The 40th International Conference on
Mechanics of Solids, Acoustics and Vibrations &
The 6th International Conference on
"Advanced Composite Materials Engineering"
ICMSAV2016& COMAT2016
Brasov, ROMANIA, 24-25 November 2016

NUMERICAL SIMULATION AND EXPERIMENTAL BENDING COMPOSITE SANDWICH PLATES

Florentina Rotaru (Paraschiv)[1], Ionel Chirica[2], Elena Felicia Beznea[3], Ionel Iacob[4]
[1]University "Dunarea de Jos", Galaţi, ROMANIA, Florentina.Rotaru@ugal.ro
[2] University "Dunarea de Jos", Galaţi, ROMANIA, Ionel.Chirica@ugal.ro
[3]University "Dunarea de Jos", Galaţi, ROMANIA, Elena.Beznea@ugal.ro
[4]University "Dunarea de Jos", Galaţi, ROMANIA, Iacob.Ionel@ugal.ro

Abstract: *Composite sandwich materials are a class of engineering materials that present a particularly technical and scientific interest. They are not only a perfect replacement of classical materials, but the key to solve many technical problems in various industries, many of them being difficult or sometimes even impossible to solve using traditional materials. This paper presents a study on the bending sandwich plates having clamped boundary, loaded by a uniformly distributed pressure on the skin surface area. Numerical analysis is performed using ANSYS software's 15 ACP modul (ANSYS Composite PrepPost). The results of numerical analysis (stresses and deformations arising in plate skins material) will then be used to be compared with experimental results. The materials used to manufacture plate: epoxy resin (matrix) reinforced with fiberglass for the two faces and extruded polystyrene for core. The types of meshing elements used are for faces using elements of plate type SHELL181 and for core volume elements SOLID185 type..*
Keywords: *Ansys ACP, sandwich structure, numerical analysis, FEM post-processing, bending*

1. INTRODUCTION

Sandwich plates are formed by two faces which are relatively thin but with high structural stiffness and a thicker core, which is lighter but less stiff than the faces. The faces and core are bonded using adhesives.
Nowadays sandwich panels are used in many industries such as aerospace, shipbuilding and civil industries.
Sandwich panels are simple structures with better performance than classical laminates in the conditions of bending loading. In the last decades many research works have been focused on bending behaviour under xternal loads.
In his thesis Pokaharel [1] tests were conducted to determine the main characteristics of extruded polystyrene. The results will be used in this work because they correspond mainly to polystyrene [2]. In his work Padade [3] a series of tests have been conducted on various specimens of the material EPS (Extruded Polystyrene) Geofoam to get shear strength parameters. Four different densities of EPS Geofoam 0.15, 0.20, 0.22 and 0.30 kg/m^3 were used to prepare specimens for triaxial testing. Shear strength parameters were calculated for three different pressures: 50, 100 and 150kPa. The tests were performed up to 15% of axial strain. Arbaoui et al. [4] made an experimental investigation, analysis analytical and numerical modeling of four-point bending test on a panel sandwich honeycomb polypropylene multi-layer, then the results are compared with each other. Venugopal et al. [5] have conducted a study which addressed a finite element modeling software ANSYS to predict the response of composite sandwich panels under static bending. For the study Nomax flexible core is used as core material (thickness 15 mm) and carbon fiber reinforced polymer composite is used for girls. FEM results were compared with the experimental results. Bending behavior sandwich panels for aluminum honeycomb core and glass fiber composite for skins, they were analyzed in the paper [6] by Jan S. et al, and the results were compared with analytical ones.
This paper a comparison between experimental results and those obtained with the tests using the finite element software ANSYS Composite PrepPost (ACP) is performed. Two sandwich plates made out of epoxy resin reinforced with glass fiber for skins and the core made out of extruded polystyrene are studied.

2. EXPERIMENT

For the experiment two sandwich plates with identical cores made out of extruded polystyrene. For their skins two types of fabric have been used: Twill fabric with thickness of 0.2 [mm] (Figure 1) and fabric Plain with thickness of 0.33 [mm] (Figure 2). Epoxy matrix was used for skins. In the first case where the thickness of the fabric is 0.2mm to create the faces plates were used and five layers in the second case when the thickness was 0.33 were used three layers. This reasoning was used to create skins 1mm follows: 0,2x5=1[mm] and 0,33x3=0.99[mm].

