Mathematical modelling of the composite materials structure used for leisure crafts

C L Dumitrache¹, R Dumitrache¹, C Faitar² and N Buzbuchi²
¹Constanta Maritime University, Faculty of Navigation and Naval Transport, Department of Navigation, 104 Mircea cel Batran Street, Constanta, 900663, Romania
²Constanta Maritime University, Faculty of Naval Electromechanics, 104 Mircea cel Batran Street, Constanta, 900663, Romania
E-mail: cosmin_dumitrache@yahoo.com

Abstract. For a leisure craft, chosen materials are mainly made of fibre reinforced composite. This type of materials have more advantages than disadvantages, having a good resistance to corrosion, resistance to impact, are reducing vibrations, are light and are easy to use during construction including maintenance and repairing. In the practice of using composite materials, the most common structural element is the lamina. Within the calculation of a composite material structure, there can be various approaches: to ignore the layering of the composite laminae, considering them isothropic or orthotropic and they can be calculated as standard metallic structures or to consider the layering of the composite laminae, but to ignore the exact orientation of fibers into the matrix. For the calculation with finite element method is used COSMOSM/MATLAB software and it includes several types of laminae, with or without layers. It is carried out a comparison of the results from COSMOSM software, MATLAB software and finite element, being observed a good consistency between the results of the software and the calculated results by other authors. It is worth noting that the study of the specialty literature shows that several programs have been designed to calculate the strength of laminates, but they are meant for internal use. Given that the strengths of laminates can be efficiently calculated, although the calculations may be complicated, it becomes obvious the importance to design such a software program.

1. Introduction
Materials generally used in ships construction are steel and aluminium. Based on evolution and advancement of technology to produce composite materials, they began to replace the traditional materials, especially in small craft construction, in medium size vessels and sometimes in the structures of large vessels, where stresses are not reaching high values. For leisure crafts, materials that are made of are mainly fibre reinforced composite materials. The composites show many advantages, being light, resistant to corrosion, resistant to impact, capable of vibration reduction and of course, it is very easy to use them in the construction process, to maintain and to repair. The composite materials are made of two main components: the fibres and the matrix. The fibre represents the reinforcement material and the matrix represents the binding agent in which the fibres are steeped in order to create together, after the polymerization process, the composite material. Three approaches may be used in the calculation of mechanical behaviour of composite materials structures:

- Ignoring of the layers of the composites plates - the plates are considered either isotropic or orthotropic and the calculation applied is as the one for a metallic structure.
- The layering of composite plate is taken into account but the fibre direction within the matrix is ignored – such a suggestion needs a greater calculation effort and much more precise information about the material.
- Taking into account the direction of the fibres into the matrix and running at micro mechanical level – it has a very high accuracy and the calculated effort is considerably higher. Unfortunately, the method cannot be applied to complex structures [1].

2. Calculation methods for composite material structures
In the practice of using composite materials the most frequent structural elements are the laminae. For this reason, this chapter will show the calculation of a composite laminae material structure. There are at least three approaches to the calculation of composite material structures:

- The layering of composite laminae might be ignored, in which case it is considered that these laminae are either isotropic or orthotropic and are calculated just as the standard metallic structures; Surely, stresses should not occur in the proximity of the material failure limits; this approach is the simplest and the more accessible when attempting to calculate complicated structures such as vessels; if the material doesn’t have a quasi-isotropic or a quasi-orthotropic behavior, then this approach is not appropriate.
- The layering of composite laminae is taken into account, but the exact orientation of the fibers into the matrix is being ignored; this approach is surely more accurate than the previous one, but it implies a bigger calculation effort and more accurate information about the way in which the material was made; in this approach there is no need for the material to be quasi-isotropic or quasi-orthotropic.
- The operation method targets the micro-mechanic level and it takes into account the exact distribution of the fibers within the matrix. The accuracy is high, the calculation effort is big, and the method is practically inapplicable to complex structures.

