THE FINITE ELEMENT ANALYSIS OF A POLYMER BASED TRIANGULAR CELL SANDWICH COMPOSITE

ION DURBACĂ[1], ADRIAN-COSTIN DURBACĂ[2]

[1] *„Politehnica" University of Bucharest,* ion.durbaca@yahoo.com

[2] *Richard Alan Engineering LTD, Dewsbury* (UK)*,* costin.durbaca@yahoo.com

Finite Element Analysis (FEA) method is used for analysing the mechanical behaviour of standardized tensile and bending (3-point) test specimens made from a multi-layered composite material with two outer polymer facesheets (synthetic glass / plexiglass) and a triangular cellular core. This analysis uses for design and modelling, the software Autodesk Inventor Professional, which will import later the model in the FEA analysis software - ANSYS 14.5. Highlighting the mechanical behaviour of the composite structure specific to the test specimens having cellular polymeric core (ABS, PLA, PC and CF triangular cells, with thicknesses 3 and 5 mm) and analysing the stress/strain state and specific deformations and correlating the FEA simulation results with the experimental tests lead to quasi-equivalent results under similar stress conditions until the specimen fails. Therefore, the FEA analysis of the mechanical assemblies is taking an important step forward in the modern design process, being one of the ways of identifying the von Mises equations and deformation fields in the composite above structures.

Keywords: test specimen, composite, sandwich, finite element analysis, cellular core, simulation.

INTRODUCTION

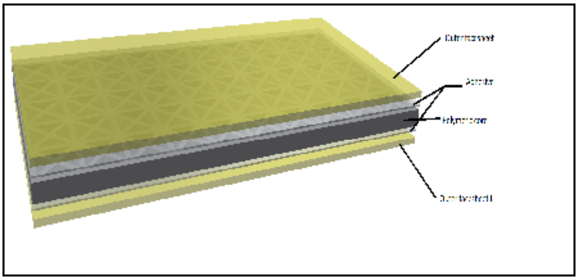
Because the highly complex analytical methods, it is almost impossible to apply them for a wide variety of mechanical structures. Therefore, many calculation of composite structures problems were solved with the use of numerical methods. It eliminates the need to write and solve complex equations with partial derivatives that characterize composite materials. Numerical calculation methods have the advantage of being applicable to more general classes of problems. Among the numerical methods, the finite element method (FEA) takes an important place in the analysis of structures made of composite materials, in general, or multi-layer materials, in particular (Durbacă, 2018).

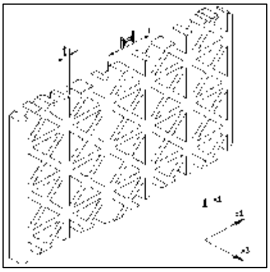
The current development of sandwich composite structures makes it possible to use them with very good results in many industrial fields (aeronautics, aerospace, naval, motor, railway, industrial process equipment etc.), thus representing a scientific research direction in full progress. Cellular materials made from polymeric materials are now readily available, although prices are higher than standard products and they continue to have a massive market introduction as a result of the manufacturing process. Such materials are used in a variety of applications: sandwich panel cores, starting from simple and inexpensive parts to complex and advanced aerospace components (Syngellakis, 2016).

In literature, the cellular structure is referred to as the cellular core due to its positioning in a sandwich assembly comprising of two outer skins on the other side of the cellular core together with the joint attachment between the core and the outer shells (i.e. attaching with thin and ultra-adherent film coatings). This joining addition causes the cellular core and outer shells to behave as a continuous structure, thus transferring axial and transverse loads to and from the cellular core that provide sufficient rigidity to maintain equidistance between the outer skins. From a structural point of view (see Figure 1), the main function of the cell core is to stabilize the outer shells to avoid

https://doi.org/10.24264/icams-2018.VII.1

buckling and deformation and to withstand the shear stresses along its thickness. The outer shells bear alongside some of the local stresses and tension and compression stresses, and their primary function is to provide the bend and shear stiffness of the sandwich assembly.





(a) (b)

Figure 1. The main components of the sandwich composite structures (a) and the geometry of the cellular core (b)

The intense preoccupations in the international academic environment, demonstrated by the large number of scientific papers published in the field and supported by the results obtained in various industrial applications, reflect the practical importance of the finite element analysis of any composite layered structures (Itu, 2014).

Since many of the drawbacks found in the laboratory tests stem from their dependence on the prototype configuration, and the construction of an experimental prototype for each composite structure variant would sometimes be difficult to achieve in terms of execution, it is fully justified to apply the FEA as an innovative and of great interest for the future, both economically and in terms of simulation time.

Therefore, the use of modelling and simulation through the finite element analysis of the mechanical system components occupies an important place in the modern design process and is one of the ways to identify deformation fields and equivalent stresses within the analysed composite structures.

DESCRIPTION OF THE ANALYSIS MODEL WITH FINITE ELEMENT

The Finite Element Analysis Model with Triangular Cellular Polymer Layered Composite Specimens have been modelled and designed using 3D Autodesk Inventor, and imported in the FEA Analysis software, ANSYS 14.5 – Workbench (Durbacă and Durbacă, 2011).