Figure 1: Twill fabric with thickness of 0.2mm

Figure 2: Plain fabric with thickness of 0.33mm

After the skins fabrication, they have been bonded by the core made out of poliuretanic foam (extruded polystyrene). (Figure 3). The plates have square shape with the side of L=380[mm]. The core thickness is of 20 [mm]. The total thickness of the sandwich plate is about 22[mm] ±0.5[mm].

Figure 3: Plate specimens

2.1. Working procedure for static loading

Sequentially, the plate, simply supported on a frame (5 in the Figure 4), was loaded by the weights (Figure 4). The weights, that have been placed, sequentially, in the center of the upper face of the plate, have the values: 5.355kg, 2.4kg, 0.830kg, 0.665kg and 0.005kg.

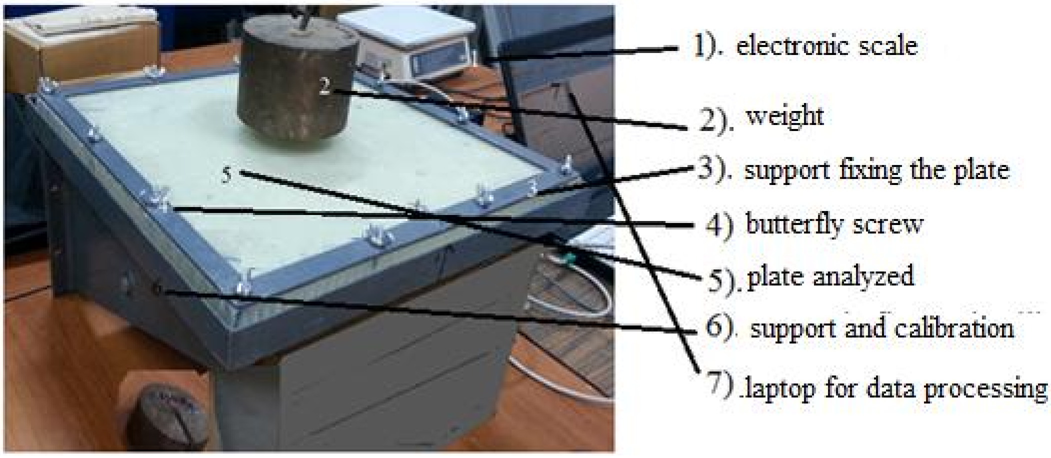

Figure 4: The sandwich plate bending rig

The bending rig has seven pieces:
1) electronic scale,
2) weight,
3) Support fixing the plate,
4) butterfly screw,
5) plate analized,
6) support and calibration
7) laptop for data processing.
The displacements are measured with the transducer LVDT, placed back to the plate according to Figure 5. The signals are acquired by the Quantum strain gauge system (Figure 6). The results of the static experimental tests are illustrated in tables 3 and 4.

Figure 5: Displacement transducer LVDT

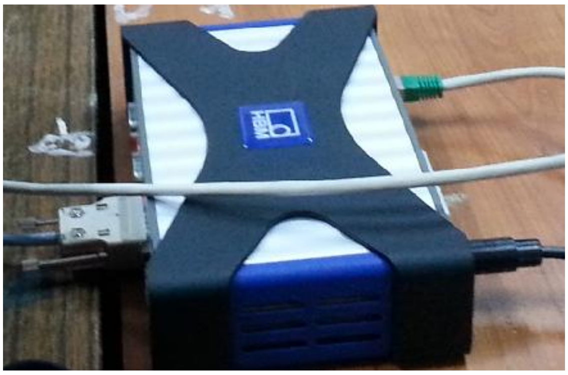

Figure 6: System Quantum

Figure 7: Weights for static tests

3. FINITE ELEMENT MODELING

3.1Material properties

Material properties for polystyrene are adopted after Pokaharel [1] and shown in Table 1.
The modulus of elasticity was rounded to 3,8 [MPa] to 4[MPa] as adhesive for bonding layers are added. Extruded polystyrene is considered isotropic material. The main characteristics for Epoxy_Glass_UD are shown in Table 2. The material is considered orthotropic. Source characteristics: From ANSYS library.