Normally, when it comes to structures, methods A or B are employed, based on material characteristics, on the existing information about the material, on the complexity of the structure. Next we will take into account only approach B. Approach A is merely a particular case of approach B. Several theories have been used until now, as classified below [2]:

1. Monolayer equivalent theories (2D):
   - Kirchhoff’s classic theory of laminated plates.
   - Laminated plates’ theories that take shearing into account (Mindlin).
2. Three-dimensional elasticity theory (3D):
   - Traditional definition of 3-D elasticity.
   - Theories that account for material layering.
3. Mixed theories (2D and 3D)

2.1. Calculation hypothesis
The used calculation hypotheses are considered to be known. Only some additional remarks will be made here. The coordinate axes, the forces and the moments are shown in the figures 1, 2 and 3:
It is assumed that each lamina layer is orthotropic, linear-elastic and of constant thickness. The thickness is considered very small when compared to the length and width of the structural element. It is assuming that the stresses occur in the $x-y$ plane of the laminate, namely, it is assumed that the structural element is under plane stress:

$$
\sigma_z = 0, \quad \tau_{xz} = 0, \quad \tau_{yz} = 0
$$

(1)

It is assumed that the corresponding strains are also negligible, namely:

$$
ev_z = 0, \quad \gamma_{xz} = 0, \quad \gamma_{yz} = 0
$$

(2)

This hypothesis group is conflicting, but this too is neglected. For instance, although the transverse shear strains $\gamma_{xz}, \gamma_{yz}$ (2) lead to zero transverse shear stresses, the computed values of these stresses and their resultants $Q_x, Q_y$ are other than zero. However, this does not stop the Kirkchoff thin plate theory from being widely used in practical work. In this respect, the classic Euler-Bernoulli beam theory is widely used in structure calculations, although in that case the calculation hypotheses are even more conflicting. Moreover, it is assumed that the relations between strains and moments are linear. If we assume that the resultant forces and the distribution forces act upon an infinitesimal element within the structure, then we can write down the equilibrium equations (of forces and moments):

$$
\frac{\partial N_x}{\partial x} + \frac{\partial N_y}{\partial y} = 0, \\
\frac{\partial N_y}{\partial x} + \frac{\partial N_x}{\partial y} = 0, \\
\frac{\partial Q_x}{\partial x} + \frac{\partial Q_y}{\partial y} + q_z = 0, \\
\frac{\partial M_y}{\partial x} - \frac{\partial M_{yx}}{\partial y} - Q_x = 0,
$$

(3)
Since the presence of shear forces resultants, $Q_x$, $Q_y$ (3) unnecessarily complicate problem solving, the last three equations may be used in order to eliminate them. It is obtained a set of equilibrium equations, which is the starting point in the use of the finite element method [2]:

$$\frac{\partial M_{xy}}{\partial x} + \frac{\partial M_y}{\partial y} - Q_x = 0,$$

$$\frac{\partial N_x}{\partial x} + \frac{\partial N_{xy}}{\partial y} = 0,$$

$$\frac{\partial N_{xy}}{\partial x} + \frac{\partial N_y}{\partial y} = 0,$$

$$\frac{\partial^2 M_x}{\partial x^2} + 2\frac{\partial^2 M_{xy}}{\partial x \partial y} + \frac{\partial^2 M_y}{\partial y^2} + q_x = 0.$$

Upon solving this equation group (4), the shear forces resultants $Q_x$, $Q_y$ may also be computed using the relations:

$$Q_x = \frac{\partial M_x}{\partial x} + \frac{\partial M_{xy}}{\partial y},$$

$$Q_y = \frac{\partial M_{xy}}{\partial x} + \frac{\partial M_y}{\partial y}.$$

3. Calculation of composite structures by the finite element method

The finite element method is a numerical method intended to solve differential equations with limiting conditions. Besides its use to solve both simple and complex nonlinear problems, there is also the advantage of its implementation over various steps that are well documented in the specialty literature.

Moreover, the use of programs based on the finite element method does not require significant theoretical knowledge. For instance, a user of such program, who intends to solve an elasticity problem, doesn’t necessarily have to know the elasticity equations. The use of such software implies a mere description (drawing) of the material and a description of the loads, followed by the reading of results. Because of this, the method is widely used in the engineering practical work. The method has a wide applicability, for instance: structure mechanics, plastic deformation processing, geomechanics, bioengineering, etc. Several problems can be solved, such as linear and non linear, stationary and non-stationary, applied to various types of calculations, finite or infinite, and with various types of limiting conditions. Although the method has been officially presented in the mid 50’s, it is still being developed, and thousands of pages with further theoretical, numerical, algorithmical or applicative developments are being published annually.

At the present moment the method is included in the Generalized Finite Element Method, along with the other two widely spread discretization methods: the finite difference method and the boundary element method [3].

