Combined Researches for Validation of a New Finite Element for Modelling Fiber Reinforced Laminated Composite Plates

NICOLAE ILIESCU, ANTON HADAR*, STEFAN DAN PASTRAMA
University "Politehnica" of Bucharest, Department of Strength of Materials, 313 Splaiul Independenþei, 060042, Bucharest, Romania

The paper presents the results of a combined numerical and experimental study for validation of a new three-dimensional specialized finite element, used for modelling the behaviour of laminated composite materials, reinforced with fibers. First, the stress and strain state in a circular plate loaded with an uniformly distributed force and fixed on the contour is evaluated using the proposed finite element. Two types of laminated composites are considered, in order to test the behaviour of the proposed element and the models conceived for calculus of laminated composites having different structures. To validate the new element, calculations were made also using the finite element code ANSYS-11. In the second part, the results of an experimental study on the behavior of plates made of the considered laminated composites are presented and compared with the previous numerical ones. The good agreement between the obtained data showed that the new proposed element is adequate for modelling the bending behaviour of fiber reinforced multilayer composite materials.

Keywords:specialized finite element, laminated, composite, fibres, validation

High mechanical performances of state-of-the-art composite materials due to their structure and mechanical properties have greatly improved the manufacturing technologies from top industries, producing important changes in engineering design.

Unlike the traditional design, in which the main dimensions of a structure are calculated taking into account the loads and mechanical characteristics of the material, the design philosophy for composite materials is based on structural concepts. Starting from the functional architecture of the structure and loading, the design calculations aim to the macrostructural organization of the material, to ensure a high degree of safety with minimum material consumption.

As it is well known, the classic theory of laminated composites is based on a series of simplifying hypotheses. Thus, the laminae are considered as very thin, made of a homogeneous, orthotropic and linear elastic material, and in a plane stress state. Also, perfect adhesion between the laminae is considered, and the Kirchhoff hypothesis is valid when the laminated composite deforms.

Starting from these hypotheses, many methodologies have been published in the scientific literature for the analytical calculus of plane plates manufactured from fiber reinforced laminated composites [1-7]. Most of the algorithms and models based on these theories lead to a global stress and strain state analysis without offering information about the contribution of the structural components, very important for characterization of the mechanical behaviour of these materials.

Due to the large scale use of composite materials for strength structures in different fields, it is necessary to introduce new design concepts and new approaches for elaboration of global calculus models, able to simulate the mechanical behaviour of the designed structures as close to reality as possible.

Efficient design methodologies with structural optimization facilities may be elaborated, by modelling composite structures with specialized finite elements that take into account the macromechanical structural organization of the material and by use of numerical methods with complex analysis codes.

In order to validate the methodologies, for further use in the design of strength structures, a verification of the numerical model is necessary. This can be done by experimental investigations of the mechanical behaviour of the studied structure, using different techniques (strain gauges, photoelasticity, interferometry). A comparison of the obtained experimental and numerical results is further made in order to establish if the proposed models are adequate and can be validated.

Experimental part

The results of a numerical determination of the stress state in a circular plate are further presented. The plate has a diameter of 184 mm, is fixed on the contour and loaded in bending with an uniformly distributed pressure p = 0.2 MPa. The material of the studied plates is a laminated composite, reinforced with E-epoxy glass fibers included in a matrix of NESTRAPOL-450 unsaturated polyester resin. Two types of such composite were considered: one having two 1 mm thick laminae (0/90) and the other with six 1 mm thick laminae (0_6). The elastic constants of the materials, given by the manufacturer, are listed in table 1.

A special finite element (COMPOZ.HAD), proposed by one of the authors [5] was used to model the composite plate in the two considered cases, in order to obtain the stress and strain fields. This element (fig. 1) is three dimensional, has eight nodes with three degrees of freedom per node and a constant thickness (equal to the thickness of a lamina). The same interpolation functions are used to describe both the displacements and the geometry (isoparametric element).

