COBEM-2017-1238
INFLUENCE OF LAMINATION TECHNIQUES IN COMPUTATIONAL SIMULATION: COMPARISON WITH EXPERIMENTAL RESULTS OF TENSILE TESTS

Aline Peres Leal
Thiago Pontin Tancredi
Gabriel Benedet Dutra
Universidade Federal de Santa Catarina, Centro Técnológico de Joinville
aline_peres_leal@hotmail.com, thiago.tancredi@ufsc.br, gabriel.d@ufsc.br

Abstract. In this work, we present a comparison between the experimental results of tensile tests and computational simulations performed with Finite Element Method. The use of Finite Element Method to simulate composite materials is a recent development that has low use when compared to simulations of metal structures. There are several approaches to using the Finite Element Method in the simulation of structures of composite materials, but the influence of the laminating technique is still little known. This work reduces this gap with a discussion about the influence of the consideration of lamination techniques in the computational simulation. Experimental results include tests with specimens fabricated using manual lamination, vacuum lamination and infusion. It was observed that the FEM simulation using shell multi-layer elements showed an average of 3,6% of variation and using solid multi-layer elements showed an average of 2,3% of variation in the value of the tensile strength in relation to the tensile tests, regardless of the manufacturing process used.

Keywords: Lamination Techniques, Finite Element Method, Computational Simulation.

1. INTRODUCTION

Currently, the great competition in the industry demands more and more innovation in the projects, searching for high technology products at the lowest possible cost (Nicolais; Milella, 2011). In this context, the selection of the material must be made with care, so that the necessary physical and mechanical properties are obtained economically. As a result of this search, the use of composite materials has been amplified in several industries, such as naval, aerospace and automotive.

The composites belong to a class of engineering materials obtained by the union of two or more components that together confer unique properties, higher than that of each material alone. When the composite is formed of a matrix and a reinforcement, it is referred to as fiber reinforced composite material. In this case, the fiber is the element that confers resistance, rigidity and hardness while the matrix is responsible for the distribution of the loads, the resistance of the material to the environment (Mazumdar, 2002) and the resistance to buckling.

The composite materials present a good alternative for this type of project, since they combine the properties lightness and mechanical resistance. For this reason, in the nautical industry, the composites denominated Glass Fiber Reinforced Polymer (GFRP) are widely used (Mazumdar, 2002). These materials combine low density with excellent mechanical properties, deterioration tolerance, corrosion resistance and fatigue resistance. However, it is known that its quality and cost are strongly influenced by the manufacturing process (Carvalho, 1993; Marinucci, 2011; Shenoi, 1993).

The three methods of obtaining composites most used in the industry are: manual lamination (ML), vacuum lamination (VL) and infusion (I). ML is considered the basic technique, whereas the VL and I methods are most improved and use low pressure to distribute the resin; using a vacuum pump. Among the techniques that use low pressure, the VL technique is the cheapest, while Infusion generates a composite with higher mechanical resistance and better quality. (Airex, 2014 and Lombard, 2015). In addition, when taking into account the health benefits of workers, resource saving and environmental preservation, the Infusion stands out from the other techniques (Riche; Pabon; Cohen, 2009).

It is not simple to analyze the mechanical behavior of a composite material. It is necessary to perform tests and studies in different requests to predict the stress distribution in the material. The application of numerical models makes it possible to estimate in a more complete and precise way the mechanical behavior of these structures.

The Finite Element Method (FEM) is a robust and recognized technique to determine the distribution of effort in complex structures, but depends on a numerical model capable of representing the physical phenomena so that the results have adherence to reality. Thus, a previous study of the properties of the material must be done with care to approximate the real mechanical properties of those in the simulation as much as possible. In the case of composite materials, the characterization of the mechanical properties gains complexity as these properties vary according to the manufacturing methods employed. This paper follows this line and discusses the influence of the manufacturing process on the simulation of composite elements subjected to tensile forces.

2. THEORETICAL FOUNDATION

2.1. Behavior of a Composite Material

According to Gay (1991), it is traditionally considered

> The mechanical characteristics of the reinforcement / matrix junction can be estimated from the characteristics of each of the constituents and each proportion in the composite material. The literature provides several relationships obtained theoretically or semi-empirically, and from which results do not always agree with the values observed in experiments. One of the reasons is that the fibers present, even when isolated, different degrees of anisotropy.

