ref 3

The R parameter or reversal index allows us characterizing the kind of tension taking place. Values of R between $-\infty$ and 0 states a tensile- compression(T-C), a value of -1 is a symmetric oscillating load. Values between 0 and +1 state tensile-tensile (T-T) and finally values between $+\infty$and +1 describe C-C stresses. It's definitely a really important parameter due to the fact that the behavior of a material relies on the value of it. Even though in homogeneous materials the compression are not inducers of fatigue, in composites is different because of the fact that there are different phases with different elastic features, so the fractures keep propagating under compressive states. Nevertheless, in a general manner and according with reference 3, laminates with fatigue T-C tend to have more damage than T-T.

2.2 DEGRADATION CHARACTERIZACION

The most usual way to describe the behavior of a composite material under cyclic loads is the S-N curve(Stress- Life expectancy curve) This curve represents, in semi-logarithmic plane stress-log(N) the maximum stress applied and the number of cycles to produce the failure.

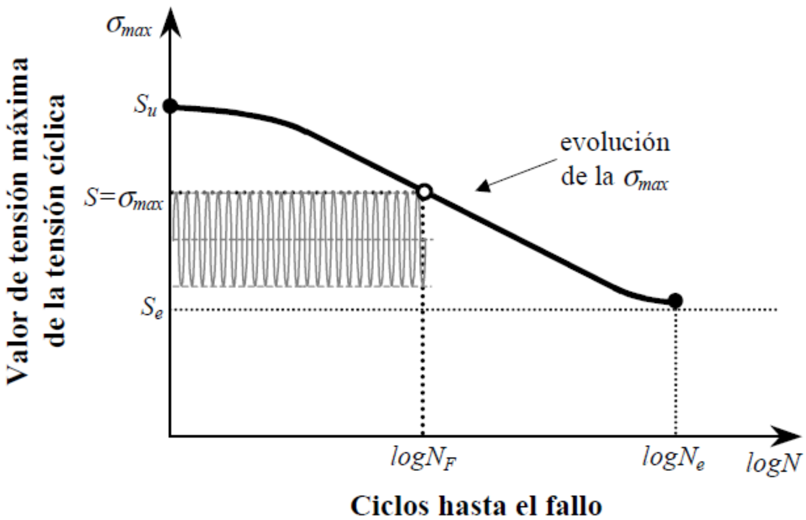

Fig. 4 The S-N curves show the drop of the resistance (S) from its static value (Su) until its fatigue limit (Se). For each level of stresses (S) states the medium level of the material life expected(N). Ref 3

The curve S-N is the first point used for the designer. In fact, nowadays most of the designs are still thought by using this tool. The problem is that it's is not giving us any information about the stiffness decrease, failure mechanism suffered, or changing in material features. Moreover, once the material and the orientation fibers are fixed, we have seen that the behavior under fatigue depends on the kind of stress applied.

Therefore, the information provided by only one S-N curve is not enough to describe the composite behavior. We would need a family of curves in the same plane as a function of their reversal index.

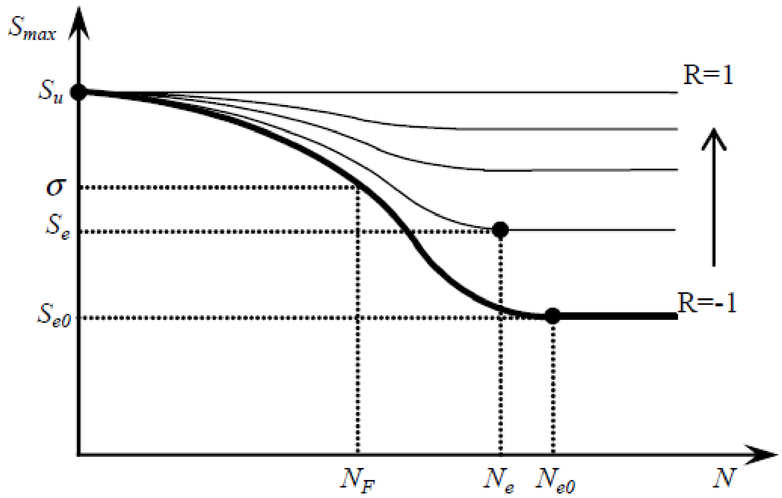

Fig. 5 Family of S-N curves for values of the reversal index (R) between 1 and -1. Ref 3

Thus, It is worth to say that these curves are completely defined for the metallic materials and, regarding the composite's sake, the topic is still being investigated. Some of these investigations consist on obtaining the S-N curves, while others seek to come up with the best mode to simulate the composite failure.

2.3 MECHANICAL FAILURE

The S-N curves don't provide us any information about which one will be the damage mechanism taking place, neither how is going to be the pattern followed by the fracture during its propagation until the failure. Thus, If we want to be able to analyze the behavior of a laminate under fatigue in a more wide way, we must know the main several phenomena observed.

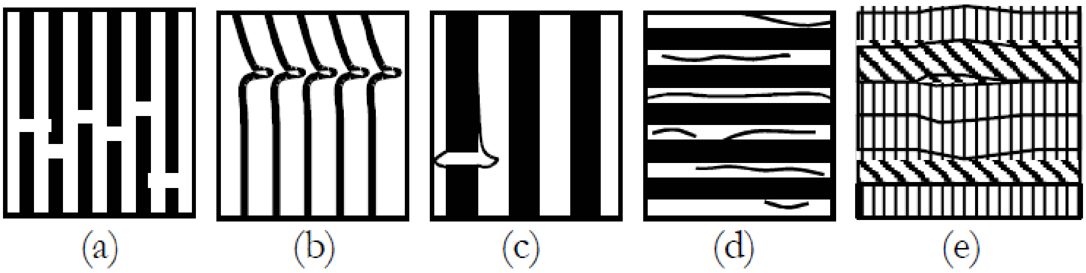

Fig. 6 Kind of mechanical failure in composites. A) Fibres breaking B) Fibre buckling due to compression effort C) Debonding D) Cracking of the matrix E) Delamination. Ref 3

In the composite study is necessary to separate the static case and the cyclic one, just because their features and the answer provided by the element are different. Nevertheless, the damages observed are pretty much the same. In composites, the mechanism of damage are common and in this case we are not just dealing with one fracture, but rather the interaction between each other induce a wide generation of fractures which produce that the damage not be located only in one point, instead distributing along the whole material. The relative importance of each one of these mechanism will depend, like we said, on the geometry of the element, material and on the load state applied.

2.4 DELAMINATION

One of the most frequent mechanism of damage and that one that will be studied further is the delamination process. The importance of this phenomenon is demonstrated by the amount of authors that have developed theories and formulations to deal with it.

Delamination is basically a process induced by the high heterogeneity of the material. The laminates are oriented in different directions so, under a specific stress state, the material cannot develop an uniform response, this means a loss of adherence between the layers. It can occur as a local point or in a global manner, but basically occurs by loss of stiffness in the matrix material produced by the shear stresses in it, avoiding the whole material to transmit the efforts. Is caused by a weakening of the bonds holding the layers together, meaning that and adhesive starts to break down. Since this happen, the material won’t necessarily show any signs of weakness, making its breakdown unexpected

All authors that have studied the problem agree that the delamination process is characterized by two main phenomenon, the crack initiation and its propagation along the composite. Although, most of them differ from the approach to deal with it. The high degree of heterogeneity found in the material leads to a several different procedures proposed in order to analyze it from a numerical point of view. As we will explain in the next points, we will use the Serial/Parallel mixing theory(developed by F.Rastellini) whose main feature is its simplicity, avoiding complicate techniques but, at the same time, testing its capacity to provide good and accurate enough results.

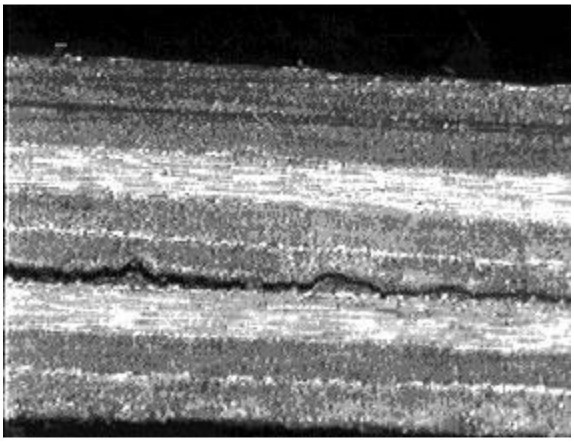

Fig. 7 Phenomenon of delamination taking place from overload or fatigue. Ref. 7 (Interlaminar Fracture Testing of Composites)

CHAPTER 3.CALCULATION METHOD

3.1 FINITE ELEMENT METHOD (FEM)

Like in many engineering problems, in composite materials it's also possible to reach some specific analytic solutions assuming several approximations. The problem is that these solutions are only valid either in simple load states or in those examples where the geometry is not complex. Due to the fact that there are not so many cases in which these requirements are satisfied, we must have other tools in order to deal with any general problem. Behind this background the numerical methods started to make sense since several years ago and nowadays are widely used to develop approximate but representative solutions.

The calculation method used in this work is supported by the FEM theory, a well known and one of the most used numerical technique for finding approximate solutions to boundary value problems for differential equations. By using this method we aim to split our domain in several elements defined by its nodal coordinates (mesh procedure). Then, imposing the characteristics of our sought function, the problem will be simplified to get the value of the function in the nodes and then, interpolate in the whole domain with that same function used before.

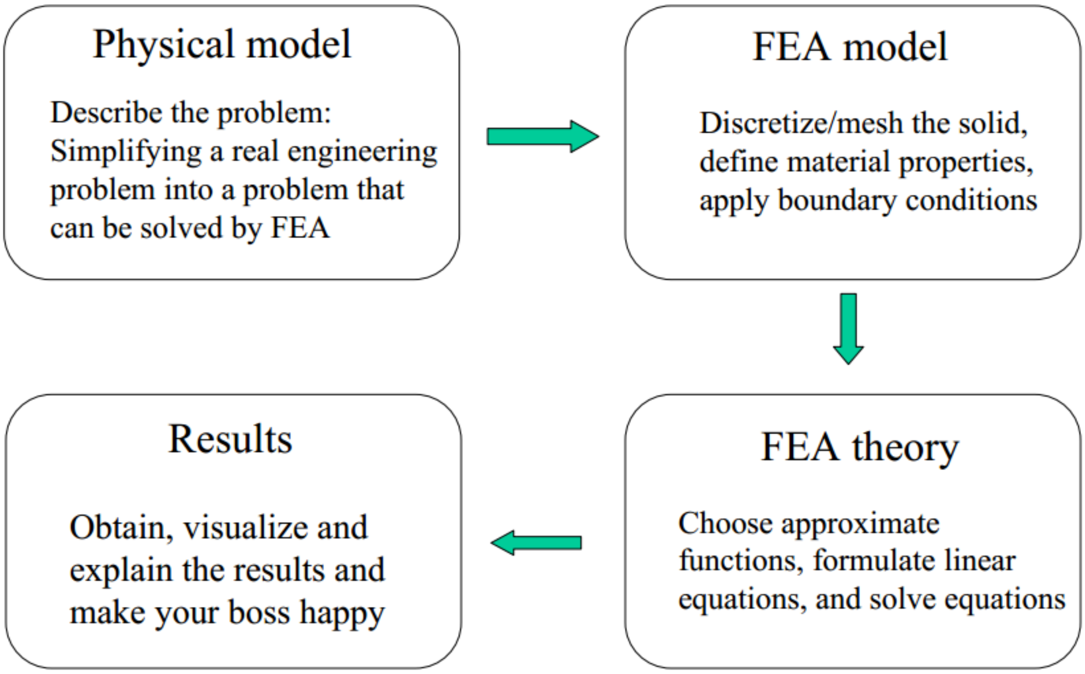

Fig. 8 General procedure description of the FEM approach. Ref. 6

The main advantages of the FEM over other numerical methods are that allow to represent any complex domain and, generally, their mesh procedure it's pretty intuitive and easy to follow. On the other hand, it also allows to capture the local effects, really important in complex problems such this one that we are going to study in this work. Generally, the use of this method will be followed by some other numerical techniques in order to solve the arisen system of equation or just in some dynamics solicitations in order to go forward along time, all this will depend on the complexity of the problem. In turn, like in any numerical method, our solution will be approximated, usually defined by the number of elements that we are going to use. We always could get closer to a better solution, but this will be limited by the computational cost that we actually want to assume. That's the reason why it is really important to have a wide knowledge in the method, being able to build effective meshes that allow us to reach accurate enough solutions. As we said, we will always have a constant struggle between the accuracy of our solution and the computational cost required.

3.2 PLCD

The main tool and the calculation program used to reach the solution of this work is the PLCD, a program developed by the RMEE (Department of Structures and Strength of Materials, UPC) and CIMNE(International Center for Numerical Methods in Engineering). This FE code allows to work with some several mechanic problem conditions, such as lineal or not lineal behaviour in both material and geometry, small or large deformations and thermal and thermal mechanic coupled analysis.

In the resolution of solid mechanic problems also allows to carry out nearly-static analysis and dynamics, and to represent some mechanism of damage and plastic analysis, really important when we are facing with some composite laminates problems that will be under fatigue solicitations.

It's worth noting that the code works with general anisotropy and relies on the several composite materials dealing, including the reinforced concrete, so important in civil engineering. It's also incorporates several methods to solve the system of equations, setting the Newmark method to reach solutions in dynamic problems. It uses various constitutive laws to predict the material behaviour (Elastic, visco-elastic, damage, damage plasticity) and uses different yield surface to handle and control their evolution(Von-Mises, Mohr-Coulomb Drucker-Prager, etc)

The program is currently in development, not fully extended but it has already won some important awards and acknowledgements in the engineering field due to its application to the numerical simulation of structures and material behaviours.

Among its main advantages we find that it can be adapted to any current computer software and it has an interface with some commercial software like GID , the one that we will use in this work. This means that we will be able to create our both geometry and mesh in a simple way, as well as introduce some input data that otherwise would require a wide knowledge in the specific code.

As far as we move on in this work, the approach followed by the code will be better explained. Even though the code can be used in any kind of conditions, it's important to say that it has been made to reach its full performance in 3D analysis, so we will always consider this 3D analysis despite being able to represent the same problem by some 2D simplifications which usually would reduce the computational cost. Therefore, the finite elements used to split our working domain will be hexahedral either lineal or quadratic, with its corresponding number of gauss integration point. So for each problem, according with the study of it, we will have to decide with is the more proper element to represent the fact in a consistent and efficient way.

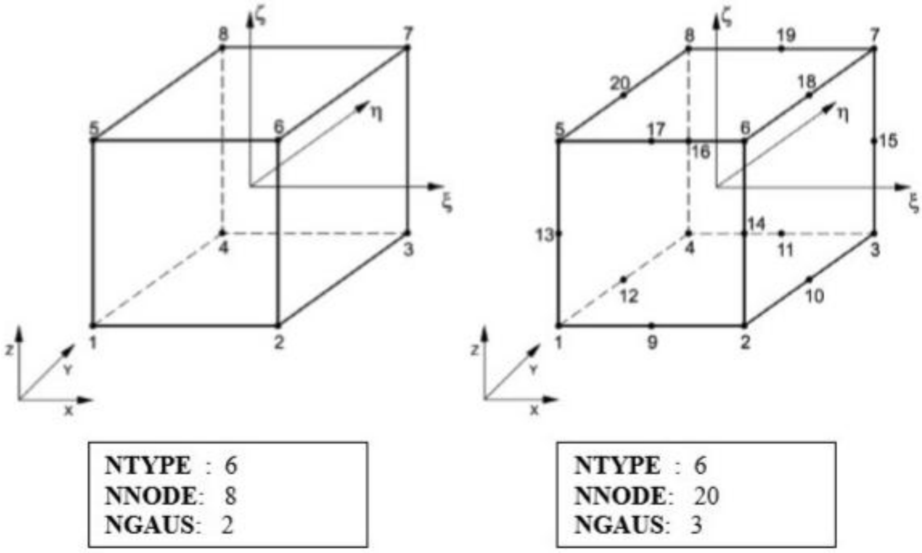

Fig. 9 Description of the linear and quadratic hexahedral elements, with their nodes and Gauss points implemented. Ref. 2

3.3 PREPROCESS AND POSTPROCESS

As we said above, one of the most important features of the PLCD code is that it has an interface with different commercial software, allowing to work in a more intuitive way when we have to face with the problem of making the geometry, building the mesh and reading the final results. Thus, during all the examples worked, we will be using the software GID, widely used in all kind of engineering problems.

Working with FEM, the building of the mesh is probably the point where we might have more problems if we don't perform it properly. We will be transforming a continuous volume to a discrete one, covering it with our finite elements. As a general rule to follow, we may think that the more regular and smaller the elements, the better and more accurate our solution will be. Indeed, that's true, but we must be aware that the efficiency and the computational cost are really important so we must try to reach accurate solutions using reasonable meshes. For instances, performing higher elements concentrations there where we are going to have important gradients in stresses and allowing to have less there where our stresses barely changes.