3.1. Types of finite elements implemented in finite element programs

There are various programs for the general use, which include types of finite elements intended for composite structure calculations. It is worth noting that, if the used laminae are not bound, and the calculation doesn’t lead to stresses that exceed the limits, then there is no need to use a special type of finite element (layered), and standard types of finite elements which don’t account for laminae layering are enough. Elements intended for the analysis of composite material structures are said to be linear if they don’t allow layers’ failure, while they are said to be non-linear if they allow layers’ failure and the change of the mechanical characteristic of the laminae during the calculation. For
instance, the ANSYS software provides several non-layered plates: SHELL43, SHELL63, SHELL93, the first two being 4-noded elements, and the last being 8-noded element. The first two don’t account for shearing, while the last one does [4].

For layered plate calculations ANSYS provides the elements SHELL91, SHELL99 and SHELL181. The first two are 8-noded elements, while the third is a 4-noded element. The first allows non-linear calculation, the second doesn’t, while the third also allows it. COSMOSM software provides more elements:

- non layered plates which don’t include shearing: triangular elements SHELL3, SHELL6 and quadrangular elements SHELL4 and SHELL9,
- non layered plates which include shearing: SHELL3T (triangular), SHELL4T (quadrangular)
- layered plates SHELL3L (triangular), SHELL4L and SHELL9L (quadrangular), where the first two may be used in non linear calculations.

3.2. Validation using experimental results or results derived from other authors

To assess the program’s ability to compute close to real values for the laminae strength, systematic calculations were performed on a special type of cross-ply laminate. For this special type of laminate both computed values and measured results are available in the speciality literature. Next, a special type of cross-ply laminate is given, which is made of an odd number $N$ of unidirectional layers. The layers are placed at 0 and 90 degrees alternatively. The layers with odd numbers are placed in the direction $x$, while the layers with even numbers are placed in the direction $y$. Additionally, it is considered that the layers with the same orientation also have the same thickness. It is denoted $M$ the ratio between the sum of odd thicknesses and the sum of even thicknesses. The computation $N = 3$ is made and $M$ varies from 0.2 to 4. The program was used at $\Delta T = 0^\circ F$ and the following results were obtained:

|M| Layer | Mode | N (lbf/in) | N/t (psi) | eps_0 (%) |
|---|---|---|---|---|---|
| 0.20000 | 2 | 1 | 3.21916e+002 | 5.36527e+003 | 1.54255e-001 |
| | 1 | 1 | 7.21871e+002 | 1.20312e+004 | 9.07375e-001 |
| | 3 | 1 | 7.21871e+002 | 1.20312e+004 | 9.07375e-001 |
| | 2 | 2 | 1.70000e+003 | 2.83333e+004 | 2.13408e+000 |
| | 3 | 2 | 1.70000e+003 | 2.83333e+004 | 2.13408e+000 |

|M| Layer | Mode | N (lbf/in) | N/t (psi) | eps_0 (%) |
|---|---|---|---|---|---|
| 0.25000 | 2 | 1 | 3.38360e+002 | 5.63933e+003 | 1.54349e-001 |
| | 1 | 1 | 8.26145e+002 | 1.37691e+004 | 8.65632e-001 |
| | 3 | 1 | 8.26145e+002 | 1.37691e+004 | 8.65632e-001 |
| | 2 | 2 | 1.99200e+003 | 3.32000e+004 | 2.08386e+000 |
| | 3 | 2 | 1.99200e+003 | 3.32000e+004 | 2.08386e+000 |

|M| Layer | Mode | N (lbf/in) | N/t (psi) | eps_0 (%) |
|---|---|---|---|---|---|
| 0.40000 | 2 | 1 | 3.80757e+002 | 6.34595e+003 | 1.54617e-001 |
| | 1 | 1 | 1.11086e+003 | 1.85144e+004 | 8.15464e-001 |
| | 3 | 1 | 1.11086e+003 | 1.85144e+004 | 8.15464e-001 |
| | 2 | 2 | 2.74286e+003 | 4.57143e+004 | 2.00854e+000 |
| | 3 | 2 | 2.74286e+003 | 4.57143e+004 | 2.00854e+000 |
For the sake of comparison with the results derived from the specialty literature, the following were extracted:

- $A_{l}^{\text{init}}$ rigidity $A_l$ before the first failure
- $A_{l}^{\text{fin}}$ rigidity $A_l$ before the last failure
- $N_l$ the maximum force allowed by the laminate

All these dimensions have been scaled with the thickness of the laminate $t$.