Table 1: Material properties for extruded polystyrene

Property	Value	Unit
Density	30	Kg/m^3
Young Modulus	4	MPa
Poisson’s Ratio	0,08	
Bulk Modulus	1,5873E+06	Pa
Shear Modulus	1.8519E+06	Pa

Table 2: Material properties for Epoxy_Glass_UD

Property	Value	Unit
Density	2000	Kg/m^3
Young Modulus x direction	50000	MPa
Young Modulus y direction	8000	MPa
Young Modulus z direction	8000	MPa
Poisson's Ratio xy	0,3	
Poisson's Ratio yz	0,4	
Poisson's Ratio xz	0,4	
Shear Modulus xy	5000	MPa
Shear Modulus yz	3846,2	MPa
Shear Modulus xz	5000	MPa

3.2Problem description

Modeling of sandwich plates was conducted using ANSYS package ACP using finite element method. SHELL181 is the element type used for the skins. SOLID185 is the element type used for sandwich plate core. Forces are placed on the skin, on center plate as in illustrated in Figure 8. As an element of support was used "fixed support". In general it tried to meet all the conditions to compare experimental results. In Ansys Composite PrePost there are some special commands that allow arranging the thickness of layers, but on different angles and fiber orientation.

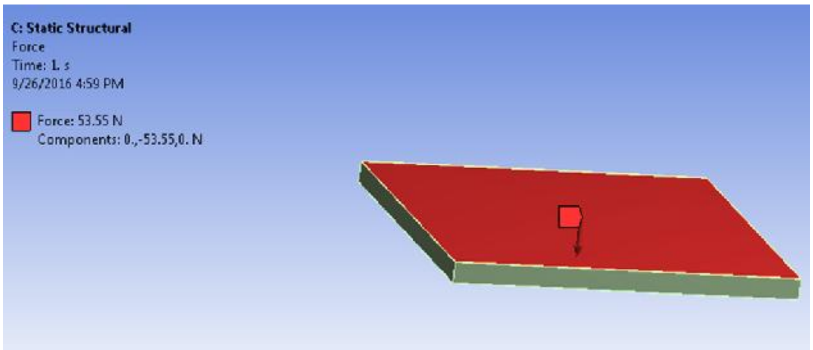

Figure 8: Sandwich plate model

The commands used in the numerical modeling are:
- materials: this command allows the user to choose the material that wants to work.
- fabrics: allows the user to select material and to give a thickness
- stackups: allows the user to arrange an angle fibers ex after [0/90], [45 / -45] etc.
- element sets: is a command that enables the user to create items such face1 face 2 (was chosen for this work);
- rosettes: this command creates the origin and directions that the user wants to use them. Oriented selection sets: with this command is oriented material. -modeling groups: with this command creates groups of layers.
- sampling points: this command enables the user to select one item from the ex. a girl sis to work on it so that to see certain characteristics.
- solid models: this command creates solid as in Figure 10. The mean thickness of the element can be seen.[7], [8].

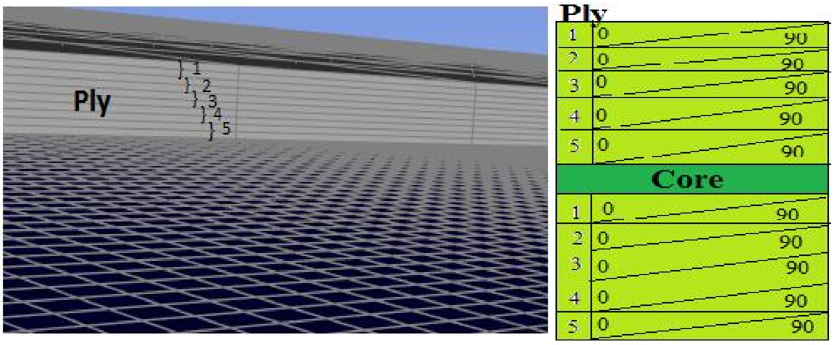

Figure 10: The solid modeling in ACP. Skins layers modeling

4. RESULTS AND DISCUSSION

The results are obtained both in experimental and numerical modeling using Ansys in ACP by static bending. The elastic deformations will not occur remanent deformations. The two specimens were tested in bending loading by forces: 8.3 [N], 14.95 [N], 38.95 [N] and 53.55 [N]. In the first table the specimen is noted SF20 / 0,2X5. SF means foam sandwich, 20 means core thickness. 0.2x5 refers to the thickness of the skins and number of layers. The bending results (displacements) can be seen in Tables 3 and 4. The related diagrams are illustrated in Figures 11 and 12.