Due to its properties and its working capabilities, such an element can be used to analyze the stress and strain fields in areas with high stress gradients (around the

* email: antonhadar@yahoo.com; Tel: +40.21.4029204

Table 1
ELASTIC CONSTANTS AND MECHANICAL PROPERTIES OF THE COMPOSITE MATERIAL

Elastic constant / mechanical property	Value	Unit
Young's modulus of a lamina in the longitudinal direction - E_l	39000	MPa
Young's modulus of a lamina in the transversal direction - E_t	8600	MPa
Shear modulus of a lamina - G_{lt}	3800	MPa
Poisson's ratio in the plane of the plate - ν_{lt}	0.28	-
Poisson's ratio in the plane normal to the plate - ν_{tz}	0.42	-

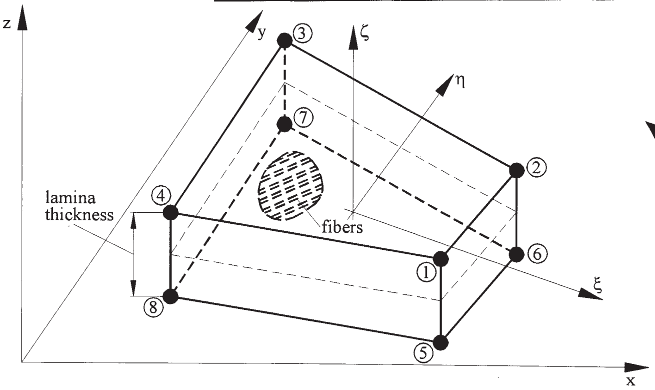

Fig. 1. New finite element

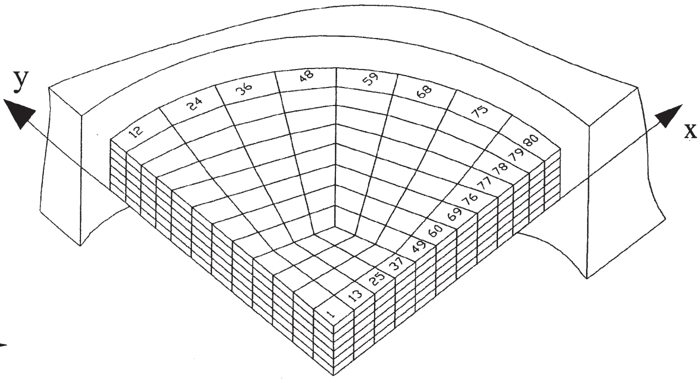

Fig. 2. Finite element mesh of the plate

stress raisers). At the same time, the use of this element in a finite element mesh allows the user to perform post critical calculations, after the appearance of local damages in the laminae. The carrying capacity of the structure can be thus evaluated for different damage modes, and a methodology for prediction of mechanical behaviour may be proposed.

An important simplification of the modelling process for composite structures is achieved by using this new finite element. Refined, simple and efficient calculus models may be obtained. For large structures, in order not to increase exaggeratedly the number of elements, the element should be used only for the local mesh in the areas with stress concentration. In the rest of the structure, other types of elements will be used.

The considered plate was meshed in a number of layers equal to the number of the laminae. A very refined mash was thus achieved, in order to have a model as close as possible to the real structure. The loading type, constraints and geometry were taken into account in the modelling process.

Owing to the symmetry, only a quarter of the plate was meshed with 80 elements and 194 modes per lamina (fig. 2).

A finite element analysis was also performed using a specialized finite element code (ANSYS-11) in order to validate the calculations undertaken with the new proposed element. In this later analysis, the laminated composite was meshed with a shell element (SHELL-93), adequate for modelling such materials.

Results and discussions

Three comparison criteria were used to analyze the data yielded from the numerical calculations in order to evaluate the performances of the proposed finite element: the vertical displacements (w), and the radial (σ_r) and circumferential (σ_t) stresses, determined along a radius.

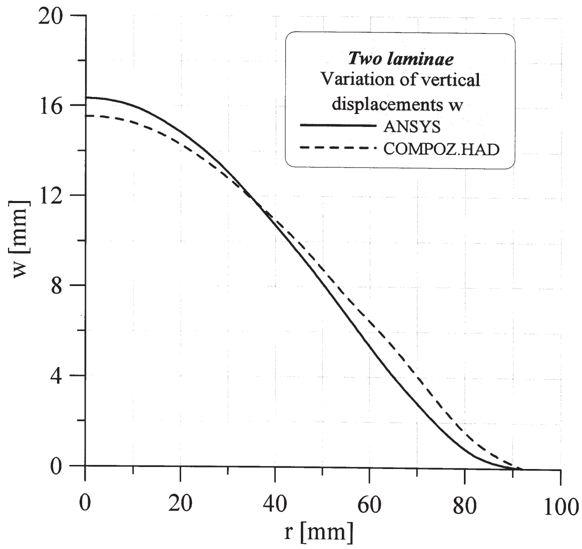

Fig. 3. Variation of vertical displacements on the radial direction for the plate with two laminae