For the sake of comparison with the results derived from the specialty literature, the following were extracted:

$A_{l}^{\text{init}}$ rigidity $A_l$ before the first failure

$A_{l}^{\text{fin}}$ rigidity $A_l$ before the last failure

$N_l$ the maximum force allowed by the laminate

Then the program was used at $\Delta T = -200^\circ F$ as well and the following results were obtained:
| Layer | Mode | N (lbf/in) | N/t (psi) | eps_0 (%) |
|-------|------|------------|-----------|-----------|
| 2     | 1    | 2.01372e+002 | 3.35619e+003 | 9.18595e-002 |
| 1     | 1    | 2.01372e+002 | 3.35619e+003 | 9.18595e-002 |
| 3     | 1    | 2.01372e+002 | 3.35619e+003 | 9.18595e-002 |
| 2     | 2    |             |           |           |
| 1     | 2    | 1.91427e+003 | 3.19045e+004 | 2.00255e+000 |
| 3     | 2    | 1.91427e+003 | 3.19045e+004 | 2.00255e+000 |

M = 0.400000

| Layer | Mode | N (lbf/in) | N/t (psi) | eps_0 (%) |
|-------|------|------------|-----------|-----------|
| 2     | 1    | 1.86177e+002 | 3.10296e+003 | 7.56024e-002 |
| 1     | 1    | 2.02313e+002 | 3.37189e+003 | 1.48514e-001 |
| 3     | 1    | 2.02313e+002 | 3.37189e+003 | 1.48514e-001 |
| 2     | 2    |             |           |           |
| 1     | 2    | 2.65525e+003 | 4.42542e+004 | 1.94439e+000 |
| 3     | 2    | 2.65525e+003 | 4.42542e+004 | 1.94439e+000 |

M = 1.000000

| Layer | Mode | N (lbf/in) | N/t (psi) | eps_0 (%) |
|-------|------|------------|-----------|-----------|
| 2     | 1    | 1.53584e+002 | 2.55974e+003 | 4.89653e-002 |
| 1     | 1    | 4.99630e+002 | 8.32716e+003 | 2.10163e-001 |
| 3     | 1    | 4.99630e+002 | 8.32716e+003 | 2.10163e-001 |
| 2     | 2    |             |           |           |
| 1     | 2    | 4.53779e+003 | 7.56298e+004 | 1.89882e+000 |
| 3     | 2    | 4.53779e+003 | 7.56298e+004 | 1.89882e+000 |

M = 2.500000

| Layer | Mode | N (lbf/in) | N/t (psi) | eps_0 (%) |
|-------|------|------------|-----------|-----------|
| 2     | 1    | 1.33860e+002 | 2.23099e+003 | 3.51724e-002 |
| 1     | 1    | 1.57772e+003 | 2.62953e+004 | 4.66552e-001 |
| 3     | 1    | 1.57772e+003 | 2.62953e+004 | 4.66552e-001 |
| 2     | 2    |             |           |           |
| 1     | 2    | 6.44395e+003 | 1.07399e+005 | 1.88752e+000 |
| 3     | 2    | 6.44395e+003 | 1.07399e+005 | 1.88752e+000 |

M = 4.000000

| Layer | Mode | N (lbf/in) | N/t (psi) | eps_0 (%) |
|-------|------|------------|-----------|-----------|
| 2     | 1    | 1.32291e+002 | 2.20485e+003 | 3.24946e-002 |
| 1     | 1    | 2.65178e+003 | 4.41963e+004 | 7.01874e-001 |
| 3     | 1    | 2.65178e+003 | 4.41963e+004 | 7.01874e-001 |
| 2     | 2    |             |           |           |
| 1     | 2    | 7.20989e+003 | 1.20165e+005 | 1.88560e+000 |
| 3     | 2    | 7.20989e+003 | 1.20165e+005 | 1.88560e+000 |

The values $A_{i}^{\text{initial}}$ and $A_{i}^{\text{final}}$ are the same as in the previous case, while the values $N_{i}$ are very close to those from the previous case.

$$\frac{N}{t} = \left\{ 27149.0 \ 31904.0 \ 44254.0 \ 75630.0 \ 107400.0 \ 120166.0 \right\} \text{psi}$$
Figure 4 explains the correlation between rigidities and strengths calculated in MATLAB and the measured values presented in. It is worth noting that the computed rigidities are very close to the measured ones. Consequently, if a linear-elastic calculation is intended, then these rigidities may be used without performing direct measurements of the laminate.