Nevertheless, we must take care of having too many different elements sizes, because otherwise we could have problems of distortion which could affect our results. In a general manner, we must try that the characteristic length of an element not be larger or smaller than the half of any of the others. It's what we call the spectrum purpose.

On the other hand, on the postprocessor step we will be able to read numerical values of the different variables which leads a specific problem, such as stresses, strains, displacements, or even intern variables which will allow us interpreting in our model complex phenomena like the reductions of stiffness in a material due to a fracture process.

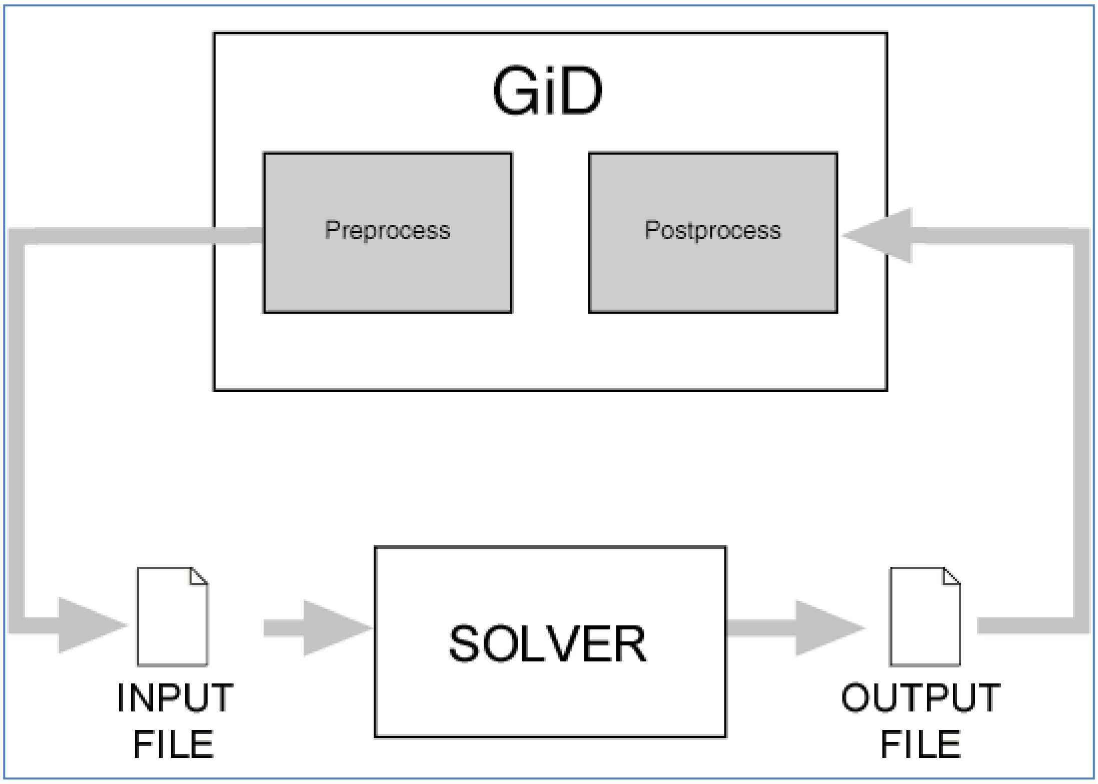

Fig. 10 Sketch of the working procedure. Gid will be used as a Preprocess and Postprocess and, as intermediate process, we will use the PLCD as a Solver. Obtained from Ref. 5

3.4 COMPOSITE SIMULATION

Like we have been saying, the composite nature itself is giving an heterogeneous component to the material, as well as a significant anisotropy. This results in several problems in order to define the global and effective properties (Tensile Modulus, strength, general behavior under a load) from the properties of the constituents.

Several approaches have been proposed in order to characterize the different behaviors of the composite. Most of them are related with the rule of mixtures and, in fact, the Classical mixing theory first formulated by Toupin(1960) has been widely used in several problems. The researcher of the university, as well as all the staff who is focused on the composite numerical simulation, have been using this theory, studying its strengths and limitations and finally being able to come up with an improvement of this same theory. This new theory is the Serial/Parallel mixing theory developed by F.Rastellini(2006).

3.4.1 SP MIXING THEORY

The SP mixing theory is a numerical implementation whose main purpose is to determine the constitutive behavior of the composite from the known constitutive law of the several components. It's arisen as an improvement of the Classical model, in which a parallel behavior in terms of iso-strain was imposed along all directions. The main problem of this was that the performance of the specimen is widely depending on the direction in which the load is applied.

Thus, the new SP model is imposing an iso-strain condition only along the fibres directions and an iso-stress conditions along the remainder directions.

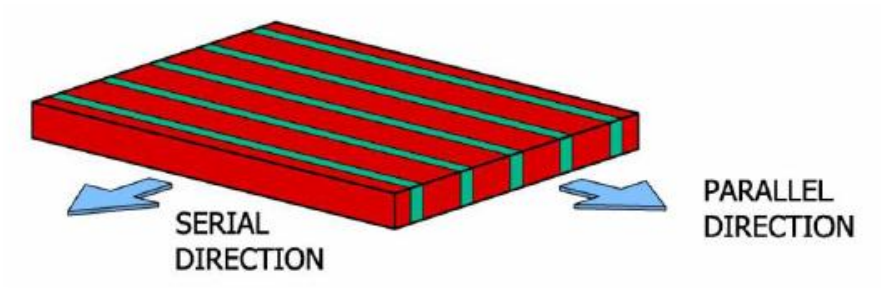

Fig. 11 The SP Mixing Theory imposes an iso-strain condition only along the fibers direction and an iso-stress condition along the remainder serial directions. Picture obtained from Ref. 11

Three main hypotheses must be satisfied in order to carry out the implementation of the numerical model within the PLCD code:

1. All the components of the composite, as well as the composite, are affected by the same strain in the parallel fiber direction.
2. All the components of the composite, as well as the composite, are affected by the same stress in the serial direction.
3. The performance of the composite is directly related with the volume fraction of the component materials.

So the equilibrium and compatibility equations imposed are the nexts:

$$\text{Parallel behavior :} \quad \begin{aligned} {}^{c}\varepsilon_P &= {}^{m}\varepsilon_P = {}^{f}\varepsilon_P \\ {}^{c}\sigma_P &= {}^{m}k\,{}^{m}\sigma_P + {}^{f}k\,{}^{f}\sigma_P \end{aligned} \tag{4}$$

$$\text{Serial behavior :} \quad \begin{aligned} {}^{c}\varepsilon_S &= {}^{m}k\,{}^{m}\varepsilon_S + {}^{f}k\,{}^{f}\varepsilon_S \\ {}^{c}\sigma_S &= {}^{m}\sigma_S = {}^{f}\sigma_S \end{aligned} \tag{5}$$

Where the superscripts ^{c}, ^{m} and ^{f} are respectively composite matrix and fiber and k represents the volume fraction of each component. The numerical implementation is only taking into account two components, fiber and matrix.

3.4.3 ALGORYTHM SCOPE.

The algorithm scope proposed to satisfy and met the compatibility equations is an iterative process which is handle it in terms of convergence.

1. Our FE code will bring us the Stress tensor at one step, this will be entered into the algorithm.

2. The first operation to perform is to split-up the strain tensor in two parts. The Paralel direction and the serial direction.

$$\varepsilon = \varepsilon_P + \varepsilon_S \qquad (6)$$

According with the equation 4, the parallel strain component is the same for both materials and for the composite. On the other hand, to know the serial strain of each component it is necessary to predict the expected strains in one of the components. This is obtained considering that the parallel and serial components of the total strains are distributed according with the composite stiffness obtained in the previous time step.

3. Once we know the strains of each component, by using their constitutive equation we are able to calculate the stress tensor of each one of them.

$$\varepsilon^f \rightarrow \mathit{Constitutive\ equation} \rightarrow \sigma^f \qquad (7)$$

$$\varepsilon^m \rightarrow \mathit{Constitutive\ equation} \rightarrow \sigma^m \qquad (8)$$

4. With the stresses of the components we are able to calculate the predicted serial tension and check if the equation 5 is satisfied. If the residual stress is smaller than the tolerance, the computed strains and stresses are considered correct and the calculation can move on. However, if the tolerance is got through, the initial prediction of the component stress tensor must be updated, and this correction is done with a Newton-Raphson scheme.

It's worth to notice that a damage formulation must also be implemented in order to update the possible changes in the effective stress of the components arisen from the damage found in them.

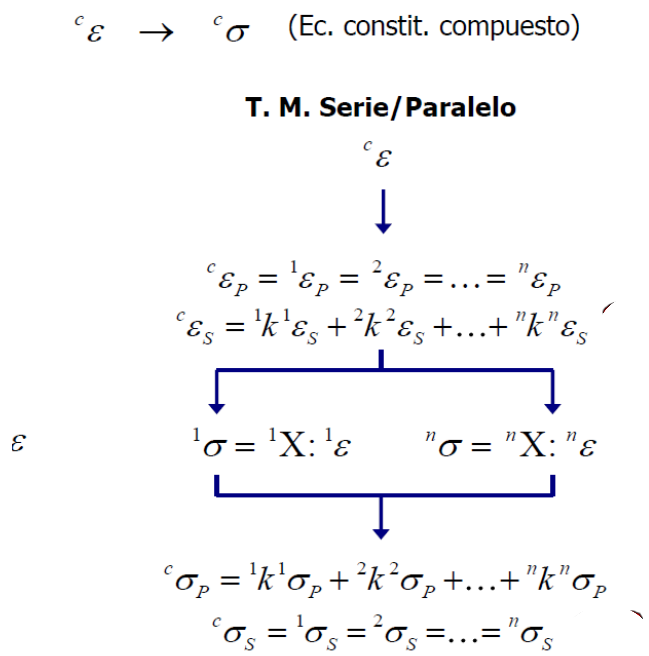

Fig. 12 Algorithm scope of the SP Mixing Theory. Ref.11

Hence, with this numerical tool provided by the SP mixing theory the phenomenon it's reproduced naturally, with no necessity of the definition of special tools or computationally expensive techniques. The mode of failure associated with the delamination can be perfectly simulated, as the final results of the work will show. The Matrix-failure, caused by the stress state found in it, results in a reduction of the stiffness and strength capacity of the composite in its serial direction. This same reduction provides a composite performance equivalent to that found in a delaminated natural material.

CHAPTER 4. MESH ANALYSIS

4.1 STEEL BEAM EXAMPLE

Even though the work seek to simulate the fatigue behavior in a laminated composite material, we thought that, in order to make an iterative learning process, it would be a good idea to start with a simpler problem in which we worked with a conventional homogeneous material such as the steel. This will help us to interpret and become familiar with some aspects of the code which are really in all the works, never mind the material. Those aspects could be the building of the mesh and which would be the more proper type of element to use, linear or quadratic hexahedral, for instance.

Hence, we came up with an example of a 2 meter spam beam made of steel. The beam would be simply supported and loaded with a punctual load at mid-span with a value of 1000KN. The cross section of the beam would be a square of 0.1m side. With this problem description and knowing the elastic properties of the steel as well as the analytic solutions provided by the material resistance's theory, we will be able to make some studies leaded to know which the more proper features of and specific mesh are.

Beam Properties	
Tensile Modulos	210000MPa
Span	2m
Inertia	8.33e-6 m^4
Analytic Displacement	9.5238 cm
Analytic Normal Stress	3000MPa

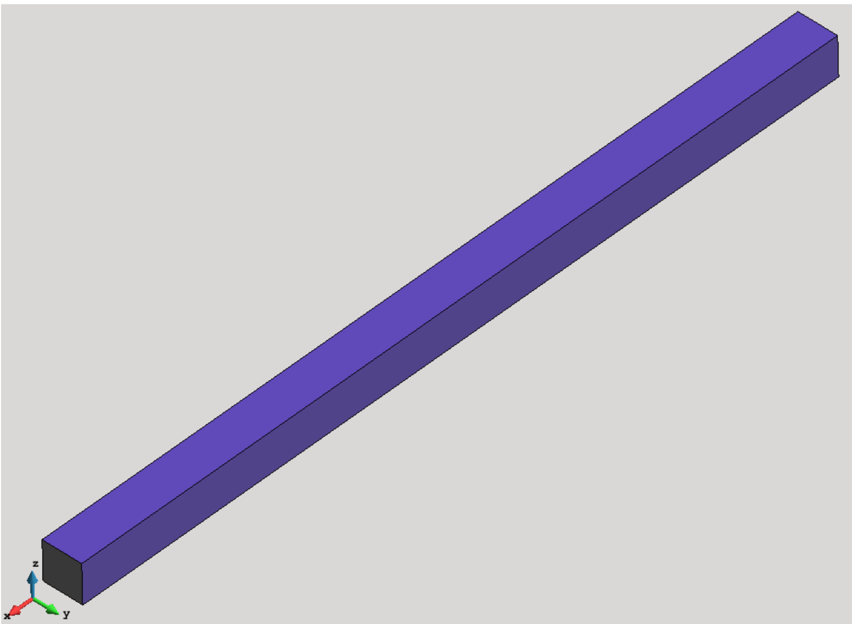

Fig. 13 Geometric representation of 2 m span steel beam

In order to build the mesh, regardless of we use lineal or quadratic elements, we must try to increase the mesh density in those points where we have changes in stresses. In this example, geometry conditions of such problem are completely regular. Same happen with the load. So we are sure that all the changes in tension will be due to the elastic normal stress general distribution along each cross section, as well as along the span of the beam. We will assume the assumption of the beam theory which says that in a vertical point of an specific cross sections the normal stress remain constant through the whole thickness. So following this, we have two main directions where our study must be focussed on and a third one where we may reduce the number of elements.

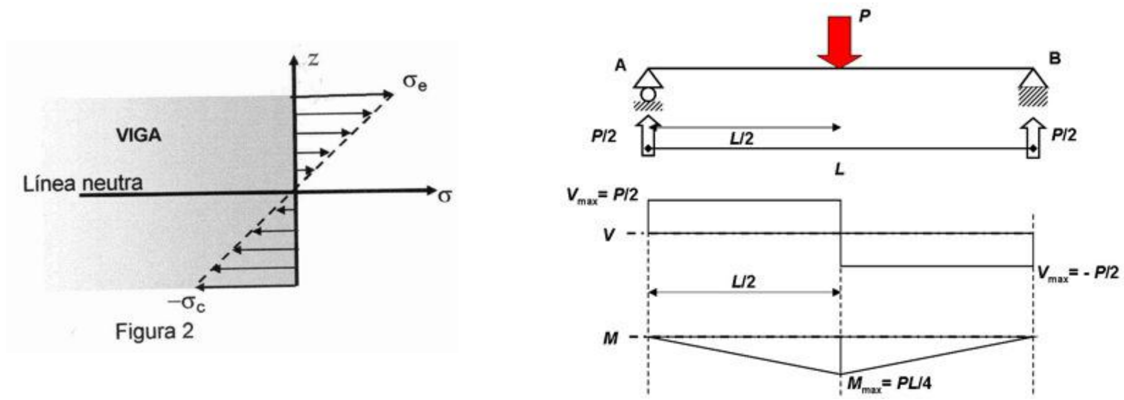

Fig. 14 Phenomenon which induces normal stresses in a double supported beam with a punctual load at mid span.

Let me remind that the solver provided by the PLCP code goes into the solution of the system of equations by using numerical and iterative processes. Thus, in a general problem, we define a proper number of loads times with it value and convergence condition at each time. This is really important in such examples in which we might find material non-linearities, like those one we will study later on. In this case, as we are dealing with an uniform and linear problem, we could even try to solve our system by using only one time step. However, in order to reproduce the deformation of the beam, we will solve the problem by adding a couple of extra steps.

4.2 CROSS SECTION ANALYSIS

As a first study, we will consider how our final solution converges due to changes in the number of elements along the cross section for both lineal and quadratic elements. It's worth noting that in order to do that, we previously fix the number of divisions along the other 2 lines. So we must not expect to get the analytic result, so we will already have an error associated to the other directions. In this case we fix 16x2 divisions when using quadratic and 64x2 using lineal elements. So the criteria followed to characterize the number of elements needed will be the coincidence between the final and the analytic result and also the convergence achieved. In other words, the number of elements beyond which the result starts not change.