Comparative graphs are presented for the variation of the displacements w (fig. 3) and stresses σ_r, σ_t (fig. 4) for the case of the plate with two laminae, having a total thickness of 2 mm.

Using the data obtained from the numerical simulations for the plate having six laminae (with a total thickness of 6 mm), the comparative graphs for displacements and stresses are plotted in figure 5 and figure 6 respectively.

An experimental verification of the proposed numerical models was also performed. The experimental determinations were done on two plates made of the two considered laminated composites, using a special gripping and loading device. The plates were fixed on the contour through steel and rubber gaskets, tighten

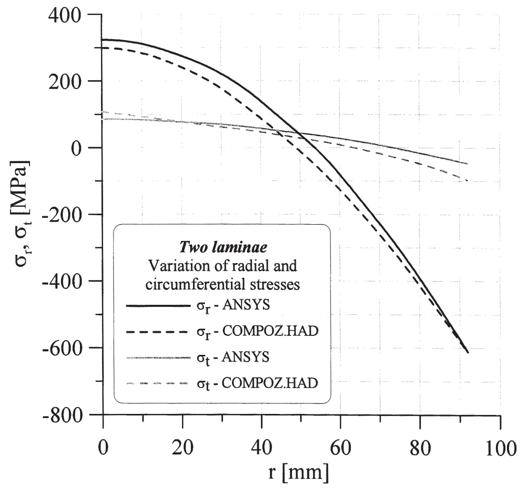

Fig. 4. Variation of radial and circumferential stresses on the radial direction for the plate with two laminae

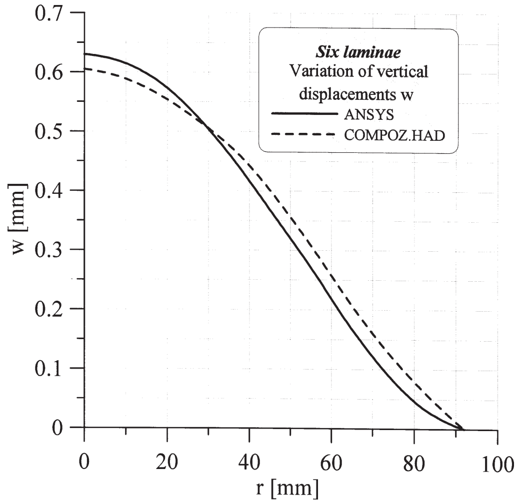

Fig. 5. Variation of vertical displacements on the radial direction for the plate with six laminae

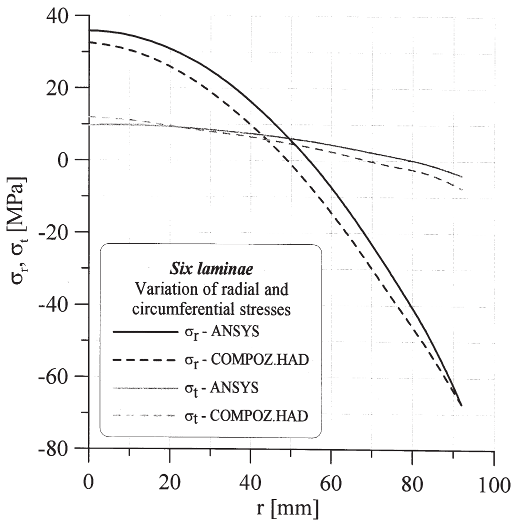

Fig. 6. Variation of radial and circumferential stresses on the radial direction for the plate with six laminae

with 12 bolts. A hydrostatic uniformly distributed load was applied using a watertight chamber in which pressurized oil at 0.2 MPa was maintained. The experimental set-up is shown in figure 7.