At the same time, it is observed that the values of the calculated strength are a little higher than the measured ones, but the maximum difference remains under 11%. At the same time, it is noticed that the measured values have a fairly high dispersion, generally higher than the deviation of the calculated values from the measured values.

In other words, if more accurate strength values are preferred, then the measuring procedure should be very rigorous and it should include several repeated measurements. Given that the calculated values are high quality and imply a minimum effort and that the measured values have a high dispersion and imply a considerable effort, it should be clear which approach is chosen in practice.

![Figure 4. Rigidities and strengths of cross-ply laminates. [5]](image)

| M    | Measured Strengths Nx/t [psi/10^3] | Error [%] |
|------|-----------------------------------|-----------|
|      | Min | Med | Max | Calc |           |
| 0.20 | 26.16 | 29.51 | 32.85 | 27.15 | -8.69     |
| 0.25 | 16.15 | 29.52 | 37.39 | 31.90 | 7.46      |
| 0.40 | 33.68 | 39.40 | 40.96 | 44.25 | 10.96     |
| 1.00 | 52.03 | 69.94 | 76.03 | 75.63 | 7.53      |
| 2.50 | 96.15 | 101.86 | 106.78 | 107.40 | 5.16      |
| 4.00 | 103.86 | 111.23 | 116.84 | 120.12 | 7.39      |
Table 1 contains the measured values (minimum, medium and maximum), the calculated values and the RELATIVE error calculated with respect to the medium measured value. It can be noticed in the above table that the RELATIVE maximum error is 11%. To the end of verifying the MATLAB program, it was calculated the strength of an “angle-ply” laminate with three equal thickness layers, placed at a constant angle $\pm \alpha$. More precisely, the orientations were: $\pm \alpha, \mp \alpha, \alpha$. Angle $\alpha$ varied in the range $[0^{\circ}; 90^{\circ}]$ and in each case the strength was calculated. The mechanical properties of the layers were identical with the ones from the previous problem, while the thicknesses of the layers were 0.005 in, which means that the thickness of the laminate was 0.015 in.

The values calculated in MATLAB were compared with the results found in the specific literature. The values calculated in MATLAB are practically identical to those calculated in and are very close to those measured in. The calculated values and the measured ones are represented in the figure 5. The calculations presented above are performed based on the methodology proposed by Tsai and picked up by Jones. If the newer (and more rational) methodology proposed for instance is used, the following results are obtained:

The same results are obtained for the cross-ply laminate, irrespective of the temperature:

$$\frac{N}{t} = (25000.0, 30000.0, 42857.0, 75000.0, 107144.0, 120000.0), \text{ psi}$$

For the angle-ply laminate, at angles:

$\alpha = (15.0, 30.0, 45.0, 60.0, 75.0, 90.0)$, degrees

at temperature $\Delta T = 0^\circ$ the strenghts.
And at temperature $\Delta T = -200^\circ$ the strengths:

$$\frac{N_c}{t} = \begin{pmatrix} 37087.0 \\ 16875.0 \\ 8673.2 \\ 5332.7 \\ 4248.6 \\ 4000.0 \end{pmatrix}, \text{ psi are obtained}$$

3.3. Validation using finite element analysis results

For an additional verification, the cross-ply laminate was analysed through the finite element method. The generic COSMOSM finite element program was used to calculate the strength of the laminate. Since the COSMOSM software uses the Tsai-Wu failure criterion the calculations shown above were performed again using this criterion. Additionally, in MATLAB, the unstressed (unloaded) initial state was used as reference every step of the way during the load. In order to use a generic finite element program to calculate the strength of a laminate, a section of finite dimensions is extracted from the laminate, which is then subject to test forces and limiting conditions. This section is hereafter referred to as “structure”. The forces are applied incrementally (small steps) and the gradual failure of the structure is being observed. The occurred stresses depend upon the way in which the forces are applied, on the limiting conditions and shape of the structure. All these three factors may act as stress concentrators. Consequently, the way in which the forces are applied, the shape of the domain and the type of limiting conditions are established in such a way as to ensure that the stress/strain distribution corresponds to the theoretical one, in order to be able to compare results. Generic finite element programs don’t accurately identify the forces at which the matrix or the fibers fail within the various layers. Certain limits (lower and upper) are obtained for the said forces, which fully depend upon the force increment sizes. The calculation error of failing forces is a force increment at most.