With these considerations, we take the following relations to characterize a unidirectional layer:

Table 1. Properties for a unidirectional layer.

Property	Equation	
Modulus of elasticity towards fibers	$E_l = E_f V_f + E_m \left(1 - V_f\right)$	(1)
Modulus of elasticity in cross-fiber direction	$E_t = E_m \left(\frac{1}{(1-V_f)+\frac{E_m}{E_{ft}}(V_f)}\right)$	(2)
Shear modulus	$G = G_m \left(\frac{1}{(1-V_f)+\frac{G_m}{G_f}(V_f)}\right)$	(3)
Poisson Coefficient	$v_{lt} = v_f V_f + v_m \left(1 - V_f\right)$	(4)

Where V_f is the volumetric fraction of fiber, E_f is the modulus of elasticity of the fiber, E_m is the modulus of elasticity of the matrix, G_f is the shear modulus of the fiber, G_m is the shear modulus of the matrix, v_f is the Poisson coefficient of the fiber and v_m is the Poisson coefficient of the matrix.

Laminate is the result of superposition of several layers of composite, where each layer may have a fiber orientation. To determine the tensions in each layer it is necessary to find the overall stresses in the laminate, calculate the apparent moduli of elasticity of the laminate and the associated coupling coefficients. If the fiber orientation angle in the layer is different from the overall stress angle, a base change is necessary.

With the membrane stresses N_x, N_y, T_{xy} and the moments per unit width M_x, M_y, M_{xy}, it's possible to determine the deformations of the laminate through the ratio shown in Eq. (5) (Gay,1991):

$$
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy} \\
\frac{\partial^2 w}{\partial x^2} \\
\frac{\partial^2 w}{\partial y^2} \\
2\frac{\partial^2 w}{\partial x \partial y}
\end{bmatrix}
=
\begin{bmatrix}
A_{ij} & B_{ij} \\
B_{ij} & C_{ij}
\end{bmatrix}^{-1}
\cdot
\begin{bmatrix}
N_x \\
N_y \\
T_{xy} \\
M_x \\
M_y \\
M_{xy}
\end{bmatrix}
\tag{5}
$$

Where N_x is the normal force per unit length in the x direction, N_y is the normal force per unit length in the y direction, N_{xy} is the shear effort per unit length, M_x is the bending moment per unit length around the x axis, M_y is the bending moment per unit length around the y axis and M_x is the torsional moment per unit length.

The stiffness matrix of the laminate consists of the matrices [A], [B] and [C], which are function of the geometry, material properties and stacking sequence of the blades.

$$A_{ij} = \sum_{k=1^{st} \, layer}^{n \, th \, layer} E_{ij}^k \left(z_k - z_{k-1} \right) \tag{6}$$

$$B_{ij} = \sum_{k=1^{st} \, layer}^{n \, th \, layer} E_{ij}^k \frac{z_k^2 - z_{k-1}^2}{2}) \tag{7}$$

$$C_{ij} = \sum_{k=1^{st} \, layer}^{n \, th \, layer} E_{ij}^k \frac{z_k^3 - z_{k-1}^3}{3}) \tag{8}$$

Where k is the index of each sheet in the laminate, z is the coordinate (along the thickness) relative to one point in the laminate and E_{ij}^k is the stiffness matrix on the k-th sheet.

The deformations of the laminate found by Eq. (5) allow to calculate the tensions in each layer by the following relations:

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix}_{layer \, k} = [E_{layer \, k}] \cdot \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix}_{laminate} \tag{9}$$

Where [E] is the layer stiffness matrix.

Being θ the angle of the fibers in the layer, the stresses in the orthotropy reference of the layer are given by:

$$\begin{bmatrix} \sigma_l \\ \sigma_t \\ \tau_{lt} \end{bmatrix}_{layer \, k} = \begin{bmatrix} \cos^2 \theta & \sin^2 \theta & -2 \cos \theta \sin \theta \\ \sin^2 \theta & \cos^2 \theta & 2 \cos \theta \sin \theta \\ \cos \theta \sin \theta & -\cos \theta \sin \theta & \cos^2 \theta - \sin^2 \theta \end{bmatrix}_{layer \, k} \cdot \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix}_{layer \, k} \tag{10}$$

Where σ_l is the tension in the longitudinal direction to the fiber, σ_t is the tension in transverse direction and τ_{lt} is the shear stress in the plane of the layer.