Hence, the results can be summarized in the next picture:

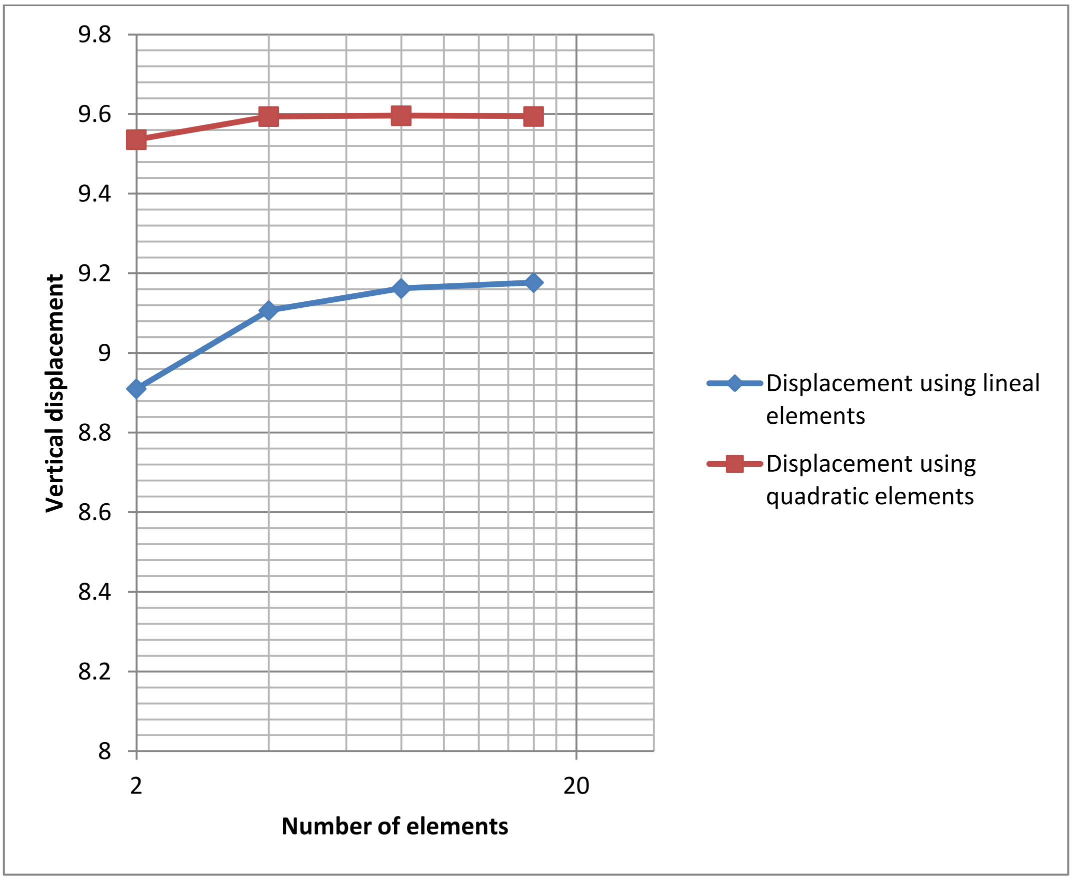

Fig. 15 Variation of the vertical displacement due to changes in number of element. Cross section analysis.

As we can see, the type of convergence in such case is pretty similar. The number of elements used is not affecting so much the results, specially when we go beyond 4 elements. From this point the displacements value using quadratic elements has already converged while the lineal one will do it a bit later. The utility of this information relies on the fact that we can define the number of elements to use in each case, both linear and quadratic elements.

4.3 LONGITUDINAL ANALYSIS

The next analysis will be aimed to characterize how changes in the number of divisions along the dominant length of the beam could change our final results. As before, the procedure followed was based on the same. Fixing the number of elements along the other two directions we can study the possible error committed by using a wrong number of elements along the longitudinal direction. Thus, the elements fixed in this case are 8x2 in both situations.

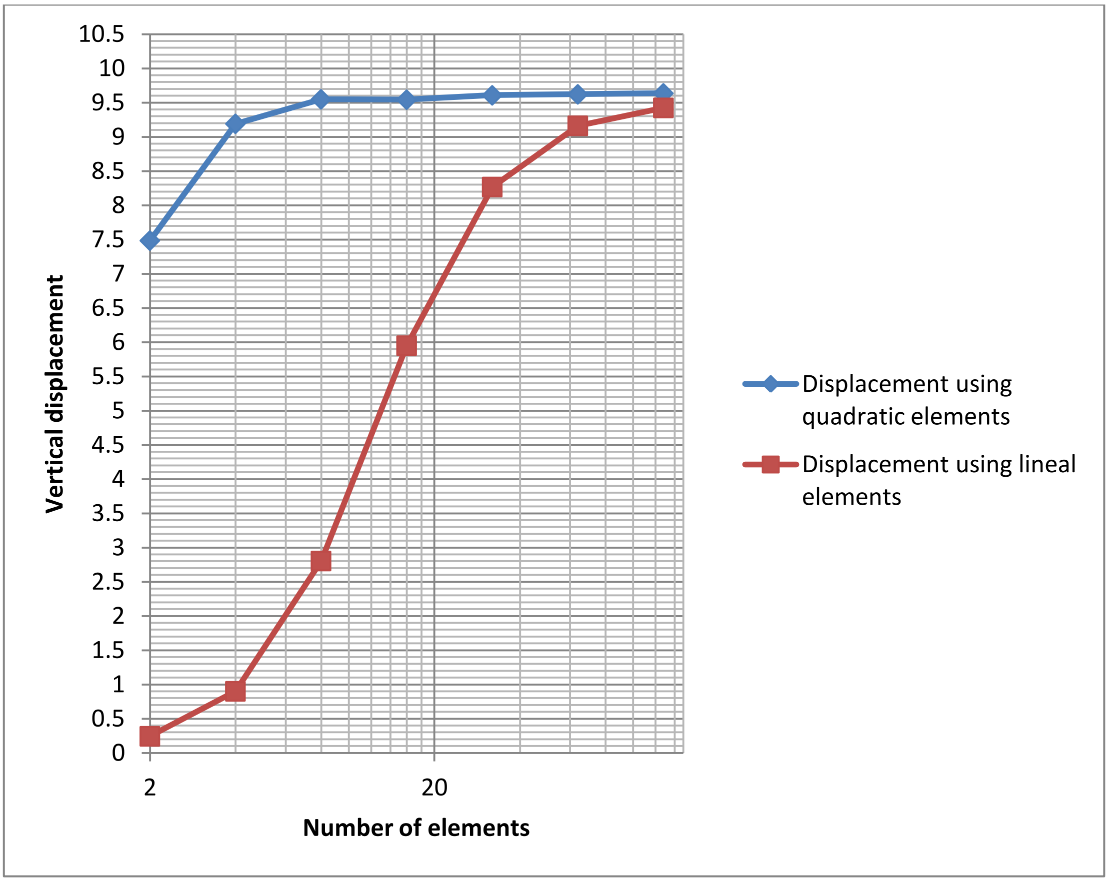

Fig. 16 Variation of the vertical displacement due to changes in number of elements. Longitudinal analysis

From the figure 16 , we can realize that in this case there is definitely a significant change in results, even huge when we consider just a few elements. This is because of we are performing the analysis along the dominant length so it's where our solution is more sensitive to change. Here is also important the type of element, not only the number. The quadratic experiences a fast and smooth convergence and this is achieved at 8 elements. On the side the lineal convergence is slower and rougher and it's not achieved until we select nearly of 128 elements.

4.4 NUMERICALS RESULTS IN TERMS OF STRESSES AND DISPLACEMENTS

Once we have studied the different effects of each one of the elements, we are in conditions to choose the proper mesh in order to get those results that we are interesting in. In this case we will be studying the stresses and the displacements.

The mesh selected was made with 512 quadratic elements, really well defined in both directions. The final displacement reached is the one expected by the analytic procedure and we can also trace the deformation of the beam.

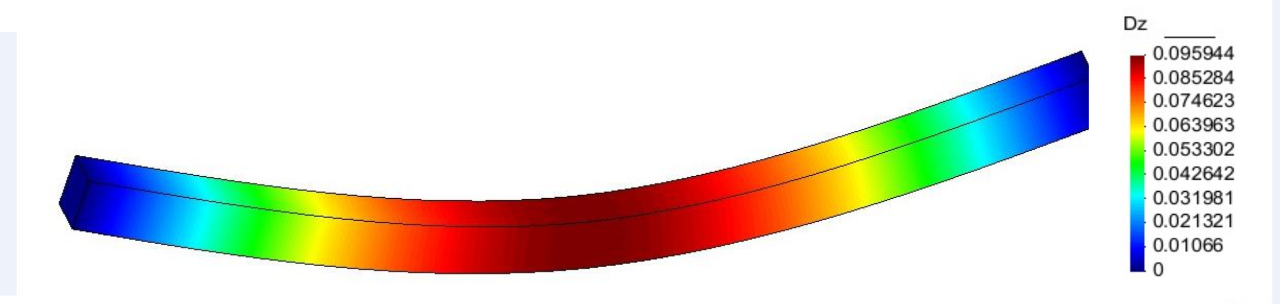

Fig. 17 Deformation and vertical displacement distribution of the 2m span beam. Values in meters

On the other hand, another important information for any bending problem is the distribution of the normal stresses as a result of the bending moment. As we know, this one will be maximum at mid span and, by using the analytics equation of the bending, we expected to reach a value for the normal stresses of 3000MPa. Proceeding with the calculations we got that same value along the cross section:

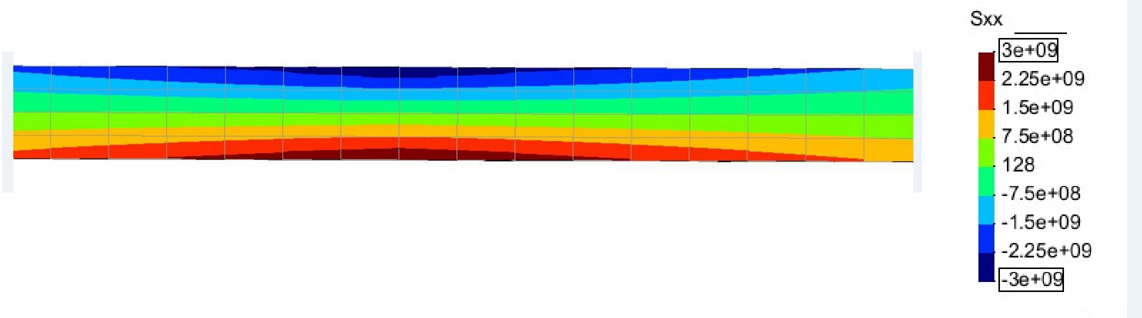

Fig. 18 Distribution of the Normal Stresses (Sxx) along the cross sections of the beam. Values in MPa

As we already knew, the distribution pattern followed in each one of the cross sections, is the classic lineal distribution showed in the figure 14. Both blocks of tension and compression are separated by the neutral line (or neutral surface if we take into account the whole beam). The tension block bellow and the compression above. There where we have the maximum moment (mid span) is where we also have the maximum normal stress(3000MPa).

It is worth noting that, just in the point where we applied the load, we were having stresses slightly higher than those values. This is because of the fact that in a beam of 2 m span with a really important local load, so we are indeed going to have local shear problems in those points loaded. If we wanted to go further trying to solve this problem, we could try to split the local load in two or more near points, rather than one.

4.5 QUADRATIC VS LINEAR AND CONCLUSIONS

As we saw, we will always be able to reach convergence, whether using lineal or quadratic elements, but ones will obviously need more elements to achieve it. The computational cost used it is not only reflected on the number of elements, but rather also on the element's sort. The quadratic element has 20 nodes rather than 8 of the lineal, so this will finally indeed generate much more nodes to the mesh even though the elements may be less. Moreover, the quadratic has associated more gauss integration points so this means that it will need more evaluations of the shape functions in order to define the matrix K, something that will increase even more the cost.

On the other hand, the quadratic element is able to represent better results, specially in tensions. This is because of the fact that in a finite element, the more integration points we have the better sorted they will be. So we will be taking into account the 'whole' element instead of just one part of it, problem which could happen by using the linear one with less integration points. So, as always, we are struggling between cost and accuracy. The engineer, with his decisions and analysis capacity, decides in which side we are.

It's also interesting to see that when we were using the quadratic element with a mesh well refined in both directions we obtained a displacement slightly higher than the predicted by the analytic solution. This might be due to the fact that this value has been obtained only considering the strain induced by the bending. In the usual beams where the span is pretty long we can assume that the strains due to shear are insignificant, but in such example, when the span of the beam is not long enough and it is loaded with a large punctual load, the shear might be significant and we will have to add both results.

CHAPTER 5. SIMULATION OF A DELAMINATION COMPOSITE BY USING THE SERIAL/PARALLEL MIXING THEORY

5.1 AIM OF THE PROCEDURE

Following the iterative-learning process from the previous subsection, in this part of the work we will be trying to simulate a delamination process taking place in a composite by using the Serial/Parallel theory described in sections above. The work analyzed in this section has been taken from a published work of the International Center of Numerical Methods in Engineering(CIMNE). The target of this section is, once finished, to get enough knowledge to reproduce any composite material simulation as that one selected from the literature to carry out the fatigue analysis.

As we explained before, the SP mixing theory is one of the simplest and computationally cheaper procedures to reproduce the actual behavior of a composite material where the delamination process is taking place. The process imposes an iso-strain relation along the fibre (parallel direction) and a iso-stress through the remaining directions (serial directions). In such manner, and also implementing a damage formulation to characterize the constitutive behavior of matrix component, we will be able to simulate the delamination from the beginning and propagation, without expensive techniques or complicates approaches.

To prove the ability of the formulation proposed to solve delamination problems, the End Notch Failure test is numerically simulated and the results obtained are compared with experimental ones.

5.2 ISOTROPIC CONTINUUM DAMAGE FORMULATION

If we want to simulate the degradation behavior of the matrix material as the result of a fracture process, we must define a damage formulation which takes into account the reduction of the effective strength area of the material by a reduction of its stiffness properties. In this example, we will be implementing an isotropic continuum damage formulation, which would be simulated by using internal variables.

Just as a general description, a damage process can be simulated in the context of continuum mechanics by the introduction of a material internal variable, M, representing the amount of damage found in it. This variable transforms the real stress tensor σ into an effective stress tensor σ0. Hence:

$$\sigma_0 = M^{-1} : \sigma \qquad (9)$$

Moreover, if the damage is isotropic, all the directions of the stress tensor are having the same damage. So we can define an internal variable d as a function of an scalar variable, becoming the damage equation as:

$$\boldsymbol{\sigma}_0 = [(1-d)\boldsymbol{I}]^{-1} : \boldsymbol{\sigma} = \frac{1}{(1-d)}\boldsymbol{\sigma} \tag{10}$$

When the material is not damaged, the value of the damage variable is 0 while, when the material is completely damaged, d=1. The effective stresses, shown in the previous equation 22, correspond to the stresses that would be obtained in the material if it is not damaged:

$$\boldsymbol{\sigma}_0 = \mathbb{C}_0 : \varepsilon \tag{11}$$

And the real stress tensor can be obtained, from the strain state of the material by coupling the previous equations:

$$\boldsymbol{\sigma} = (1-d)\boldsymbol{\sigma}_0 = (1-d)\mathbb{C}_0 : \varepsilon \tag{12}$$

For further information related with the damage criterion implemented in the simulation, the reference 12 can be visited.

5.3 DEFINITION OF MATERIALS AND GEOMETRY

To test the ability of such formulation in a delamination process, we will aim to simulate the End Notch Flexure(ENF)test, defined by *the European structural integrity society*(ESIS) and analyze it in terms of the experimental ones. The test seeks to obtain the toughness of a crack propagation, corresponding to a shear crack, in unidirectional fibre reinforced polymer composites(FRPC)

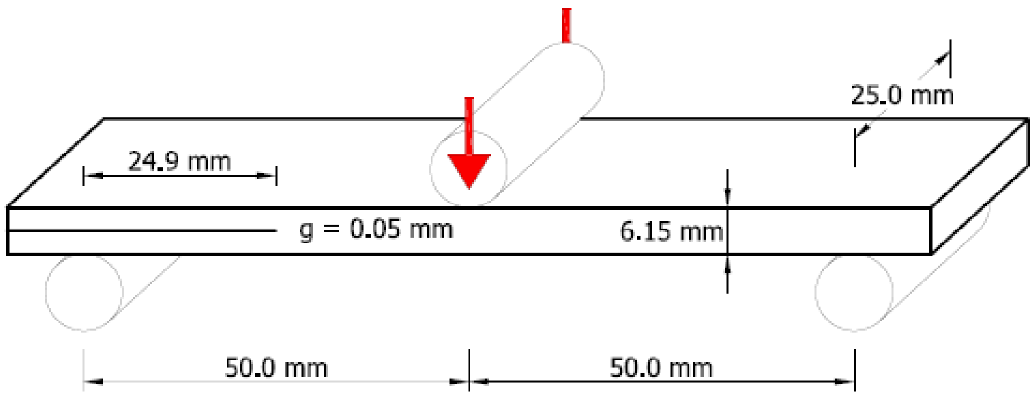

Fig. 19 Sample geometry used for the ENF test. Reference

The essay is made with a displacement controlled mechanism. The experimental essay applies a vertical displacement to the beam until the initial crack starts its propagation. The imposed displacement is kept it until the crack progression stops and the beam retrieves its linear behavior. At this time, the sample is unloaded. There are two results that are interesting: The force –displacement graph, which shows the structural performance of the beam, and also the final length of the crack. These two results obtained from the numerical model will be compared with the experimental.