A strength calculation is usually concluded by an error message or a notification on the structure’s failure. For the calculation it was considered that the laminate (structure) is square and 1 in x 1 in in size. The calculation domain (square) was discretized in four finite elements (see figure 6). In order to simulate the theoretical deformation state, spins were blocked in all nodes. Additionally, on the vertical side from $x=0$ all translations were blocked in direction $x$. On the vertical side from $x=1$ a (fictional) a pressure of 1 lb/in2 was applied. The pressure 1 lb/in2 was then multiplied by incremental factors in order to simulate incremental load. Then, 100 loading steps, 30lb/in2 each, were applied to the structures. Namely, were applied incrementally 30 lb/in2, 60 lb/in2, 90 lb/in2, ... Naturally, at a certain point, the program displayed a notification stating that the structure has failed and the calculation has eventually been stopped.

![Figure 6. Discretization in laminate’s finite element.](image)
Given the calculation domain shape, the limiting conditions and the way in which the loads were applied, the stresses and the strains in all four elements are identical. After calculation with COSMOSM we go through the results list and identify the loading steps at which failures occurred.

At load step number 178, for element 1 there are:

StressCalculation for Element Group No. 1, Step No. 178
Element Group ..... = Nonlinear
Type of Nonlinearity = Material
Material Model ..... = Tsai-Wu failure criterion
Element Type ..... = Four Node Composite Shell

ELEMENT NUMBER = 1

CNTR FRC NX NY NXY MX MY MXY VX VY
(ECS:-1) 3.20E+02 7.11E-15 1.24E-15 -2.98E-07 2.63E-18 -1.15E-24 0.00E+00 0.00E+00

LAYER STATE CNTR SIGMA-X SIGMA-Y TAU-XY TAU-XZ TAU-YZ FAIL CRI.
NO. STR
3  OK  TOP 1.213E+04 6.370E+02 -9.992E-14 0.000E+00 0.000E+00 1.90207E-02
3  BOT 1.213E+04 6.370E+02 -9.992E-14 0.000E+00 0.000E+00 1.90207E-02

INTERLAMINAR SHEAR STRESSES TAUZ = 0.00000E+00 TAUZX = 0.00000E+00
2  OK  TOP 3.981E+03 -1.274E+02 4.471E-14 0.000E+00 0.000E+00 9.91425E-01
2  BOT 3.981E+03 -1.274E+02 4.471E-14 0.000E+00 0.000E+00 9.91425E-01

INTERLAMINAR SHEAR STRESSES TAUZ = 0.00000E+00 TAUZX = 0.00000E+00
1  OK  TOP 1.213E+04 6.370E+02 -9.992E-14 0.000E+00 0.000E+00 1.90207E-02
1  BOT 1.213E+04 6.370E+02 -9.992E-14 0.000E+00 0.000E+00 1.90207E-02

At loading point 179 the displayed values for element 1 are:

StressCalculation for Element Group No. 1, Step No. 179
Element Group ..... = Nonlinear
Type of Nonlinearity = Material
Material Model ..... = Tsai-Wu failure criterion
Element Type ..... = Four Node Composite Shell

ELEMENT NUMBER = 1

CNTR FRC NX NY NXY MX MY MXY VX VY
(ECS:-1) 3.22E+02 -3.55E-15 -3.52E-15 -3.00E-07 -2.03E-17 3.28E-24 0.00E+00 0.00E+00

LAYER STATE CNTR SIGMA-X SIGMA-Y TAU-XY TAU-XZ TAU-YZ FAIL CRI.
NO. STR
3  OK  TOP 3.222E+04 2.517E+03 3.714E-12 0.000E+00 0.000E+00 3.06981E-01
3  BOT 3.222E+04 2.517E+03 3.714E-12 0.000E+00 0.000E+00 3.06981E-01

INTERLAMINAR SHEAR STRESSES TAUZ = 0.00000E+00 TAUZX = 0.00000E+00
2  FAIL1 TOP 1.053E-04 -5.034E+02 -8.133E-13 0.000E+00 0.000E+00 2.00000E+00
2  BOT 1.053E-04 -5.034E+02 -8.133E-13 0.000E+00 0.000E+00 2.00000E+00