2.2. Failure Criteria

A wide variety of deformation modes can lead to failure of a composite. The operational failure mode depends, among others, on the loading conditions and the microstructure of the particular composite.

The characterization of the microstructure of a composite consists of the determination of fiber diameter, fiber volume fraction, fiber distribution and possible thermal stresses that can be developed during the manufacturing and / or service process (Chawla, 1998). Therefore, approximate methods for the determination of the failure stress have been created, but still the correct determination of the microstructure of the material remains.

The criterion of maximum tensile strength stipulates that the mechanical strength of the analyzed blade is reached when one of the three stresses to which the blade is subjected reaches the value of the corresponding rupture stress. In this way, the fault then occurs when one of these critical values, σ_{lR}, σ_{tR} e τ_{ltR}, is reached.

There are also other failure criteria for orthotropic materials. The most commonly used for pre-dimensioning calculations is the Tsail-Hill criterion. The Tsai-Hill criterion represents the interaction between the different stress components in failure mechanisms. Experimental observations on isotropic and orthotropic materials reinforced with fibers show that such interactions can affect the failure of the material. This criterion can be interpreted as analogous to the von Mises criterion for isotropic materials (Gurdal, 1999).

According to Gay (1991), for an orthotropic material the principal directions of stresses generally do not coincide with the directions of orthotropy. When the stress state is restricted to the plane defined by the longitudinal and transverse axes of the fibers, we have Eq. (11), named Tsai-Hill criterion:

$$\left(\frac{\sigma_l}{\sigma_{lR}} \right)^2 + \left(\frac{\sigma_t}{\sigma_{tR}} \right)^2 - \left(\frac{\sigma_l \sigma_t}{\sigma_{lR}^2} \right) + \left(\frac{\tau_{lt}}{\tau_{ltR}} \right)^2 \geq 1 \tag{11}$$

Where, σ_l is the tension in the direction of the fiber, σ_t, is the tension in the transverse direction to the fiber, τ_{lt} is the shear stress in the plane of the layer, σ_{lR} is the breaking strength in the longitudinal direction, σ_{tR} is the breaking strength in the transverse direction and τ_{ltR} is the breaking strength by shear in the plane of the layer. Values equal to or greater than 1 indicate rupture of the composite layer.

2.3. Finit Element Method

The Finite Element Method is widely used to solve engineering problems in solid mechanics, heat transfer, structural mechanics, fluid mechanics, etc. (Desai; Eldho; Shah, 2011). According to Soriano (2011), in the FEM, the

domain is divided into a discrete number of subdomains called finite elements, which are connected to one another by nodal points, or simply nodes. According to Soriano (2003) it is possible to classify the different types of elements according to their geometry:

- Unidimensional elements (1D): bars, rectangular beams or curves;
- Two-dimensional elements (2D): flat elasticity (deformation or tension), bending plates, curved shells;
- Three-dimensional elements (3D): elements of volume.

Recently the use of composite elements motivated the development of elements whose formulation incorporates layers with different properties. These elements are called multi-layer elements, which can be 2D or 3D elements.

Table 2 shows other papers that discuss the FEM simulation of elements made with composites.

Table 2. Works that discuss the application of composite materials in Finite Element.

Author	Model/Element	Simulation
Ferreira (2014)	Shell with set and displacement prescribed	Tensile Test
Tita (1999)	Shell multi-layer	Vibration Test
Nurhaniza *et al* (2010)	Shell multi-layer with set	Tensile Test

According to Soriano (2003), modeling is a technical task to be carried out with skill, based on knowledge and experience. Several modeling experiments are required in each new problem studied. A physical system is usually very complex to be analyzed in an exact way, and for this reason, simplifying hypotheses are adopted regarding: the behavior of the material, the bonds, the external actions, among others, to create what is called the mathematical model. Valid assumptions can be made only when there is a qualitative understanding of the structure behavior (Ochoa; Reddy, 1992).

All elements must have their material properties defined. Inserting correct materials properties is one of the critical parts to the success of composite material analysis. Mesh generation is another important modeling step, which includes discretization of the domain in elements and determines the location of nodal points (Desai; Eldho; Shah, 2011). Coarse meshes (with few elements) may differ greatly from reality, while very refined meshes may require inviable processing time.

3. DEVELOPMENT

3.1. Modeling

The numerical model realized in this work allowed the obtaining of the longitudinal tensions, transversal and also of the equivalent tensions in each one of the layers of the test body.