Table 3: Experimental and numerical results for the plate having the skins with three layers

Specimen	SF20/0,2x5		
Load [N]	Displacement [mm] Experiment	Displacement [mm] ANSYS (ACP) FEM	Error
8,30	0,01379	0,01480	0.00101
14,95	0,02627	0,02666	0.00391
38,95	0,07745	0,06946	-0.00799
53,55	0,09766	0.09549	-0.00217

Table 4: Experimental and numerical results for the plate having the skins with five layers

Specimen	SF20/0,33x3		
Load [N]	Displacement [mm] Experiment	Displacement [mm] ANSYS (ACP) FEM	Error
8,30	0,01320	0,01448	0.00128
14,95	0,02570	0,02608	0.00038
38,95	0,06702	0,06796	0.00094
53,55	0,09266	0,09343	0.00077

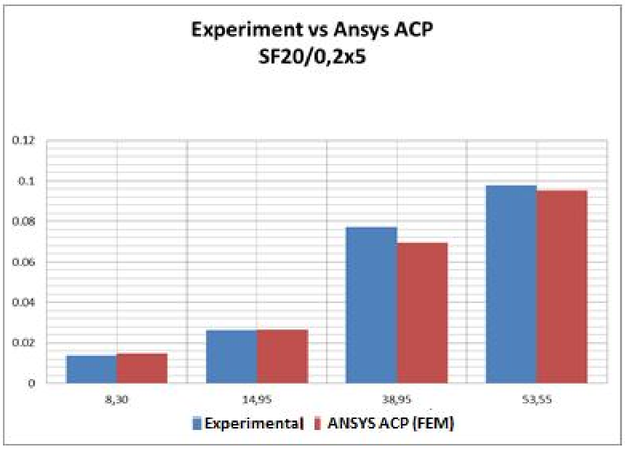

Figure 11: Experiment versus Ansys ACP for specimen SF20/0,2x5

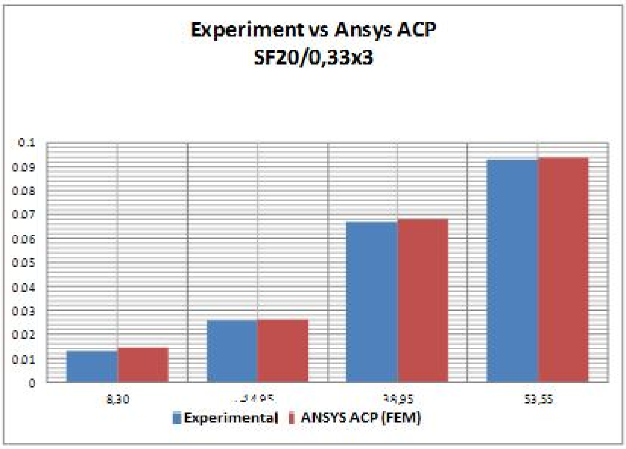

Figure 12: Experiment versus Ansys ACP for specimen SF20/0,33x3

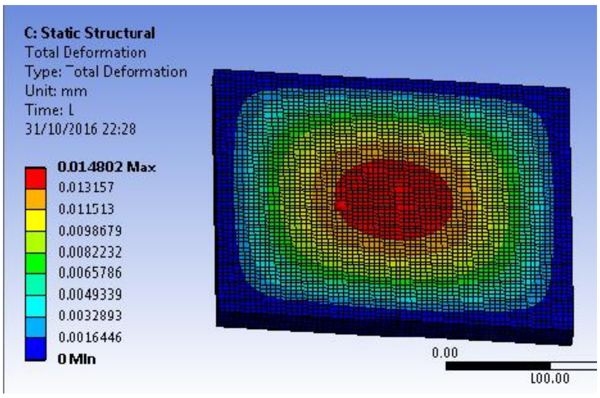

Figure 13: The displacement map for the case of force of 8,3[N]

In Figure 13 the displacement map for the sandwich plate SF20/0,2x5, loaded by a force of 8.3 [N] is illustrated. In Figure 14 the comparative diagram for both models (experimental and numerical) is illustrated.