The composite is known to be made of carbon fibres and an epoxy polymetric matrix from Hexcel composites. Regarding the simulation, two different materials have been defined. One is the composite material and another corresponds to the insert material. The composite is defined as an elastic material, while the matrix is characterized by a damage law defined above.

The composite properties, defined by the Serial/parallel mixing theory in the PLCD, are the followed:

Matrix Properties	
Tensile Strength	120.66 MPa
Tensile Modulus	4.67 GPa
Poisson Modulus	0.30
Mode I Fracture Energy	0.68 kJ / m^2
Volume Content	42.6 %

Fibre Properties	
Tensile Strength	4278 MPa
Tensile Modulus	228 GPa
Poisson Modulus	0.0
Volume Content	57.4 %

Table 3 Composite components mechanical properties

The difficulty to define the insert material was solved taking into account its structural performance. The main effect of this material in the beams is allowing the sliding of the section found above the insert along the section found bellow it. To do this, a material with a shear modulus nearly zero has been defined(not strictly zero to avoid numerical instabilities during the simulation). On the other hand, the longitudinal and transversal elastic modulus have been defined with a high value to avoid the penetration of the section above the insert into the section bellow it. The material has also been defined as an elastic material with the followed properties:

Insert Material Properties	
Tensile Modulus	1000 GPa
Shear Modulus	10^{-9} GPa
Poisson Modulus	0.0
Volume Content	100 %

Table 4 Insert material mechanical properties

5.4 MESH DEFINITION

The simulation of the model has been carried out building a mesh with 4400 hexahedral lineal elements. The mesh is showed in figure 20 :

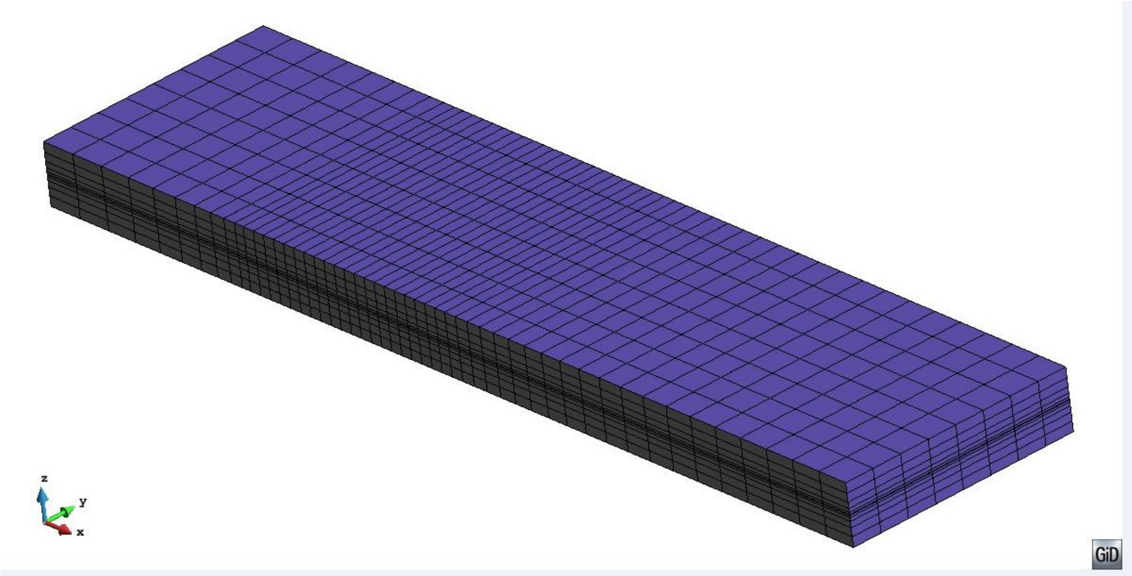

Fig. 20 Mesh description of the model

As we can see in the figure 20, the mesh used is not uniform along both longitudinal and cross directions. This is because, in problems like these, we are interested in variables meshes which allow us to concentrate more elements in zones where we have really high changes in stresses and to use less there where we know that the gradients are barely significant. Quadrilateral elements weren't taken into account just because the computational cost required was much higher.

Here, the damage evolution will be located along the crack, once the propagation starts. So, on the one hand, along the cross direction we are concentrating elements close to the crack, in the inner zone. On the other hand, the damage pattern of such material will be the delamination once the matrix material of the composite is not able to provide any shear strength. At that point, due to the iso-stress condition imposed by the Serial/Parallel mixing theory, the whole composite cannot either provide any shear strength. So, in this beam, the shear distribution is constant along mid span and cero exactly on the middle. That's the reason why, along the longitudinal direction, the distribution of the elements is located through the domain where the propagation of the crack is expected, from the initial point until mid span.

5.5 RESULTS

The results obtained from the numerical model must be analyzed and compared with the experimental ones, showing that our numerical work and program are able to simulate this kind of composite material damage provided by the delamination process.

5.5.1 FORCE-DISPLACEMENT GRAPH

The first comparison will be the global performance of the beam, through the force-displacement graph obtained for both cases. The main problem of running such kind of simulation with composite materials in which damage takes place is that, when we are working in the non-linear tram with hardening plasticity, we usually have problems of convergence. So, this means that we must really take care of the number of steps used to apply the controlled displacement, the amount, as well as the convergence requested to get through the next step. Al those concepts together makes difficult to find the whole performance and, unfortunately, we had problems of convergence on the last part of the graph, beyond the maximum loaded capacity or failure point, so we couldn't find, for instances, the final beam stiffness once the crack has reached its maximum length. However, the agreement with the numerical and experimental results in terms of the initial stiffness and the global process induced by the delamination is well described, as we show in the figure 21:

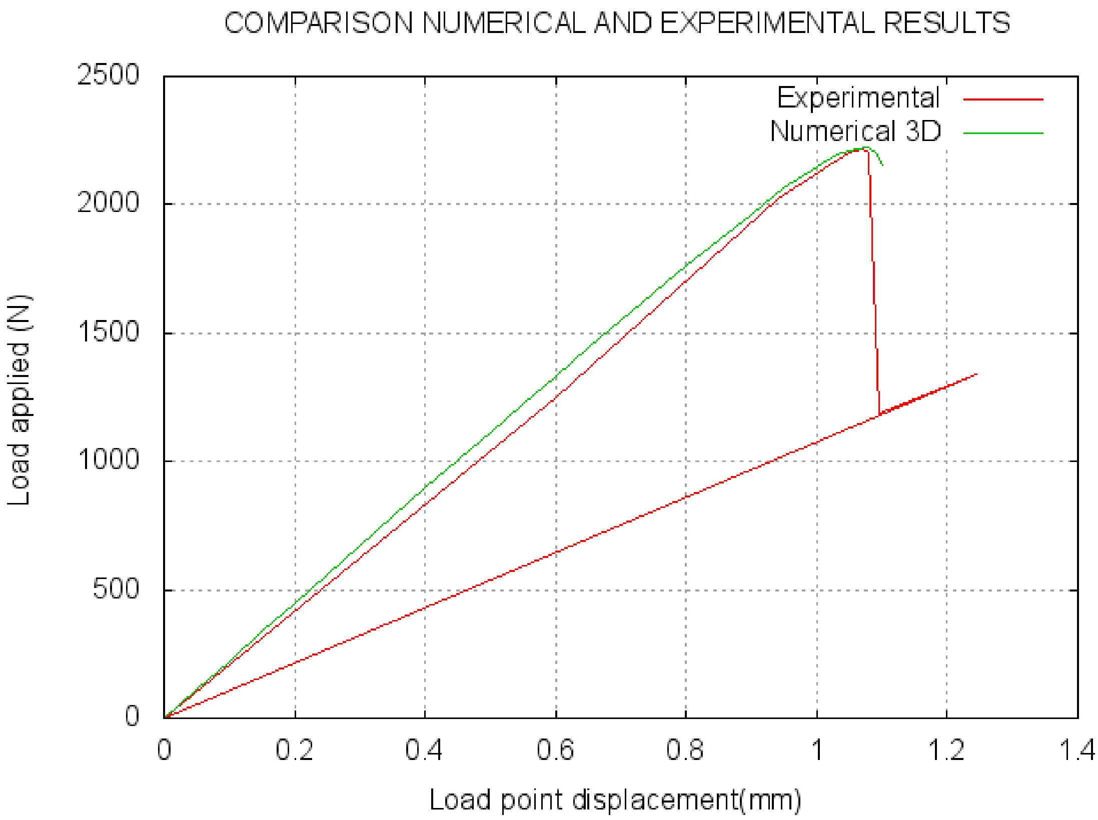

Fig. 21 Force-displacement graph. Comparison numerical and experimental results

Following the figure 21, we can verify that the initial stiffness of both experimental and numerical models is really similar and, if we move on to the beam maximum capacity of the beam or failure point we realize that the agreement is even better. As we explain above, despite of the fact that we tried with several different steps and tolerances, we weren't able to record that sudden change in the constitutive behavior which takes places beyond that point while the crack is getting through the final position. Being aware of that, we researched for the same problem that has been analyzed a couple of years ago for the CIMNE(*Ref .12*), finding that they had already had problems with the convergence of the 3D model and, facing the same problem with a 2D simulation, they were able to get over it. So, the final performances of the beam obtained from them, was that one showed in the figure 22 :

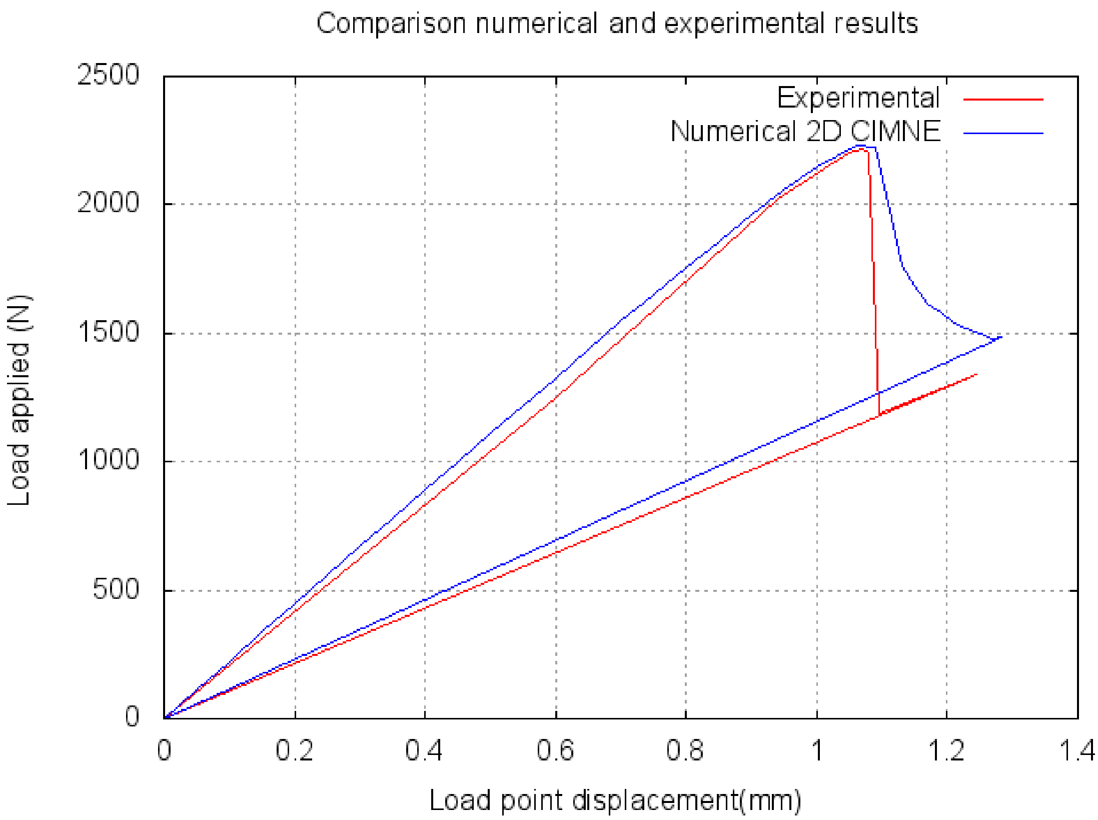

Fig. 22 Force-displacement graph. Comparison numerical from CIMNE and experimental results

So with this information, we can finally simulate the final beam stiffness once the crack propagation has stopped. In this case, the results differ slightly and the numerical simulation(1146MPa) is 6% stiffer than the experimental(1076MPa).

5.5.2 CRACK PROPAGATION

As we explain above, the phenomenon which induces the crack propagation and the damages in the composite is the delamination, which occurs when the shear resistance in the matrix material has been achieved. Due to the Iso-stress condition, the whole material cannot develop any shear strength. In order to reproduce such phenomenon by using numerical models, we previously defined a damage evolution of the material represented in terms of value by the damage parameter, from a value 0 to 1(unable to produce resistance). As we also know, the shear along the beam is uniform until mid span, where is null by symmetry. That's the reason why the crack propagation will stop at mid-span.

So, following the approach, in order to figure out the final crack length we must find the closest point at beam mid-span with a value of the damage parameter equal to one. Due to the problem of convergence running the simulation we couldn't find the final propagation of the crack, but we were able to represent the damage evolution parameter through the beam:

Fig. 23 Plan view of the damage parameter evolution through the beam

As we can see in Fig.23, we haven't already reached a value 1 in any of the sections, so the composite material is still providing shear resistance, but the crack is going forward and if we were able to solve the problem associated with the convergence it would swiftly achieved the value of 1 and would be propagated until mid span. According with the numerical results provided by CIMNE in its study for the 2D simulation, the final evolution of the parameter has the following aspect:

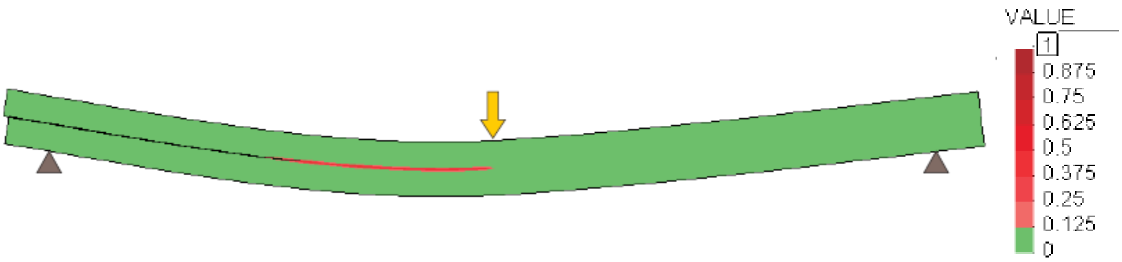

Fig. 24 Damage in matrix material when the maximum deflection has been reached

So with this simulation we can finally ensure that the crack is propagated until mid span due to the shear stresses which induce the failure of the matrix material and, by the iso-stress condition along the serial direction, of the whole composite.

Another way to understand the problem and the loss of stiffness in the beam due to the crack propagation can be studied making reference to the shear connection theory. Before the crack starts its propagation, only the left part of the beam with the crack is behaving like if we had 2 beams, one above the other, without any shear connection. On the other hand, at the end of the propagation, the whole left mid span is handle by this behavior. During the whole process, all the right side is working as though we had only 1 beam with the full interaction. Making reference to the shear connection theory, the distribution of the stresses, both shear and normal, and also the deflection will change depending on the case.