INTERLAMINAR SHEAR STRESSES TAUZ = 0.00000E+00 TAUZX = 0.00000E+00
1  OK  TOP 3.222E+04 2.517E+03 3.714E-12 0.000E+00 0.000E+00 3.06981E-01
1  BOT 3.222E+04 2.517E+03 3.714E-12 0.000E+00 0.000E+00 3.06981E-01
It is noticed that on the middle layer of the laminate the value of the failure criterion is close to limit value 1. Thus, the first failure occurs between load step 178 and 179, namely, between stresses 178*30 lb/in^2 and 179*30 lb/in^2, in other words between stresses: 5340 lb/in^2 and 5370 lb/in^2. The value 5340 is also verified from the result list. To this end, the value NX is calculated as being 5340*0.06 = 320.4 lb/in. The stress values obtained are consistent with those calculated in MATLAB software. In MATLAB the stress value 5363.0 lb/in^2 was obtained for the first failure. The stresses from the first and second layer before failure are also determined:

\[
\begin{align*}
\sigma_{yy}^{(1)} &= \begin{pmatrix} 12187.0 \\ 639.73 \\ -4.5878 \times 10^{-15} \end{pmatrix}, \\
\sigma_{yy}^{(2)} &= \begin{pmatrix} 3998.3 \\ -127.95 \\ -2.5358 \times 10^{-13} \end{pmatrix}
\end{align*}
\]

and after the failure:

\[
\begin{align*}
\sigma_{yy}^{(1)} &= \begin{pmatrix} 32178.0 \\ 2513.9 \\ 1.5393 \times 10^{-13} \end{pmatrix}, \\
\sigma_{yy}^{(2)} &= \begin{pmatrix} 0.010517 \\ -502.78 \\ -7.8299 \times 10^{-19} \end{pmatrix}
\end{align*}
\]

Next, we go again through the result list within COSMOSM software and at step 323 we encounter a new failure.

Stress Calculation for Element Group No. 1, Step No. 323
Element Group ...... = Nonlinear
Type of Nonlinearity = Material
Material Model ...... = Tsai-Wu failure criterion
Element Type ...... = FourNodeCompositeShell

\[
\begin{align*}
\text{ELEMENT NUMBER} &= 1 \\
\text{CNTR FRC NX NY NXY MX MY MXY VX VY} &\text{(ECS:-1) 5.81E+02 3.70E-27 1.18E-16 -5.41E-07 -3.67E-17 -1.10E-25 0.00E+00 0.00E+00} \\
\text{LAYER STATE CNTR SIGMA-X SIGMA-Y TAU-XY TAU-XZ TAU-YZ FAIL CRI.} &\text{NO. STR} \\
3 &\text{OK TOP 5.814E+04 4.542E+03 7.350E-12 0.000E+00 0.000E+00 9.99563E-01} \\
3 &\text{BOT 5.814E+04 4.542E+03 7.350E-12 0.000E+00 0.000E+00 9.99563E-01} \\
\text{INTERLAMINAR SHEAR STRESSES TAUZ} &= \begin{pmatrix} 0.00000E+00 \\ 0.00000E+00 \end{pmatrix} \\
2 &\text{FAIL TOP 1.900E-04 -9.084E+02 -1.468E-12 0.000E+00 0.000E+00 2.00000E+00} \\
2 &\text{BOT 1.900E-04 -9.084E+02 -1.468E-12 0.000E+00 0.000E+00 2.00000E+00} \\
\text{INTERLAMINAR SHEAR STRESSES TAUZ} &= \begin{pmatrix} 0.00000E+00 \\ 0.00000E+00 \end{pmatrix} \\
1 &\text{OK TOP 5.814E+04 4.542E+03 7.350E-12 0.000E+00 0.000E+00 9.99563E-01} \\
1 &\text{BOT 5.814E+04 4.542E+03 7.350E-12 0.000E+00 0.000E+00 9.99563E-01}
\end{align*}
\]

At the following step, no. 324, COSMOSM software displays:

Stress Calculation for Element Group No. 1, Step No. 324
Element Group ...... = Nonlinear
Type of Nonlinearity = Material
Material Model ...... = Tsai-Wu failure criterion
Element Type ...... = FourNodeCompositeShell

\[
\begin{align*}
\text{ELEMENT NUMBER} &= 1 \\
\end{align*}
\]
At step no. 323 the matrix of the first and the third layer fails. It is noticed that at the first and the third layer the failure criterion is very close to the critical value 1. The stress corresponding to the failure is comprised in the range 323*30 and 324*30 lb/in², namely between 9690 and 9720 lb/in². The stresses applied on the layers before failure are derived using MATLAB software.