Initially, a theoretical survey of data on the mechanical properties of glass fiber composites was performed, in which case the data of 55% glass fiber composite and 45% epoxy matrix were used. The table 3 shows the values of the mechanical properties used in the models, with E_{long} (GPa) the Young's modulus in the longitudinal direction, E_{tranv} (GPa) the Young's Modulus in the transverse direction, v_{lt} the Poisson's coefficient in the plane of the layer, G_{lt} (GPa) the shear modulus in the plane of the layer, σ_{lR} the breaking strength in the longitudinal direction, σ_{tR} the breaking strength in the transverse direction and τ_{ltR} the shear rupture strength in the plane of the layer.

Table 3. Mechanical properties of composite fiberglass-0,55 / epoxy-0,45. Source: Daniel, 2006.

E_{long}(GPa)	E_{tranv}(GPa)	v_{lt}	G_{lt} (GPa)	σ_{lR}(MPa)	σ_{tR} (MPa)	τ_{ltR}(MPa)
41	10,4	0,28	4,3	1140	39	89

As discussed in the theoretical basis, two types of elements were chosen for the simulations: shell model with multi-layer element and solid model with multi-layer element. The models were created in *Abaqus*[TM] software, where the composite layers were defined with their respective angles and properties. The differences related to the different types of models used will be described in Table 4.

Table 4. Different models utilized in the simulation.

Model / Element	Description
Solid model with multi-layer element	In multi-layer solid element modeling only one solid was extruded with the thickness of the laminate. In the definition of the material a composite section was assigned, in which the 4 layers were divided with the orientation angles of the fibers, as shown in figure 1, and defining the relative thickness of the material with the part. The mesh used was the square with a size of 5mm.
Shell model with multi-layer element	For shell-like elements there are fundamental premises on how the deformation of the material varies through its thickness. These assumptions are necessary to transform a 3D domain into 2D. According to the *Abaqus^{TM}* software instructions, this is the most appropriate model to simulate laminates. With the multi-layer shell element, it's necessary to assign the thickness of each layer only in the definition phase of the composite material. A square mesh with a size of 5mm was created.

The proof bodies were modeled 250 mm long and with 4 layers, being: blanket, tissue 45 °, tissue -45 ° and blanket again, as shown in figure 1. According to the weight of each layer used by Bittencourt (2015), the thickness ratio is as follows: 35% for each blanket and 15% for each layer of fabric. It was also considered as an isotropic material, since its properties do not vary according to the direction taken and, according to Casaril (2007), its properties are worth 37% of the values valid for the fabric.
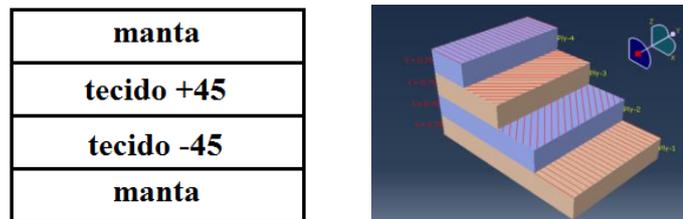

Figure 1. Laminate stacking sequence on the left and the orientation of the fibers in the different layers of the composite in the FEM modeling on the right.

To simulate the tensile test machine claws, the ends on each side of the proof bodies (PB) were embedded at one end and the other side fixed with mobility only in the direction of loading.

4. RESULTS AND DISCUSSION

4.1. Experimental Results

In his work, Bittencourt (2015) carried out 18 traction tests, 6 proof bodies made by ML, 6 proof bodies made by VL and the last 6 proof bodies by I. In the tensile test, the maximum strength was obtained for each of the samples and, based on the average dimensions of thickness and width, the maximum average tensile strength was determined for each one of the samples.

The thickness and width dimensions of each specimen were measured at three points (one at each end and one in the middle). Table 5 synthesizes the results of the assays.

Table 5. Results of tensile tests for the 18 proof bodies tested. Source: Bittencourt, 2015.