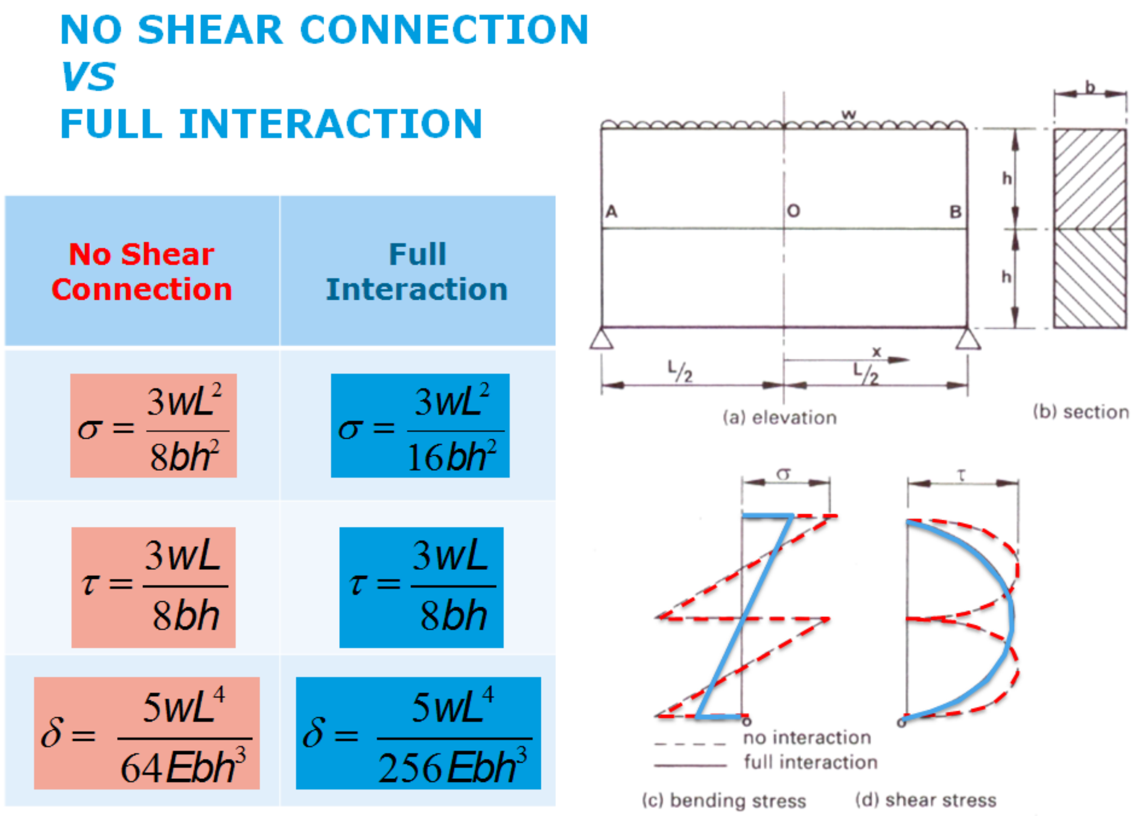

Fig. 25 Different behaviour of two equal beams superimposed with full interaction or no shear connection. The results in terms of Stresses and deflection changes, with a stiffer behaviour using the full interaction.

As we can see in the figures 25 and 26, in terms of normal stresses, when we have 2 beams working together as a single one, the overall stiffness is much better in terms of stresses and, utterly important, the deflection is smaller. This is the behavior we are having in this example. At the end of the propagation, the right hand side will be behaving as a single beam, while the behavior of the whole left hand side will be as a 2 beams superimposed with no shear connection. This will lead to an important asymmetry in the deformation of the beam, which while be rougher in the left side and smoother in the right side, as we can start to visualize in the fig 27:

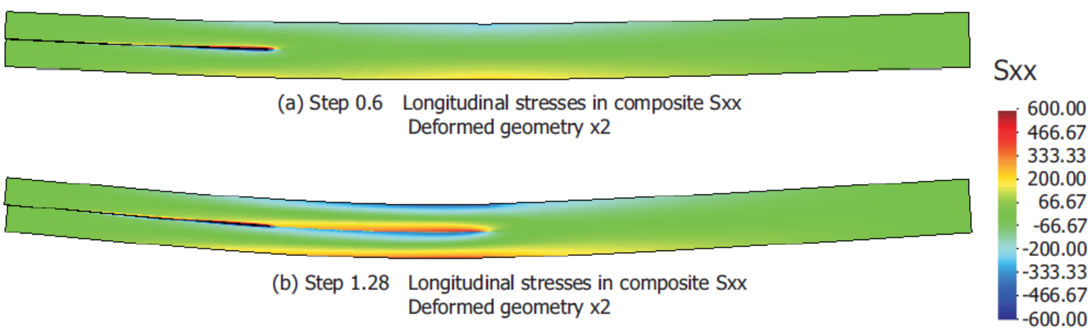

Fig. 26 Different behaviour of the beam in term of stresses showed by the numerical model. From before the propagation until the crack has stopped.

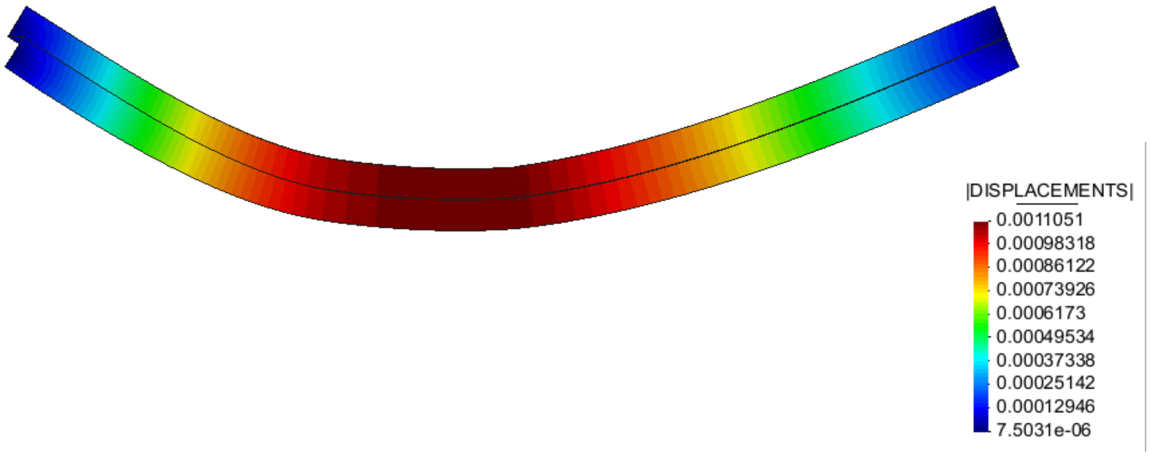

Fig. 27 Asymmetry of the displacement field trough the beam. The displacement peak is moved to the crack side as a result of the 2 beams superimposed behaviour on the left side, caused by the crack. This behaviour is showing the mechanical approach which induces the loss of stiffness in the beam

It is worth noting that, once the crack has been propagated, the phenomenon will be even easier to visualize. In this figure 27 the crack hasn't been full propagated yet, so the asymmetry is caused by the initial crack.

5.6 CONCLUSION

The worked above has definitely showed that the use of the Seriall/Parallel mixing theory is able to simulate the delamination process which usually takes place in damage composite material.

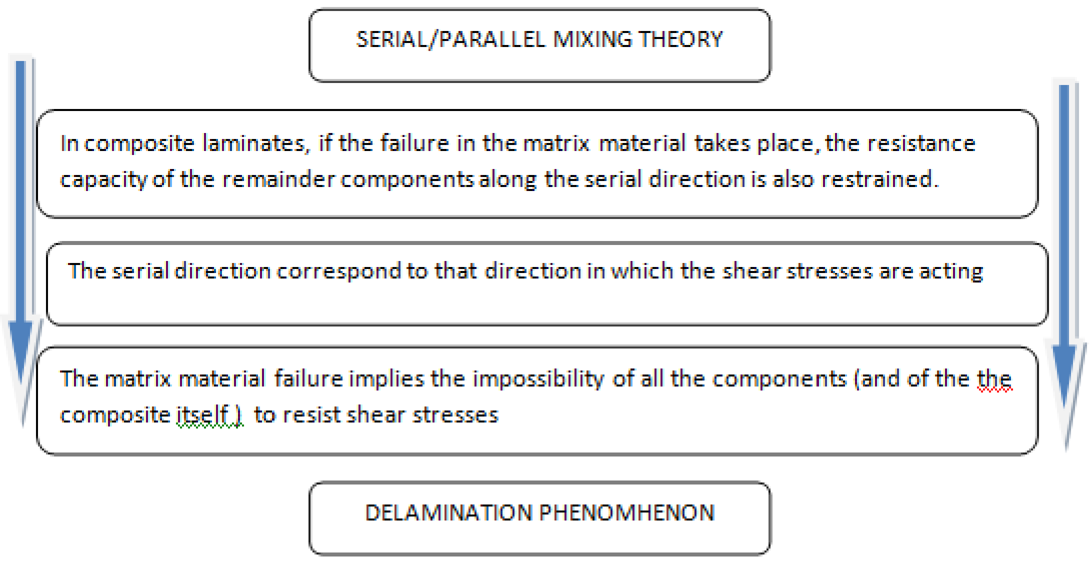

Fig. 28 SP MIXING THEORY applied on the delamination problem described in the section

In this case, the delamination problem is induced by the lost of stiffness in the matrix material as a result of the shear stresses state found in it. The failure of such material implies the failure of the whole composite along the serial direction (all directions not coincident with the fibre direction) and this cause an inability to avoid the shear deformations.

The results obtained from the numerical simulation are nearly the same that the experimental ones. No remeshing formulation where the delamination takes place nor other complicate and expensive tools are needed. On the other hand, we had a problem relating the convergence of the last part of the behavior which didn't allow us to develop the whole simulation until the end, so we had to take advantage of the work provided by CIMNE, whose 2D simulation gave the results associated with the final length of the fracture and the final global stiffness of the beam, significantly reduced. Further studies should be performed on that, because even though the 2D simulation usually reduces the computational cost of the model, a 3D simulation is always better to understand much better the different results and, as we said above, the PLCD program has been made in order to show its fully operability when the 3D simulation is carried out.

CHAPTER 6.COMPOSITE-SANDWICH FOAM-LAMINATED GLASS/EPOXY UNDER SOLICITATION STATIC AND FATIGUE

6.1 DESCRIPTION OF THE PROBLEM

Once we have performed the different examples showed previously, we are already in conditions to simulate a general composite problem trying to fit and agree the experimental results obtained from test in the laboratory and the numerical obtained by the model carried out. The iterative process followed during the work has allowed developing the enough knowledge to build, run and understand the next example obtained from the literature in reference 1

The example obtained is the study of the mechanical behavior in a experimental test (3-point bending) in static and fatigue mode, of a composite sandwich laminated foam with unidirectional reinforcements SMS [0_4]. Thus, we will be aiming to build the proper numerical model of the specimen described in the paper, obtaining numerical results that will be compared to the experimental. The work will be focused on terms of lineal region at the beginning and specially in the non linear once the damage in the composite has started, trying to capture the failure process described in the experimental test.

Even though the paper provides most of the properties of the material that we need to carry out our model, there will be some others that will have to be fitted in order to calibrate both experimental and numerical results. Our main objective was to use the static model aiming to adjust and determinate the mechanical properties required to have an agreement of both models and also understand the mechanical behavior of the material and the damage evolution. Later on, with that information, the intention is to perform the cyclic model simulation trying to obtain the information about the lifetimes of the materials and also the decrease in strength and stiffness.

The description of the essay is very similar to those described previously in the past examples. The most significant changes is that here we are dealing with a Sandwich composite, like this one showed on the figure 29 :

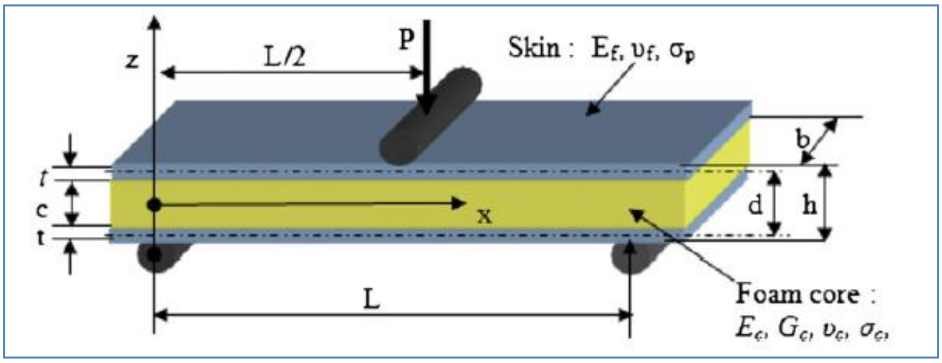

Fig. 29 Sandwich beam loaded in three point bending test

The sandwich is formed by a core which is made of foam, Herex C 70 200 with thickness c and the skin used is a laminate in four-layer [O_4] consisting of a unidirectional glass fibre with surface density (736 g [100 x 30]x 0.85) and epoxy resin 1500/SD SR 2505 (690 g of resin and 230g of hardener). All this composite will be also analyzed with the Serial/Parallel mixing theory explained in chapter 3 of present work. We will be imposing an iso-strain behavior in the parallel direction (fibres direction) and isso-stress in the serial or remained directions. The performance of the specimen will depend on the slenderness L/h. If the ratio L/h is large the ultimate stress may be due to failure in tension of the skins due to the bending process. On the other hand, if the ratio is small, the damage will be the result of the shear loading in the core. The properties of all the materials are showed in the next tables:

Young's modulus (GPa)	Tensile strength (MPa)	Flexural strength (MPa)
2.9 à 3.2	74 à 77	115 à 120

Table 5 Mechanical properties of epoxy resin SR 1500/SD 2505

Weight (g/m^2)	Nominal density (kg/m^3)	Tensile modulus (GPa)	Shear modulus (GPa)	Poisson ratio υ	Tensile stress σ_R (MPa)	Elongation at rupture (%)
300	2540	74	30	0.25	2500	4.8

Table 6 Characteristics of glass fibres.

Type of foam	Nominal density (kg/m^3)	Compression stress (N/mm^2)	Compression modulus (N/mm^2)	Tensile stress (N/mm^2)	Tensile modulus (N/mm^2)
C 70.200	200	4.8	255	6.6	178

Table 7 Mechanical properties under tension and compression of the foam PVC Herex C 70

Type of foam	Shear stress (N/mm^2)	Shear modulus (N/mm^2)	Shear failure (%)
C 70 200	3.5	81	35

Table 8 Mechanical shear characteristics of PVC Herex C 70 200

Type of foam	Thermal conductivity (W/Mk)	Maximum use temperature (°C)	Shock resistance (kJ/m^2)
C 70 200	0.034	80	6.6

Table 9 Thermal characteristics of PVC Herex C 70 200

Unfortunately, once we were already focused on making and building the numerical model of the specimen described, we realize that there were some problems in the description of the example provided by the report. Not only in terms of lack of information, which often happens, but also in terms of twisting some values or results. There were serious doubts about which were the dimensions of the specimen because, in the description, a short and stiff beam was described. On the other hand, in some pictures from the same article, the dimensions were exactly the double. All this was even worse when we started to make models with the properties provided and we observed that the numerical results obtained were quite different even in the first steps where, in theory, using the properties of the materials, we should be achieving without problems.

As a consequence of all this problems described above, we tried different models which could be feasible according with the doubtless information of the report, trying to understand and explain which was the problem of each one and finally trying to lead our final solution to that which might be in agreement with the experimental results.

6.2 FIRST MODEL. DESCRIPTION ACCORDING WITH THE INFORMATION PROVIDED

Thus, our first and logical model was that one with was described in number of value by the report. According with [1] the specimen has a length of l=200 mm , a width of 25 mm and a thickness h of 12mm(8 for the core and 2 for each skin). On the other hand, the distance L between the supports is 142mm, so there will be an outer part of the specimen which won't be working in bending and its movement will only be due to the rigid solid displacement.