The vertical displacements of the plate were measured in different points and on different radii, using a dial indicator gauge. The obtained data were statistically

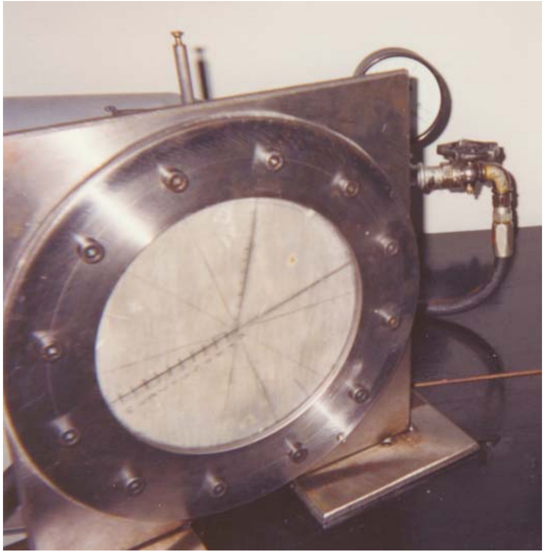

Fig. 7. The experimental set-up

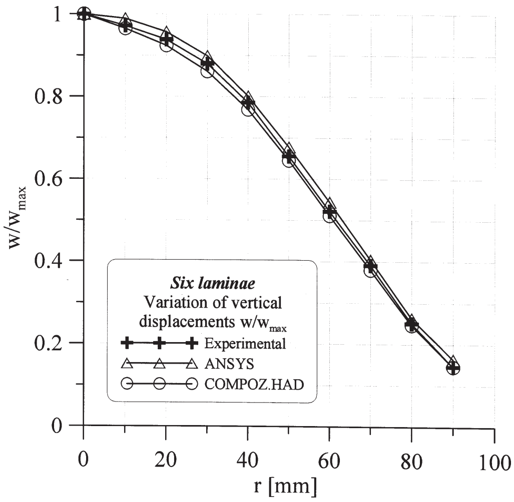

Fig. 8. Comparative variation of the nondimensional vertical displacements on the radial direction

processed and represented as variation curves of displacements along a certain radius. The normalized displacements w/w_{max} are presented in figure 8 together with the values obtained from the two numerical analyses.

Examining the graphs from figures 3-6, one can find a good agreement between the results obtained from the two finite element analyses (one using the new proposed element and the other one with the commercial code). Although the values of the stresses determined with the new element are smaller, the variation curves are similar.

The curves presented in figure 8 show also a good agreement between the numerical and experimental results for the vertical displacements of the plate.

Conclusions

The results obtained from the studies presented in this paper show that the new element proposed by the authors is adequate for modeling the laminated fiber reinforced composite structures.

The numerical simulations for the considered plates having different structures show that the element can be used for modeling composite materials with complex structure. Since the element is isoparametric, refined calculus models can be achieved, in order to study the stress field in areas with high stress concentrations. At the same time, a post critic calculus can be undertaken, for the evaluation of the carrying capacity and the mechanical behaviour of the structure.

The results of the numerical simulations may be further used for the prediction of the lifetime of the structure.

The experimental set-up used in this research can be employed for other studies, using the strain gauge technique for the determination of the stress and strain fields in the case of composite plates with different static or dynamic loadings.

References

1.BOLCU, D., STÃNESCU, M.M., CIUCA, I., JIGA, G., GHEORGHIU, H., IANCULUI, D., Mat. Plast, **45**, nr. 2, 2008, p. 137

2.CERBU, C., Mat. Plast., **44**, nr. 2, 2007, p. 98

3.GHITA, E, GILLICH, G.R., BORDEASU, I., VODA, M., TROI, C., Mat. Plast., **44**, nr. 2, 2007, p. 158

4.FETECAU, C., STAN, F., Mat. Plast, **44**, nr. 3, 2007, p. 180

5.HADAR, A., BORDEASU, I., MITELEA, I., VLASCEANU, D., Mat. Plast., **43**, nr. 1, 2006, p. 70

6.ILIESCU, N., ATANASIU, C., HADAR, A., Mat. Plast., **42**, nr. 1, 2005, p. 72

7.CURTU, I., MOTOC, D.L., Mat. Plast., **45**, nr. 4, 2008, p. 366

Manuscript received: 5.12.2008