PB	Minimum Thickness (mm)	Minimum Width (mm)	Minimun Area (mm²)	Maximum Force (N)	Maximum Stress (MPa)
ML01	3,00	26,40	79,20	9663,90	122,02
ML02	3,20	25,50	81,60	9789,30	119,97
ML03	3,15	25,35	79,85	7794,50	97,61
ML04	3,00	25,00	75,00	9496,80	126,62
ML05	3,70	24,85	91,95	9427,20	102,53
ML06	3,45	25,10	86,60	7115,70	82,17
VL01	2,60	25,60	66,56	6990,30	105,02
VL02	2,70	25,50	68,85	8396,80	121,96

VL03	2,45	24,20	59,29	7227,10	121,89
VL04	2,45	25,30	61,99	7185,30	115,92
VL05	2,45	25,90	63,46	7519,50	118,50
VL06	2,40	25,00	60,00	7432,50	123,88
I01	1,95	25,65	50,02	8174,00	163,42
I02	2,30	25,90	59,57	7603,00	127,63
I03	1,95	25,65	50,02	7853,70	157,02
I04	1,95	26,20	51,09	8174,00	159,99
I05	1,95	25,65	50,02	6607,40	132,10
I06	1,85	25,60	47,36	7143,50	150,83

Analyzing the results, it is observed that the average tensile strength of VL and I was respectively 14.6% and 48.4% higher than ML. Regarding the LV method, the I presented a mean resistance of 29.5% higher. Regarding the confidence intervals, an error of 14% for ML, 7% for VL and 11% for I was identified. Therefore, the infusion method showed a significantly higher tensile strength than the other methods and the vacuum lamination had slightly better than manual rolling.

4.2. Results of the computational analysis using MEF

In total, 18 simulations were performed using MEF, reproducing the proof bodies tested by Bittencourt (2015). Thus, in each simulation, the minimum area and the force observed in the experimental tests were used, and the stresses were evaluated in each layer of the laminate.

According to the criterion of maximum tensile strength, the composite ruptures when the tension in the direction transverse to the direction of the fibers is 39 MPa, this can be observed from the 45° layers, in which the tensions in the transverse direction are close to this limit value. Therefore, the specimens must fail in the direction of 45 ° in relation to the direction of force application.

The finite element model that best reproduced the experimental results was the model with solid element multi-layer, the results of which are compared in the following tables:

Table 6. Comparison between the stresses obtained experimentally and in simulation without considering the effect of the claws.

PB	Maximum stresses (MPa) – Shell with multi-layer element, free-free.				Maximum Stress (MPa) Experimental	Difference
	Blanket	Tissue +/-45° (Longitudinal direction to the fiber)	Tissue +/-45° (Transversal direction to the fiber)	Medium Stress in the PB		
ML01	119,10	130,70	40,94	119,10	122,02	2,39%
ML02	117,10	128,50	40,25	117,10	119,97	2,39%
ML03	95,27	104,60	32,75	95,27	97,61	2,40%
ML04	123,60	135,70	42,48	123,60	126,62	2,39%
ML05	100,10	109,90	34,4	100,10	102,53	2,37%
ML06	80,20	88,04	27,57	80,20	82,17	2,40%
VL01	102,50	112,50	35,24	102,50	105,02	2,40%
VL02	119,00	130,70	40,92	119,00	121,96	2,43%
VL03	119,00	130,60	40,9	119,00	121,89	2,37%
VL04	113,10	124,20	38,89	113,10	115,92	2,43%
VL05	115,70	127,00	39,76	115,70	118,5	2,36%
VL06	120,90	132,70	41,56	120,90	123,88	2,41%
I01	159,50	175,10	54,83	159,50	163,42	2,40%
I02	124,60	136,80	42,82	124,60	127,63	2,37%
I03	153,30	168,20	52,68	153,30	157,02	2,37%
I04	156,20	171,40	53,68	156,20	159,99	2,37%

| I05 | 128,90 | 141,50 | 44,32 | 128,90 | 132,1 | 2,42% |
| I06 | 147,20 | 161,60 | 50,61 | 147,20 | 150,83 | 2,41% |

Table 7. Comparison between the stresses obtained experimentally and in simulation considering the effect of the claws.