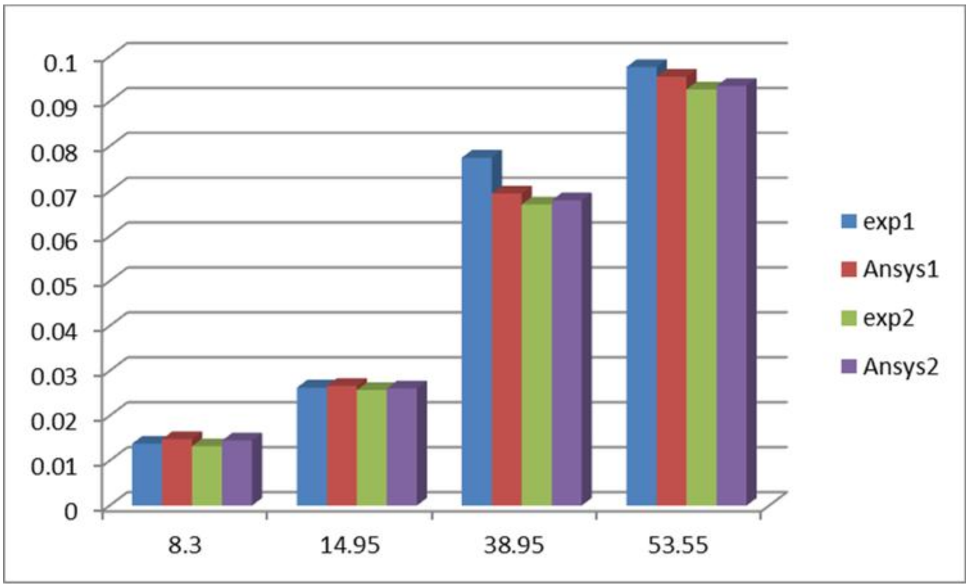

Figure 14: The maximum displacements for the tested specimens

5. CONCLUSION

The bending test results were conducted using both experimental and modeling in finite element package Ansys ACP. Finally the results obtained in both modeling have been compared. It was found that after calculus errors, the differences are very small. For specimen SF20 / 0,33x3 were obtained displacements smaller than the specimen SF20 / 0,2x5. This shows that the specimen plate with skins made out of 3 layers is more stiffened than the specimen having the skins with 5 layers. Both plates have the same thickness.

ACKNOWLEDGEMENTS

The work has been funded by the Sectoral Operational Programme Human Resources Development 2007-2013 of the Ministry of European Funds through the Financial Agreement POSDRU/159/1.5/S/132397.

REFERENCES

[1] Pokaharel N., Thesis: Behaviour and design of sandwich panels subject to local buckling and flexural wrinkling effects, Queensland, November 3003.

[2] http://www.adeplast.ro/wp-content/uploads/2014/12/Fisa-tehnica-polistiren-extrudat-XPan-Zentyss.pdf;

[3] Padade A. H., Behavior of Expanded Polystyrene (EPS) Geofoam Under Triaxial Loading Conditions, Mumbai, India, Vol. 17 [2012],

[4] Arbaoui J., Schmitt Y., Pierrot J.L., Royer F.X., Numerical simulation and experimental bending behavior of multi-layer sandwich structures, Journal of theoretical and applied mechanics pp. 431-442, Warsaw 2014;

[5] Venugopal M.M., Maharana S. K., Badarinarayan K. S., Finite element evaluation of composite sandwich panel under static four point bending load;

[6] Jan S., Ullah khan R., Ahmad S., Amjad M., Flexural strength of honey comb sandwich structures, University Islamabad, Pakistan, Vol. 4, Issue 1, 2015;

[7] https://www.youtube.com/watch?v=MxyVEEOlEJc;

[8] https://www.youtube.com/watch?v=i0OI85-KABQ;