The mesh built is again a variable mesh, basically because of this fact of the outer part without working. There, larger elements are selected and, on the inner part of the supports, the elements are smaller. The elements selected were quadratic hexahedral. After making some studies of the results, in order to describe the stresses in a more suitable way, we decided to use the quadratic ones. Hence, the geometrical features and the mesh selected are shown in the figure 30 :

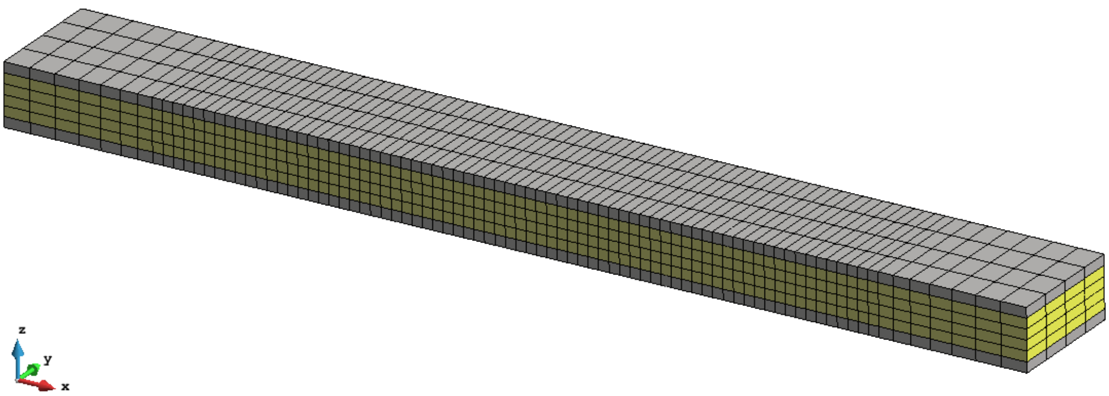

Fig. 30 Mesh of the specimen composed of 2016 quadratic hexahedral elements

With the characteristics provided in the tables, we lack the information regarding the Volume content, of fibre and matrix, in the composite material located on the skins. So what we did was, first of all, to select all the other properties to carry out the simulation and try to change that percentage value in order to fit both numerical and experimental results. We started with a percentage of fibre of 60%, widely used in composite material. The main problem of this model was that, with this value of fibre, the overall performance of the specimen was too stiff, really complicate to fit with the numerical results even in the linear behavior. Only for a value arround 10% of fibres, the stiffness of the model starts to achieve agreement with the experimental one. The results in terms of Load applied and displacement in the linear region are showed in the next picture:

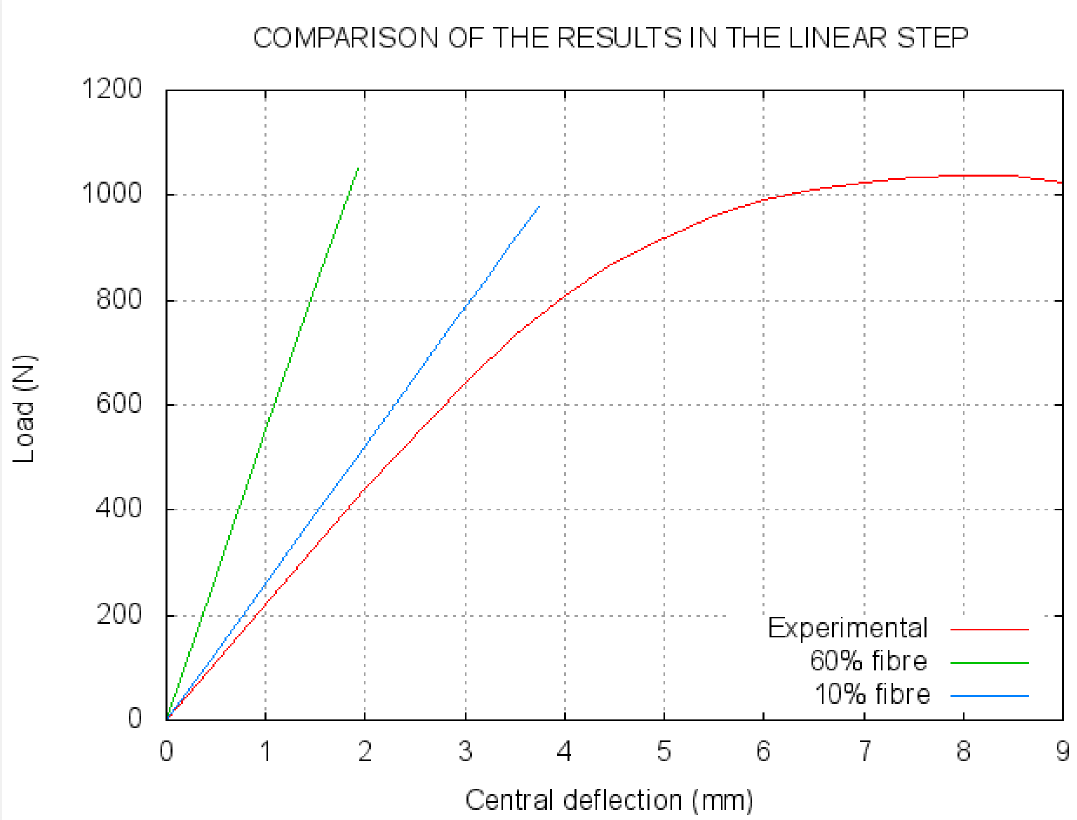

Fig. 31 Lineal behavior of the specimen depending on the % of fibres selected.

Thus, basically, the problem relies on the fact that with a value of 10% of fibres and 90% of epoxy as a composite material we are barely increasing its longitudinal strength so, that tiny percentage doesn't make sense in the real composite. In this point we started to think that maybe there were some information in the article that wasn't reliable enough, just because with the modulus defined by the article, at least the linear behavior should be in agreement with the experimental.

Moreover, when we tried to move on and get some results of the performance in the non linear step once the damage starts to propagate, we found out some problems with the numerical model. A specimen like that which has been strengthened with fibres on the skin will bear the burden in terms of bending, so mainly the mechanism of damage will be induced by the shear stresses on the foam core, which will cause the delamination of the composite. The problem that we have, showed in the figures 32 and 33 , is induced by the local effects on the points where the load is applied. Some stresses located there are not allowing the damage of the specimen due to the shear stresses on the core.

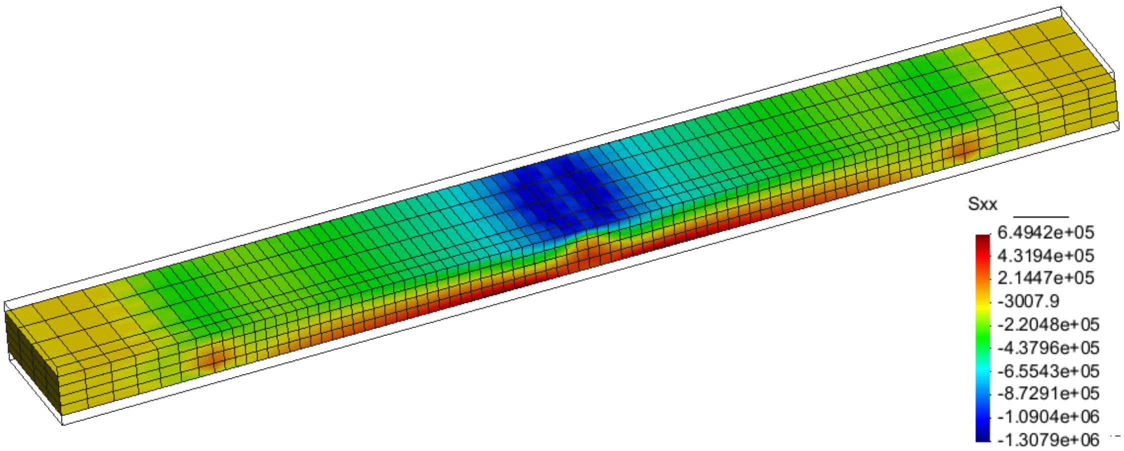

Fig. 32 Normal stresses distribution in the Core of the specimen. The local effects of the load applied are inducing an increase in the Stresses which leads to the local failure of the material.

As we can see in the figure 32, the distribution of the normal stresses is switched in that zone where the load is applied. Thus, the damage will be also local, because the specimen won't have enough time to allow the damage in the core as a result of the shear stresses.

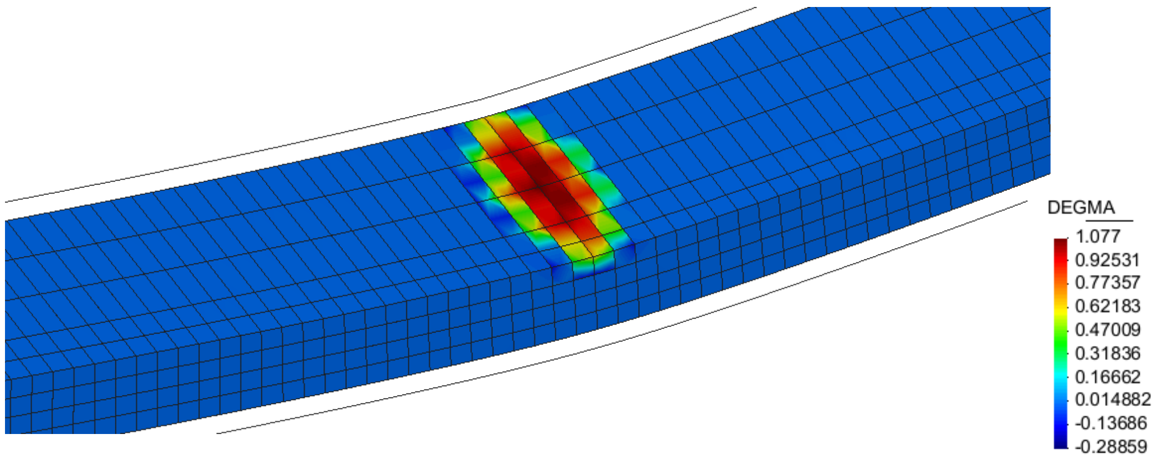

Fig. 33 Local damage of the specimen as a result of the gradient stresses caused by the local load applied.

So with this problem, we realize that this first simulation wasn't going to lead to a reliable and consistent solution, not only in terms of the 10% of fibres used, but also in terms of the local effects. So we had to solve the problems associated with the local stresses, and to do it, we moved on the second model where the damage under the load applied was restrained by using an elastic material in the zone affected.

6.3 SECOND MODEL. ELASTIC MATERIAL TO AVOID THE LOCAL EFFECTS OF THE LOAD APPLIED

The problem associated with the local effect in those points where a punctual load is applied has been showed in the last example. We must prevent that, and, in order to do it, what we did was to perform a model with the same characteristics but, just on a small region in the middle, using a elastic material as a Core, with the same properties than the other one but with the failure restrained.

With this small detail, we are preventing the formation of damage under that zone, allowing the failure of the specimen by its natural performance related with the shear stresses found in the Core. On the other hand, it's important to rely on the fact that the local stresses arisen from that load will still be there, but using an elastic material we'll never achieve the damage there and these stresses will be neglected. This is a fair approximation because in the experimental test the load is not applied in a single line of the specimen but rather in a wider region, so this stress concentration is largely mitigated.

The mesh used to perform the model is described and showed in the figure 34 :

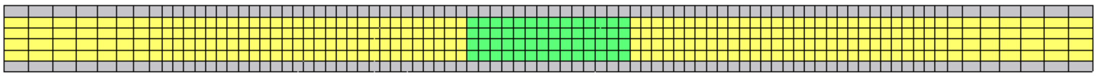

Fig. 34 Mesh used to perform the simulation. An elastic core material has been used around the mid span of the beam in order to avoid the local effects of the punctual load.

As we see, the mesh has been performed following the same criterion that was followed on the last example, allowing larger element on the outer part of the supports where the bending doesn't take place. The only difference which we can appreciate is the elastic core used to describe the sections around mid span of the specimen.

Going further and performing the simulation we were able to represent the non lineal step of the plot Load-Central deflection. Now, the initiation of the damage due to the cracking of the core was represent from the model. The comparison of the results obtained and the experimental is shown on the figure 35.

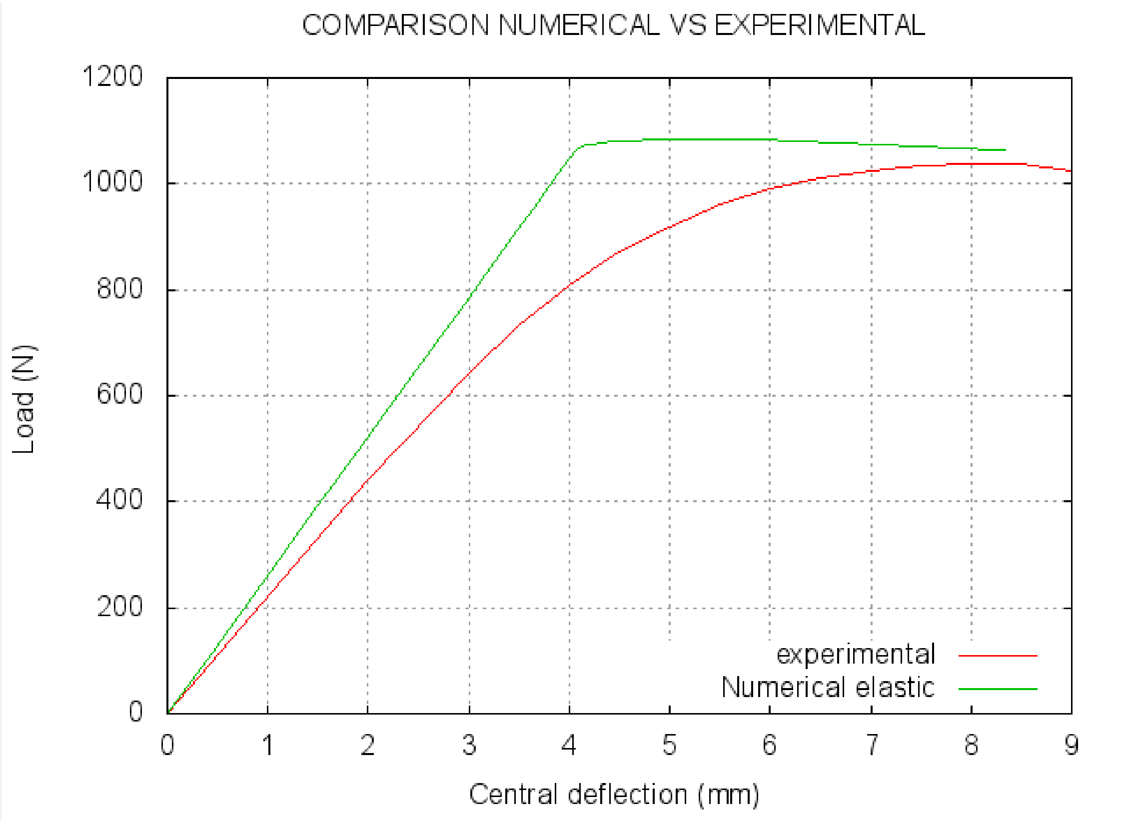

Fig. 35 Comparison of the performance obtained by the numerical model with the experimental one

So, like in the last model, the initial stiffness and the overall behavior during the linear region remain equal, but now, we are able to move on through the nonlinear region. Analyzing the results in terms of this plot, we observe that the numerical model differs from the experimental testin different aspects. The first one is the initial stiffness, our model is stiffer than the experimental. We may reach the same stiffness by reducing the percentage of fibres but as we said above, a value of 10% of fibres is already ready low, doesn't make sense to drop it even more. The second one is related with the second phase, the non-linear step. Our model is not able to predict the ductile behavior achieved for the experimental specimen, with very much changing in stiffness. Rather than that, we are getting a longer linear step and a sudden change in the stiffness. So we can say that the damage in our model starts too late.

On the other hand, we are obtaining and representing this damage as a result of the shear stresses found it in the core, so the problem associated with the local damage has been solved.

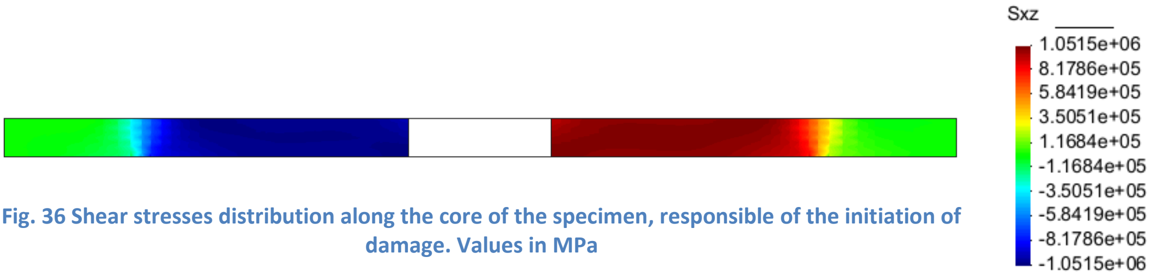

Fig. 36 Shear stresses distribution along the core of the specimen, responsible of the initiation of damage. Values in MPa

The shear stresses distribution is not altered, but in our model, we need a larger load in order to starts the damage. This may be explained in terms of the value selected as a Shear stress on the core (Defined by the article as 3.5 MPa).