| PB | Maximum stresses (MPa) – Shell with multi-layer element, embedded and released in x. | | | | Maximum Stress (MPa) Experimental | Difference |
	Blanket	Tissue +/-45° (Longitudinal direction to the fiber)	Tissue +/-45° (Transversal direction to the fiber)	Medium Stress in the PB		
ML01	126,10	185,00	54,14	126,60	122,02	3,75%
ML02	123,70	180,90	52,98	124,30	119,97	3,61%
ML03	100,60	147,00	43,08	101,10	97,61	3,58%
ML04	130,40	190,30	55,78	131,00	126,62	3,46%
ML05	105,60	153,90	45,13	106,00	102,53	3,38%
ML06	84,67	123,60	36,22	85,03	82,17	3,48%
VL01	108,30	158,40	46,41	108,80	105,02	3,60%
VL02	125,80	183,90	53,86	126,30	121,96	3,56%
VL03	125,40	182,20	53,46	125,90	121,89	3,29%
VL04	119,50	174,50	51,14	120,00	115,92	3,52%
VL05	122,30	179,10	52,44	122,80	118,5	3,63%
VL06	127,60	186,20	54,57	128,20	123,88	3,49%
I01	168,60	246,60	72,23	169,30	163,42	3,60%
I02	131,70	192,90	56,48	132,30	127,63	3,66%
I03	162,00	237,00	69,4	162,70	157,02	3,62%
I04	165,20	242,30	70,91	166,00	159,99	3,76%
I05	136,30	199,30	58,39	136,90	132,1	3,63%
I06	155,60	227,50	66,65	156,30	150,83	3,63%

Table 8. Comparison between the stresses obtained experimentally and in simulation without considering the effect of the claws.

| PB | Maximum Stress (MPa) – Solid with multi-layer element, free-free. | | | | Maximum Stress (MPa) Experimental | Difference |
	Blanket	Tissue +/-45° (Longitudinal direction to the fiber)	Tissue +/-45° (Transversal direction to the fiber)	Medium Stress in the PB		
ML01	119,00	131,20	41,01	119,00	122,02	2,48%
ML02	117,00	128,90	40,33	117,00	119,97	2,48%
ML03	95,23	104,90	32,81	95,23	97,61	2,44%
ML04	123,50	136,10	42,56	123,50	126,62	2,46%
ML05	100,00	110,10	34,47	100,00	102,53	2,47%
ML06	80,16	88,27	27,62	80,16	82,17	2,45%
VL01	102,50	113,00	35,28	102,50	105,02	2,40%
VL02	119,00	131,20	40,98	119,00	121,96	2,43%
VL03	118,90	131,20	40,95	118,90	121,89	2,45%
VL04	113,10	124,80	38,94	113,10	115,92	2,43%
VL05	115,60	127,60	39,8	115,60	118,5	2,45%
VL06	120,90	133,30	41,61	120,90	123,88	2,41%
I01	159,50	176,40	54,81	159,50	163,42	2,40%
I02	124,50	137,50	42,85	124,50	127,63	2,45%

I03	153,20	169,50	52,66	153,20	157,02	2,43%
I04	156,10	172,70	53,66	156,10	159,99	2,43%
I05	128,90	142,60	44,31	128,90	132,1	2,42%
I06	147,20	162,80	50,59	147,20	150,83	2,41%

Table 9. Comparison between the stresses obtained experimentally and in simulation considering the effect of the claws.

PB	Maximum stresses (MPa) – Solid with multi-layer element, embedded and released in x.				Maximum Stress (MPa) Experimental Tissue +/-45° (Longitudinal direction to the fiber)	Difference Tissue +/-45° (Transversal direction to the fiber)
	Blanket	Tissue +/-45° (Longitudinal direction to the fiber)	Tissue +/-45° (Transversal direction to the fiber)	Blanket		
ML01	125,10	188,40	53,69	125,10	122,02	-2,52%
ML02	122,90	184,20	52,55	122,90	119,97	-2,44%
ML03	99,95	149,80	42,72	99,95	97,61	-2,40%
ML04	129,60	193,80	55,32	129,60	126,62	-2,35%
ML05	105,00	156,80	44,75	105,00	102,53	-2,41%
ML06	84,14	125,90	35,91	84,14	82,17	-2,40%
VL01	107,50	161,40	46,03	107,50	105,02	-2,36%
VL02	124,80	187,30	53,43	124,80	121,96	-2,33%
VL03	124,50	185,70	53,03	124,50	121,89	-2,14%
VL04	118,60	177,80	50,73	118,60	115,92	-2,31%
VL05	121,30	182,40	52,02	121,30	118,5	-2,36%
VL06	126,70	189,70	54,13	126,70	123,88	-2,28%
I01	167,20	251,20	71,66	167,20	163,42	-2,31%
I02	130,70	196,50	56,03	130,70	127,63	-2,41%
I03	160,60	241,40	68,85	160,60	157,02	-2,28%
I04	163,80	246,80	70,35	163,80	159,99	-2,38%
I05	135,10	203,10	57,93	135,10	132,1	-2,27%
I06	154,30	231,80	66,12	154,30	150,83	-2,30%