Going through the result list provided in COSMOSM we notice that, at step no 816, a message is displayed in the end, which reads:
Stop: The material is considered to be "failed" due to large strain (> 100%) for element 1 layer 1 time step 816

So, the failure stress, according to the COSMOSM software, is comprised between 815*30 and 816*30 lb/in², namely 24450 and 24480 lb/in². MATLAB provides the value 25000.0 lb/in² for the failure stress. The corresponding stresses from the first layer are:

\[
\sigma_{xy}^{(1)} = \begin{bmatrix} 58153.0 \\ 4543.2 \\ 2.7819 \times 10^{-13} \end{bmatrix}, \quad \sigma_{xy}^{(2)} = \begin{bmatrix} 0.019006 \\ -908.64 \\ -1.415 \times 10^{-18} \end{bmatrix}
\]

and failure value 9692.1 lb/in². After the failure, the stress values are:

\[
\sigma_{xy}^{(1)} = \begin{bmatrix} 58153.0 \\ 4.8461 \times 10^{-9} \\ 3.8043 \times 10^{-20} \end{bmatrix}, \quad \sigma_{xy}^{(2)} = \begin{bmatrix} 0.019384 \\ -9.6921 \times 10^{-10} \\ -1.1793 \times 10^{-18} \end{bmatrix}
\]

In conclusion, it is noticed a very good consistency between the results provided by MATLAB software and the results obtained by applying the finite element method and the COSMOSM program.

4. Conclusions
The finite element method is universally accepted for structure calculation, in general, and for composite material structures in particular. The finite element method is a numerical method with rapidly convergent results and even with exact results which numerically solve differential equations with limiting conditions. The method of finite elements can be used for practical calculations and does not require a thorough knowledge of differential equations which describe the studied load, nor the thorough knowledge of the theory upon which the finite element in based, but such work could lead to
erroneous results which, unfortunately, might be considered valid by an inexperienced user. It was noticed a good consistency between the results provided by the MATLAB software proposed in the thesis, the results of the measurements and the results calculated by other authors. For the sake of additional verification, the finite element method was also used with the same results. When comparing the effort put in by a user to calculate the strengths of a laminate in MATLAB, as proposed in this thesis, with the effort put up to use finite element modelling, it is noticed that the proposed software is clearly more advantageous. Besides the modelling effort, cost is also an indicator: zero cost in the case of MATLAB use and a considerable cost needed to purchase a software based on the finite element method. The second cost is so high, that is out of the question to purchase such software solely to use it to calculate the strengths of laminates. It is worth noting that the study of the specialty literature shows that several programs have been designed to calculate the strength of laminates, but they are meant for internal use (they are not for sale). Given that the strengths of laminates can be efficiently calculated, although the calculations may be complicated, it becomes obvious the importance to design such a software program.

5. References

[1] Dumitrache R 2013 Research on the Mechanical Behaviour of Materials Used in Construction of Leisure Crafts (PhD Thesis) (Constanta: Constanta Maritime University) p 244
[2] Reddy I N 2004 Mechanics of laminated composite plates and shells: theory and analysis 2nd ed. CRC PRESS p 835
[3] Gavrilescu I 1997 Modelling of naval structures by plate in elasto-plastic domain The Annals of “Dunarea de Jos” University of Galati Applied Mechanics Fascicle X
[4] Barbero E J 2014 Finite Element Analysis Of Composite Materials 2nd Ed. CRC Press Taylor & Francis Group p 328
[5] Tsai S W 1968 Strength Theories of Filamentary Structures in Fundamental Aspects of Fiber Reinforced Plastic Composites Conference Proceedings Wiley-Interscience New York pp 3-11

Acknowledgement

“This work is supported by the project ANTREPRENORDOC, in the framework of Human Resources Development Operational Programme 2014-2020, financed from the European Social Fund under the contract number 36355/23.05.2019 HRD OP /380/6/13 – SMIS Code: 123847.”