The second phase would be trying to analyze the results in terms of the mechanism of damage generated during the failure of the composite. Regarding the information of the experimental essay, the fracture surface observations in static loading is produced mainly by a delamination between compressed skin and core, then, shearing of the foam and finally loss of cohesion of the lower skin foam (figure 37)

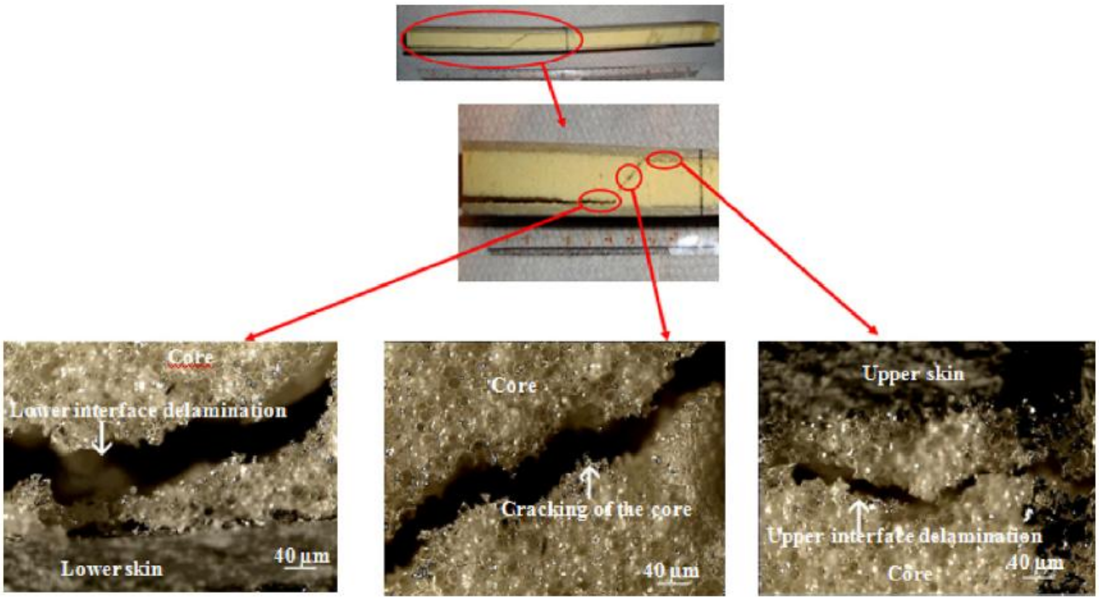

Fig. 37 Macroscopic and microscopic views of fracture surface of the SMS [04] sandwich. Ref 1

The results of the mechanical failure predicted by the numerical model shows some aspects which are in agreement with the experimental solution, specially in qualitative terms. First of all we must know the complexity of the problem we are working with. The numerical simulation of a non linear section in such kind of materials is performed with a lot of uncertainties and ,those uncertainties, must be solved in terms of conditions imposed by the model. These conditions are not only satisfied in one way. So sometimes, the iterative process set up for the system equations solver leads to some solutions with cannot be understood as though they were fully reliable in terms of exact results. The engineer, which his criterion and his knowledge, must be able to scan the result and obtain a global conclusion.

For instances, analyzing the evolution of damage showed by the model in figure 38 , our failure is produced by a delamination between core and the lower skin. This could perfectly be in agreement with the experimental results, because building our inner part with an elastic material we are avoiding the delamination with the upper skin.

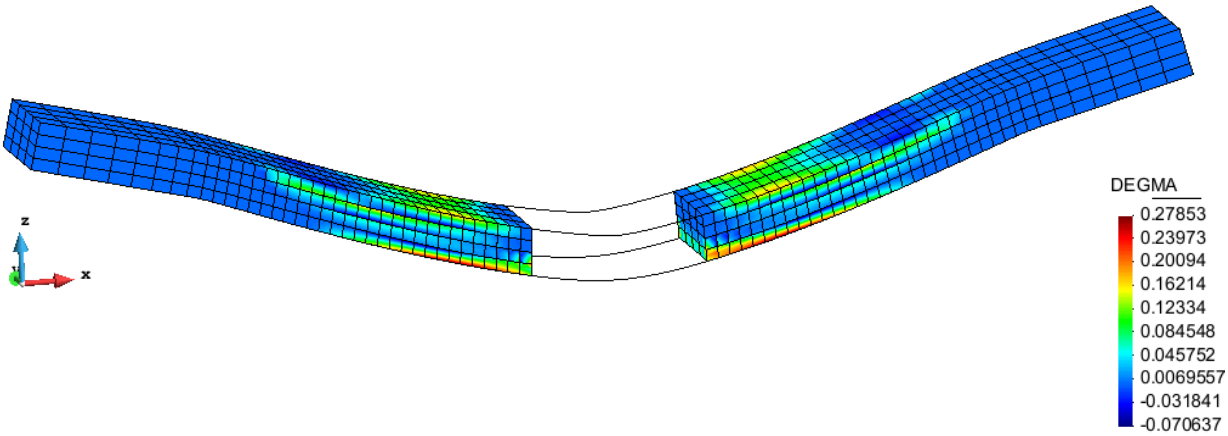

Fig. 38 Distribution of the damage once it starts to take place. The damage is started in the contact between the core and the lower skin

On the other hand, according with the experimental test, this crack should be propagated along the contact core-skin until the support but, the continuation of damage provided by our model and showed in the figure 39, is affected by a jump in the propagation and the crack propagates along the middle of the cross section.

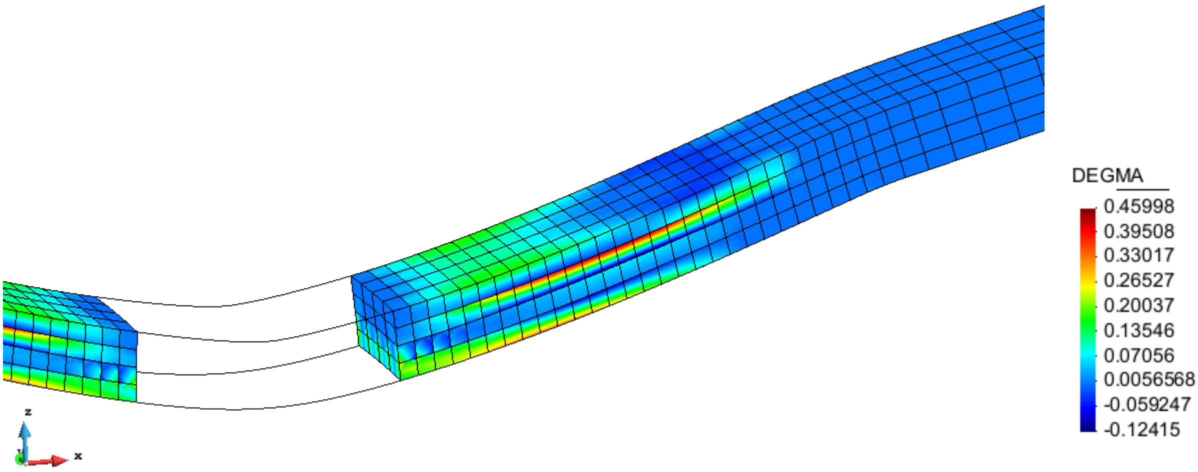

Fig. 39 Distribution of the damage at the next steps. The damage is affected by a jump and the propagation takes place along the middle of the cross section

From this step until the end, the result is perfectly reliable in terms of qualitative behavior, because the mechanism of failure is the propagation of that crack until the support, the same behavior showed by the experimental failure.

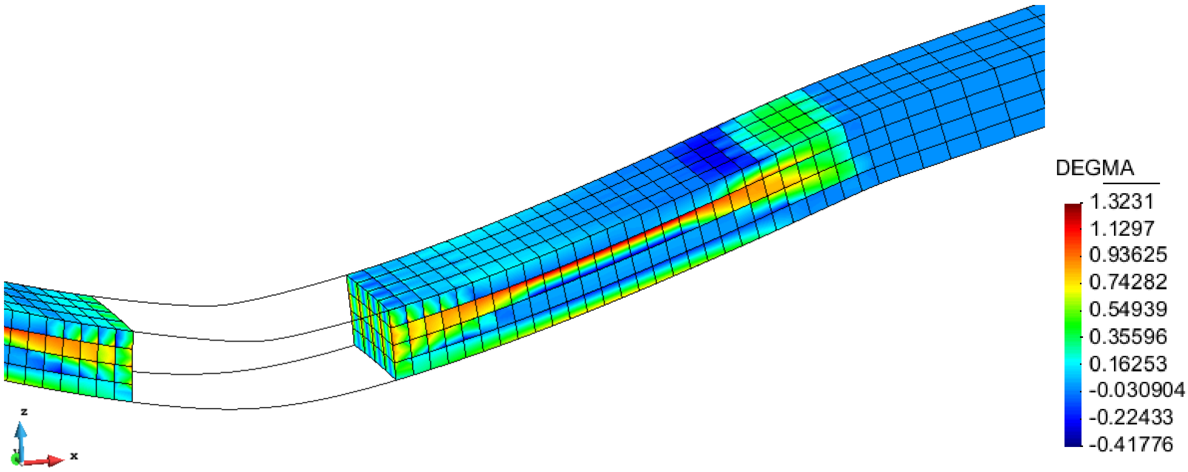

Fig. 40 Distribution of the damage at the final steps. The crack has been fully propagated until the supports.

So why in our model the propagation is not happening along the core-skin contact? So as we said, the study of the non linear section by using numerical models is complex and in this case, the solution provided by the iterative process is meeting the requirements in terms of equality between internal and external forces, in the same manner that the actual solution met. So as we said above, these results can be adopted and validated as a reliable qualitative solution. The formation of a crack and the propagation of it until the support. On the other hand, we cannot say that there is an agreement between the quantitative results of both models, because some several random process performing the numerical model has led our solution to a different final behavior. Further studies should be carried out trying to represent the actual performance and finding why we are not getting through it. One possible reason may be very homogeneous distribution of the shear stresses along the core. If we have a look at the figure 36, we can see that along the whole cross section of the core, the value of the shear stresses is pretty much the same. The crack will be propagated through the where the resistance is lower so, if we have just a little variation in numerical terms, the crack will appear at that point. In our model the weak point is on the middle, but the values are so close, that could be happening at whatever height.

Another important comment which it’s worth explaining is the values showed on the DEGMA parameter legend. As we know, such parameter is the responsible of showing the distribution of the damage along the material in terms of the effective stress and corresponds to the "d" value of the damage model described in equation 10. When the material is not damaged the value of the variable is 0 while when it’s completely damage, the value adopted is 1. So in the legend provided by GID, this variable is reaching damages higher than 1 and even smaller than 0. This doesn’t make sense, but it’s a visualization GID problem which cannot be prevented. It is arisen from the fact that the value of the parameter is calculated in the Gauss points of each element, and sprawled along the whole element by interpolation. So if in one single element we have values close to 0 and 1, once GID performs the interpolation, the values go far from their limits. In the figure 41 one element is isolated and we can observe the value of the variable DEGMA within some Gauss points.

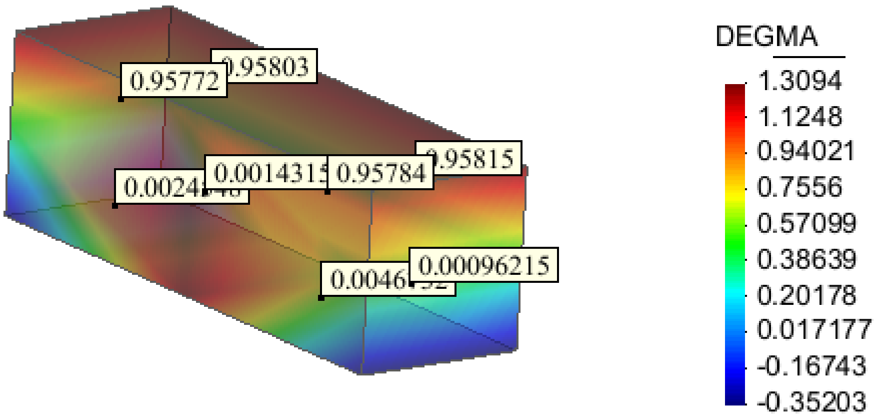

Fig. 41 Value of the DEGMA parameter within some Gauss points of an isolated element. In those points, the damage is limited between 0 and 1. The interpolation performed by GID induces values which doesn't mechanically make sense

So we have seen that this model is able to predict different aspects of the delamination fracture which takes places in the experimental specimen, specially in terms of qualitative results, not so good in overall behavior. The main problem and the reason why we decided to taste another model was the fact that the stiff behavior of the model was forcing us to select a 10% of fibres in order to fit the plot Load-Displacement in the linear step. As we already explained, this is not the actual and more suitable content of fibres for a composite material. So this reason, in addition to the existence of some controversies with the geometric characteristics of the test piece, lead us to a final model in which the content of fibres was more reliable.

6.4 THIRD MODEL. LONGER AND MORE FLEXIBLE SPECIMEN.

Even though the geometrical properties of the specimen were fully described in the article, there were also some controversies. The main one appear in a .picture in which the static test in three-point bending was performed, showed in the figure 42.

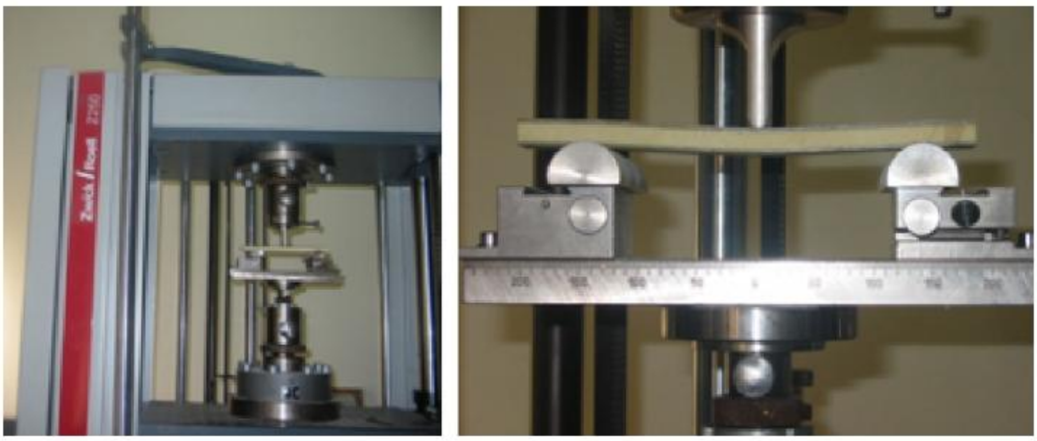

Fig. 42 Experimental set up. The measures of the specimen are the double of those described by the article. Ref 1

In this picture we saw that the length of the specimen was actually the double. So this, in addition to the fact that the global behavior obtained from the last model was too stiff, lead us to think that maybe the dimensions of the article were referred only as a mid span of the beam. With this new configuration, we would be getting a more flexible behavior due to the increase in the length and we may describe the experimental results by using a percentage of fibres more feasible.

Hence, we performed again the same model but this time changing the dimensions of the piece. The elastic core material was also used in the mid-span zone in order to avoid any problem associated with the punctual load applied. In the figure 43 we can see the building of the mesh, following the same criterion than in the past models.

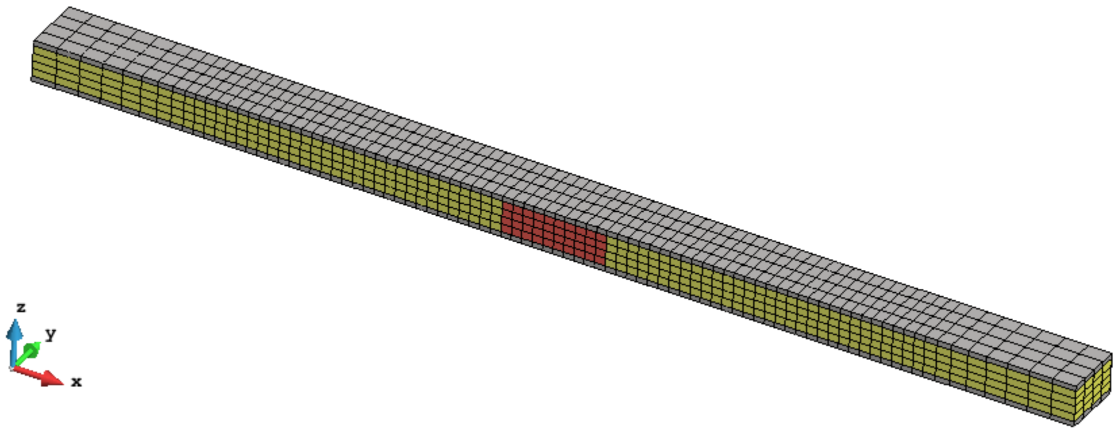

Fig. 43 Mesh used to carry out the simulation. The elastic core material at mid span is used to avoid problems associated with the punctual load applied

The results obtained from the model are really similar to those from the last one, but in this case, we are fitting perfectly the elastic-tram just selecting a 60% of fibres, a value much more feasible and reliable for a composite material. The results in terms of the plot Load-Central deflection are showed in the figure 44.

As we can see, the initial stiffness predicted by the model is nearly the same that the obtained by test. The same happen with the overall performance and in the non lineal region. We are still obtaining the damage later than the experimental model and the sudden change in terms of stiffness is also taking place.