Another characteristic observed is that the failure will occur in the region close to the clamping claw, where the stresses are concentrated, which can be seen in Figure 2. This result was also observed by Bittencourt (2015), who observed that a large part of the proof bodies break next to the claw.
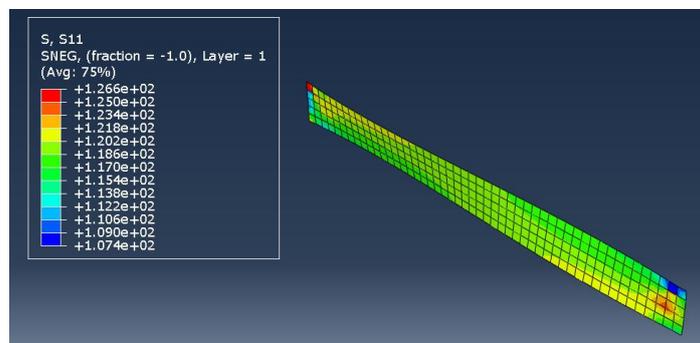

Figure 2. Shell model with multi-layer element with the maximum stress next to the region fixed by the claw.

4.3 Theoretical Results

According to the theory presented by Gay (1991) it is possible to determine the theoretical tensile values in the layers for each of the test bodies tested. A laminate only with tissue was analyzed in the stack sequence [0°/ 45°/-45°/0°]. The results of this analysis are then shown in Tables 10 and 11.

Table 10. Theoretical results of stresses in the layers +/- 45°.

BP	Theoretical Stress (MPa)		Maximum Stress in the tissue (MPa) - Simulation in shell model without claws	
	σ_l MPa	σ_t MPa	σ_l MPa	σ_t Mpa
ML01	50,43	18,78	50,43	18,78
ML02	49,59	18,47	49,59	18,47
ML03	40,34	15,02	40,34	15,02
ML04	52,34	19,49	52,34	19,49
ML05	42,38	15,78	42,38	15,78
ML06	33,96	12,65	33,96	12,65
VL01	43,41	16,16	43,41	16,16
VL02	50,41	18,77	50,41	18,77
VL03	50,38	18,76	50,38	18,76
VL04	47,91	17,84	47,91	17,84
VL05	48,98	18,24	48,98	18,24
VL06	51,20	19,07	51,20	19,07
I01	67,55	25,15	67,55	25,15
I02	52,75	19,64	52,75	19,64
I03	64,90	24,17	64,90	24,17
I04	66,13	24,63	66,13	24,63
I05	54,60	20,33	54,60	20,33
I06	62,34	23,22	62,34	23,22

Table 11. Comparison between the theoretical results and the results of different simulations for longitudinal stress in the layers +/- 45°.

BP	Theoretical Stress - σ_l MPa	Shell model without claws - σ_l Mpa	Difference with the theoretical stress	Shell model with claws - σ_l MPa	Difference with the theoretical stress	Solid model without claws - σ_l MPa	Difference with the theoretical stress	Solid model with claws - σ_l MPa	Difference with the theoretical stress
ML01	50,43	50,43	0%	100,1	98%	49,34	-2%	105,6	109%
VL01	49,59	49,59	0%	84,44	70%	42,63	-14%	90,23	82%
I01	40,34	40,34	0%	133,0	230%	67,47	67%	140,6	249%

The theoretical results and those obtained with the simulation using FEM (without considering the effect of the claws) are identical, using the simulation with multi-layer shell elements, after all, the theoretical model proposed by Gay represents exactly the same idealized situation modeled in the analysis FEM.

5. CONCLUSION

This work presented a comparative study between the tensions observed in experimental tensile tests, simulations using the finite element method and theoretical results.

It was observed that the multi-layer shell element presented results consistent with the experimental results and very close to the theoretical results when use the free-free condition, although the solid model also showed satisfactory results. The results showed that, if the model represents the dimensions of the proof body, the lamination process does not affect FEM results. Thus, due to the adhesion with the real results and the shortest time of modeling and processing, it is recommended to use the shell element type in the analysis of structures using composite materials.

It was also observed that the FEM simulation using both shell and solid multi-layer elements presented an average of 2,4% of variation in the stress value in relation to the tensile tests without considering the effect of the machine claws. Also, an average of 3,6% of variation for shell multi-layer elements and 2,3% for solid multi-layer elements considering the effect of these claws. Thus, it is observed that, in general, as long as it reproduces the final dimensions of the proof body, the FEM simulation presents results consistent with the theoretical calculations and consistent with the experimental results.

As a recommendation for future work, it is suggested to develop a procedure to consider the fraction of voids resulting from each of the manufacturing processes under analysis using FEM, as well as to deepen the influence of the claw in numerical and experimental results, as well as the influence of and the bi-directional tissue present in the proof bodies.

6. REFERENCES

AIREX, B. <http://www. 3accorematerials.com/downloads.html>. Acessado em outubro de 2014.

BARBERO, E. J. Finit Element Analysis of Composite Materials Using Ansys. 2. ed. Danvers: CRC Press, 2013.

BITTENCOURT, A. P. P. ; DUTRA, G. B.; TANCREDI, T. P. Efeito dos processos de laminação sobre as propriedades físicas e mecânicas de compósitos de resina de poliéster com fibras de vidro. Revista Matéria, 2015.

CARVALHO, A. Fiberglass x Corrosão: Especificação, Instalação e Manutenção de Equipamentos de Fiberglass para Ambientes Agressivos, 1 ed., São Paulo: Asplar, 1992.

CASARIL, A.; GOMES, E. R.; SOARES, M. R.; FREDEL, M. C.; AL-QURESHI, H. A. Análise Micromecânica dos compósitos com fibras curtas e partículas. Revista Matéria, 2007.

CHAWLA, K. K. Composite Materials: Science and Engineering, 2 ed., New York, Springer, 1998.

DANIEL, I. M.; ISHAI, O. Engineering Mechanics of Composite Materials, Oxford University Press, New York, 2006.

DESAI, Y. M.; ELDHO, T. I.; SHAH, A. H. Finite Element Method: with Appications in Engineering. New Delhi: Pearson, 2011.

FERREIRA, G. F. O. Estruturas de Material Compósito sob carregamento de Tração e Impacto: avaliação de um modelo de material. 2014. 153 f. Dissertação (Mestrado) - Curso de Mestrado em Engenharia Mecânica, Universidade de São Paulo, São Carlos, 2014. GAY, D., Matériaux Composites, Hermès, Paris, 1991.

GURDAL, Z.; HAFTKA, R. T.; HAJELA, P. Design and Optimization of Laminated Composite Material. New York: John Wiley & Sons, 1999.

LOMBARD, Z. AMT COMPOSITES, <http://www.amtcomposites.co.za/sites/default/files/media/howto/ AMTS-TP15-2011-vacuum%20infusion.pdf >. Acessado em maio de 2015.

MARINUCCI, G. Materiais Compósitos Poliméricos: Fundamentos e Tecnologia, 1 ed., São Paulo, Artliber, 2011.

MAZUMDAR, Sanjay K. Composites Manufacturing: Materials, Product, and Process Engineering. Florida: Crc Press Llc, 2002.

NICOLAIS, L.; MEO, M.; MILELLA, E. Composite materials: a vision for the future. London: Springerverlag, 2011

OCHOA, O. O.; REDDY, J. N. Finit Element Analysis of Composite Laminates. Texas: Springer, 1992.

RICHE, L.; PABON, L.; COHEN, M. "Processo de laminação por Infusão de Barcos de Lazer como Fonte de Sustentabilidade e Vantagem competitiva das Empresas do Setor Náutico Brasileiro", In: XXXIII Encontro da ANPAD, pp. 01-16, São Paulo, Set. 2009.

SORIANO, Humberto Lima. Método de Elementos Finitos em Análise de Estruturas. São Paulo: Editora da Universidade de São Paulo, 2003.

SHENOI, R. A.; WELLICOME, J. F. Composite materials in marine structures: fundamental aspects, 1 ed., New York, Cambridge University Press, 1993.

TITA, V. Análise Dinâmica Teórica e Experimental de Vigas Fabricadas a partir de Material Compósito Poliméricos Reforçados, 1999. 155 f. Dissertação (Mestrado) - Curso de Mestrado em Engenharia Mecânica, Universidade de São Paulo, São Carlos, 1999.

7. RESPONSIBILITY NOTICE

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