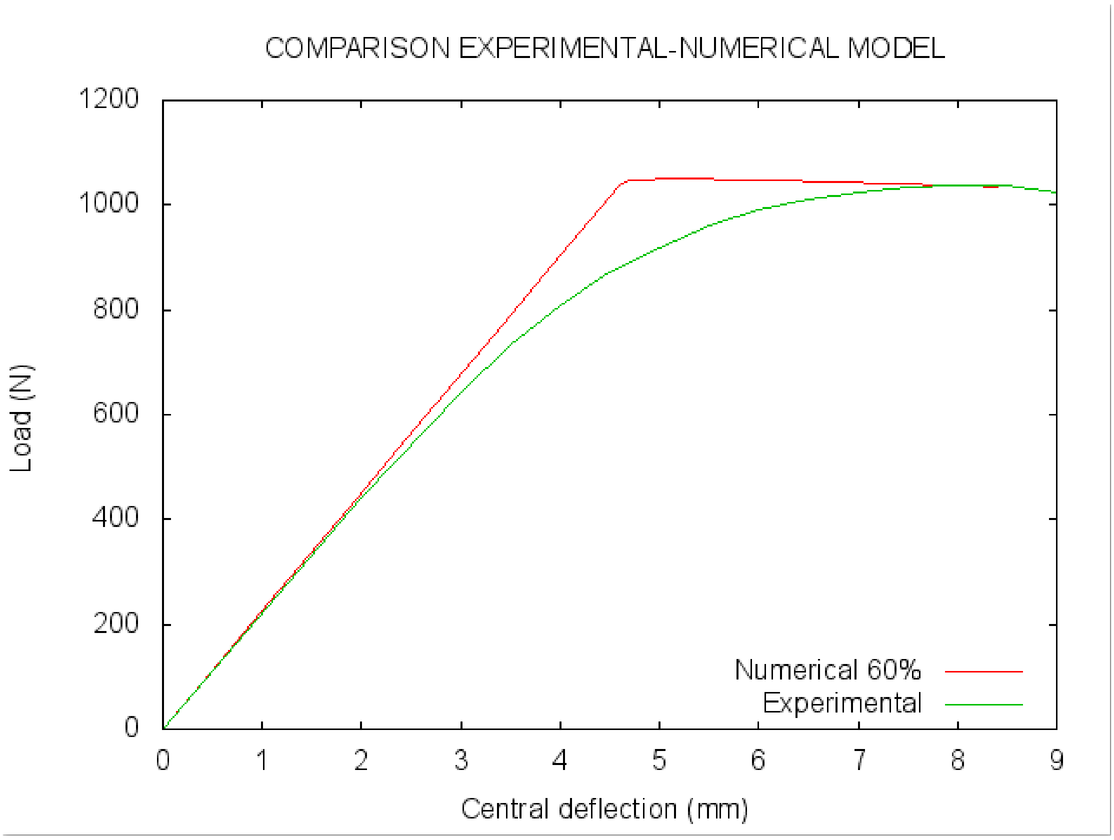

Fig. 44 Comparison of the performance obtained by the numerical model with the experimental one

The behavior of the mechanism of failure is really similar to the last showed on the past example with the 10% of fibres. The damage is started on the contact core-low skin and suddenly, there is an instability on the solution which leads to the formation of a crack through the medium point of the cross section, but in this case the damage is even more concentrated only on those points, as the figures 45 and 46 show.

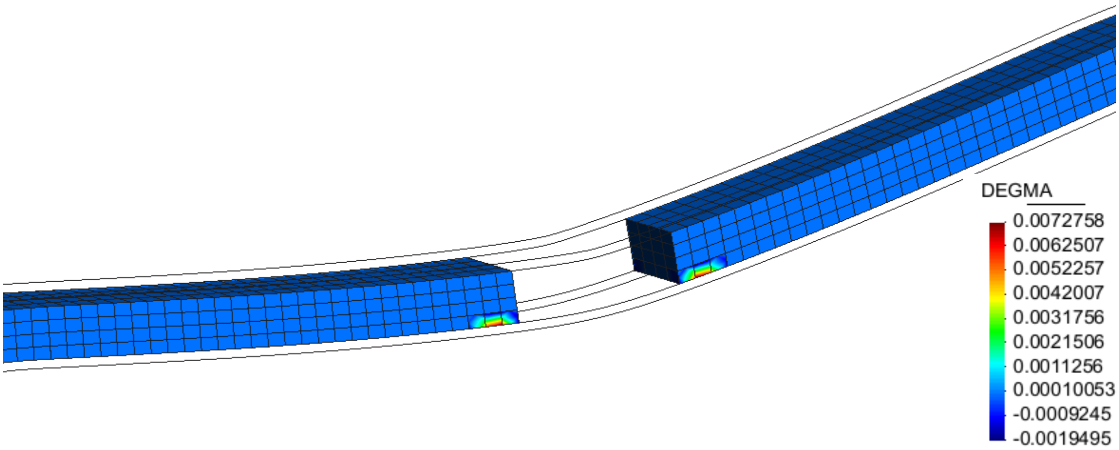

Fig. 45 Distribution of the damage once it starts to take place. The initial damage is located in the contact core-low skin.

The propagation of the damage is developed until the supports. Beyond this point, the beam is not under any solicitation, neither bending nor shear, so its movement it is not producing stresses, behaving like solid rigid.

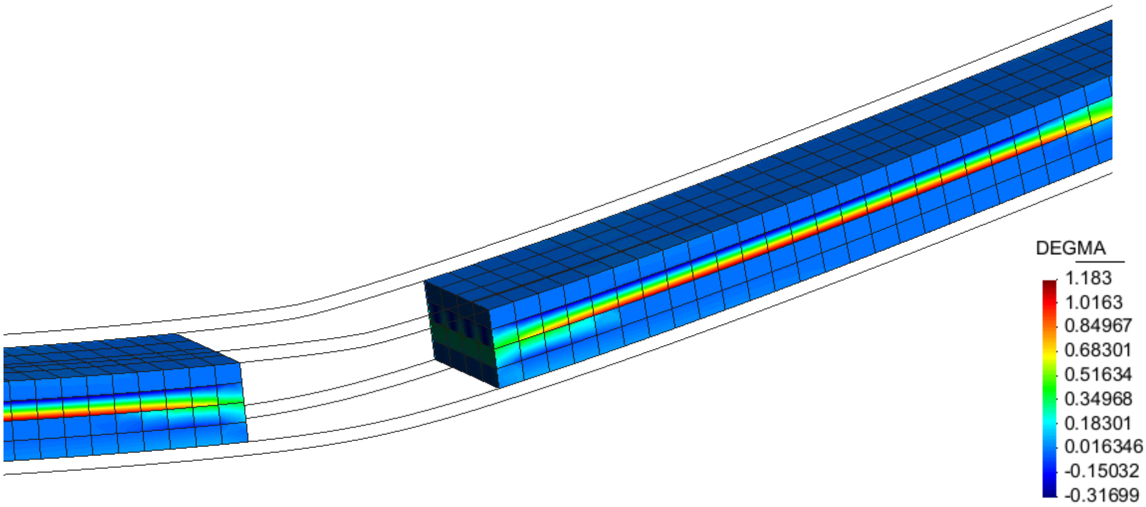

Fig. 46 Distribution of the damage at the next steps. The damage is also affected by a jump and the propagation takes place along the middle of the cross section.

Thus, with this new and last example we are getting pretty much the same results in both terms, constitutive plot and evolution of damage. The necessity to carry out this new model was due to the fact that to obtain consistent results in the last, we had to decrease the percentage of fibres up to 10%. With the new solution we are reaching the same performance with 60% , much more reliable. So, from than point of view the purpose of the new model has been successful.

On the other hand, this specimen is longer than the last one so the bending moment and the normal stresses take higher values, but rather we are reaching pretty much the same performance. This could be because of the fact that the failure in this beam is associated with the shear in the core, the core is barely affected by the bending, which is focused and resisted by the skins. Those skins, reinforced with the glass fibres, are still able to reach the loads.

Like we said above, the results must be understood from the qualitative point of view. The formation of a crack and the propagation of it are the responsible of the specimen's failure.

The analysis could perfectly show that the real geometrical measures of the specimen are these showed on this last model, basically because the content of fibres is more likely. Further studies should be carried out to ensure it though.

6.5 CONCLUSIONS

During this part of the work we were aiming to develop a numerical model which was able to simulate and predict the results of the experimental static and fatigue tests described by the article in reference 1. The simulation was started trying to fit some properties which allowed us to adjust the results during the linear step. Some problems with the predicted performance of the specimen combined with the fact that there were some controversies with the geometrical description of the piece force us to test some different model in order to achieve the agreement in the more efficient, feasible and reliable way.

Several problems associated with the numerical simulation in non-linear steps were arisen from the different models. As we explained in the current section, during the non-linear phase a lot of uncertainties take place and the conditions imposed to reach the convergence may be several, not only one. In order to move on at each step of the simulation we are imposing that the external forces must be equal to the internal found in the material, but we don't know anything else about the stress distribution sought. This leads our solution to a quantitative value which is not the experimental that we were expecting, but the solution reached is showing the same mode of failure, so it can be understood like reliable from the qualitative point of view.

Unfortunately, due to the problems found to develop the models in the static analysis and the controversy of the data provided by the paper, we didn't perform the fatigue numerical simulation, as it was meant as one of the main objectives. On the other hand, coming up with several models trying to solve the problems associated with each one has definitely helped to understand the numerical simulation background of the composite materials.

CHAPTER 7. CONCLUSIONS AND FURTHER STUDIES

The present work has been focused on the numerical simulation of the composite materials in both linear and non linear regions. Several experimental tests provided by the literature have been simulated using the PLCD(CIMNE,2008) finite element code, a code with work with three-dimensional solid geometries and allows the representation of both material and kinematic nonlinearities. The results obtained from the model has been tested in terms of agreement with the experimental tests, showing that the SP mixing theory implemented in the code in addition to the appropriate constitutive equation is perfectly capable to simulate the delamination process and predict the material behavior of the composite components. However, the most important part of this work specially in terms of learning and knowledge has been the familiarization with the simulation process and all its concepts itself, such as the iterative process of the Finite Element Method, the steps followed to carry out a simulation and, even more important, the different problems arisen, how to understand them and deal with them, being aware of the high complexity of a non-linear composite simulation in which the damage is propagated.

The content of the work has been splitted in different parts which are worth a further explanation:

1. Researching of the mechanical failure in composite materials, specially the effects of the fatigue loads and the delamination model of failure.
2. Description of the numerical tools and consequence implementations that have been used to carry out the simulations. The scope of the SP MIXING THEORY and its capability to provide consistent and reliable results.
3. The iterative process followed in order to become familiar with all the numerical simulation concepts, starting from simple examples where the linear and quadratic tridimensional meshes were tested, and finishing with the performance of different non-linear simulations were the delamination failure was predicted by using the SP mixing theory.
4. The awareness of the several problems arisen from a non-linear simulation and their complexity in terms of convergence and iteration. Not only is important the formulation of the model, but also the knowledge necessary in order to solve feasible problems and reach final conclusions about the overall capability of a model.

From the first point, a first introduction in the composite materials was utterly necessary, just because of the fact that we barely studied them before and, in order to perform the simulations, a basic knowledge in the composite behavior, its heterogeneity and how the different components of the composite, in terms of matrix and fiber, are able to interact with each other. As important as that were the mechanical failures which usually take place in a composite. Our research was focused specially on the cyclic loads related with the fatigue, how we can characterize a cyclic load and which are the effects of them in the composites. Concepts as the Reversal Index and the S-N curves in order to characterize the life expected of a composite under a specific stress state were widely analyzed. On the other hand, the mechanical failures were mainly focused on the delamination and its consequences, the formation of a crack due to the different strength of the composite along different directions and the impediment of the whole composite to resist any shear load once the failure of the matrix has been reached.

From the numerical point of view, a procedure described in terms of the PLCD finite elements code has been widely explained. Its interface with several commercial software like GID makes us easier the pre process in terms of the mesh building, and the post process in terms of results reading. Special attention has been placed on describing the SP mixing theory and its scope to deal with the composite delamination numerical simulation. Equilibrium and conditions are imposed by considering an iso-strain behavior through the parallel fiber direction and an iso-stress behavior along the remainder directions. This condition, in addition to the fact that the effects of each component in the overall composite is taken into account by its volume content allow us to describe numerically the natural behavior found in a composite delaminated material.

The process in order to get the enough knowledge to deal with the simulation of the experimental tests provided by the different literature has been got through it by performing an iterative process in which several concepts related with the FEM and composite simulations were solved. First of all an analysis to figure out which is the more proper mesh to use in a specific problem and the difference between them in terms of numerical results and computational cost have been analyzed. The main conclusion was that with both we are able to reach the pretended results, it's just a question of efficiency and computational cost which will decide our linear or quadratic mesh. Nevertheless, it's worth to say that when the values showed by the Gauss points are going to be important, for instances, when the values of the stresses or different internal variables must be fully reliable, it is better to use a quadratic mesh just because of the fact that it has more Gauss integration points so the volume represented by one element is better distributed.

Finally, and the most important conclusion it's the awareness with the really high complexity to deal with non linear numerical simulations, specially in composite materials. These complexities are showed in terms of instabilities which doesn't allow us to reach the final steps of a simulation or in terms of doubtful results from the quantitative point of view. As we explain in the final simulation of the present work, a numerical simulation tends to be really unstable. The approach followed to control the results is handled in terms of conditions in equilibrium, for instances, the equality between the external and internal forces. This equilibrium can be reached with more than one solution if we don't restrain further conditions. Thus, the iterative process set to predict the next solution and the final results provided by the simulation are very difficult to reach. Also important is the fact to dispose of experimental reliable and fully defined data in order to carry out a numerical simulation, because otherwise we couldn't reach if the results are in agreement by random process or by the real and proper simulation. That's the reason why, in some occasions, the results of a simulation must be understood only in terms of qualitative results. The engineer must be able to analyze the procedure and understand until which point the simulation can be reliable.

Regarding the further studies, it's worth noting that we have also had different problems which didn't allow us to carry out and obtain some final solutions. The problem of convergence is always present in a numerical simulation. In the example in chapter 5, further studies should be performed in order to know why we couldn't go further in the 3D simulation of the pre-cracked specimen. We mustn't forget that one of the important aspects of the work was the fatigue simulation of a composite material searched on the literature. Unfortunately we wasn't able to carry it out, just because of the problems that we had during the other simulations and, specially, the controversies found on the literature paper where the fatigue test was performed. It was a very ambitious objective, specially if we take into account the complexity of simulating the non-linear performance of composites and if we consider that this is a subject that is currently under investigation.

Nevertheless, all these problems have also allowed to understand the difficulties related with the non-linear simulation, and also the necessity to provide solutions in order to come up with models which can be in agreement with the experimental results. I can only say that the level of knowledge, experience and motivation achieved at the end of this work in the general numerical simulation are fully satisfactory.

REFERENCES

(1)Chemami, A., Bey, K., Gilgert, J., & Azari, Z. (n.d.). Behaviour of composite sandwich foam-laminated glass/epoxy.

(2)CIMNE. (n.d.). COMPACK PLCD TUTORIAL.

(3)Costa Balanzat, J. (n.d.). *Estudio constitutivo de materiales compuestos laminados sometidos a cargas cíclicas.*

(4)DELAMINACIÓN EN COMPUESTOS MEDIANTE LA TEORÍA DE MEZCLAS SERIE/PARALELO. (2007, Junio).

(5)GID USER MANUAL. (n.d.).

(6)http://www.colorado.edu/MCEN/MCEN4173/chap_01.pdf. (n.d.).

(7)Interlaminar Fracture Testing of Composites. (n.d.). Retrieved from http://www.merl-ltd.co.uk/2003_materials/composites04.shtml

(8)J. Aghazadeh Mohandesi, & Majidi, B. (n.d.). Fatigue damage accumulation in carbon/epoxy laminated composites.

(9)Ling, D. S., Fang, X. J., B.N.Cox, & Yang, Q. D. (n.d.). Nonlinear Fracture Analysis of Delamination Crack Jumps in Laminated Composites.

(10)Martinez, X. (n.d.). *Micro-Mechanical Simulation of Composite Materials using the Serial/Paralel Mixing Theory.*

(11)Martinez, X. (n.d.). Teoría de Mezclas Serie/Paralelo.

(12)Martínez, X., Oller, S., & Barbero, E. (n.d.). STUDY OF DELAMINATION IN COMPOSITES BY USING THE SERIAL/PARALLEL MIXING THEORY AND A DAMAG.

(13)Mayugo Majó, J. A. (n.d.). *http://www.tdx.cat/handle/10803/6860.*

(14)R, G. A., I, A. B., J, G. M., F, G. P., & J, S. M. (n.d.). SIMULACÍON CON ELEMENTOS FINITOS DE PROPAGACIÓN DE GRIETAS EN MATERIALES COMPUESTOS.

(15)S. Oller, B. L. (n.d.). *ESTRUCTURA DE ENTRADA DE DATOS DEL PROGRAMA PLCd4.*

(16)Wang, P., Her, Y., & Yang, J.-M. (n.d.). Fatigue behavior and damage modeling of SCS-6:titanium:titanium aluminide hybrid laminated composite.