The aim of the finite element analysis simulation is to highlight the mechanical behaviour of the composite structures specimens with cellular polymer core from ABS (acrylonitrile-butadiene-styrene), PC (Polycarbonate), PLA (polylactic acid) or CF (carbon fibre) having a thickness of 3 and 5 mm, which will study the stress and deformation conditions specific to the tensile and bending stresses (at 3 points) of the above structures.

If the results of the FEA analysis study reveal a good consistency with experimentally obtained results, it can be discarded in subsequent studies to use experimental tests, leading implicitly to direct economic effects.

https://doi.org/10.24264/icams-2018.VII.1

Geometric Definition of the Model

Using the design drawings of standardized test specimens for tensile stress tests (EN ISO 527-1: 2012) and bending (EN ISO 14125: 2000), the model was designed and modelled using the Autodesk Inventor Professional software (see Figure 2).

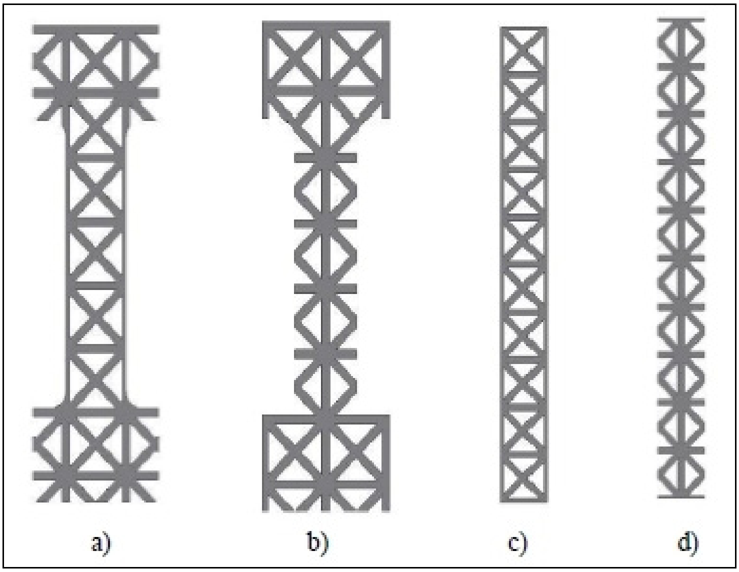


Figure 2. Representation of the 3D model made in Autodesk Inventor (Durbacă, 2018): a) closed cell tensile test specimen (type 1); b) open cell tensile test specimen (type 2); c) open cell bending test specimen (type 1); d) closed cell bending test specimen (type 2)

Meshing

By exporting the 3D models made in the Autodesk Inventor to the ANSYS software, the models were pre-processed: meshing and defining the boundary conditions specific to each type of specimen (ABS 3.1 / ABS 3.2, ABS 5.1 / ABS 5.2; PC 3.1 / CF 3.2; CF 5.1 / CF 5.2; PLA 3.1 / PLA 3.2; PLA 5.1 / PLA 5.1; CF 3.1 / CF 3.2; CF 5.1 / CF 5.2) Figures 3.

Each model was analysed separately using a static analysis, considering the nonlinear plastic characteristics of each material. It has been considered that the adhesive achieves perfect bonding of the layers (contact method between faces in ANSYS - "Bonded"). For meshing, I was considered an average size of the finite element of 1 mm and a Hex dominant method.

https://doi.org/10.24264/icams-2018.VII.1

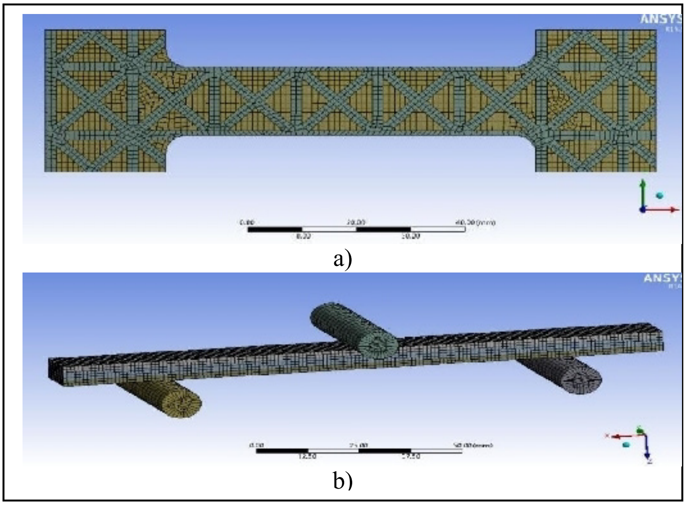


Figure 3. Pre-processing (meshing) of PLA3.1 specimen analysis on traction (a) and bending (b) (Durbacă, 2018)

Defining Structural Characteristics

The Finite Element Analysis requires knowledge of physical properties and mechanical behaviours for the calculation of stiffness matrices of finite elements. These can be grouped into geometric properties and material properties as shown in Table 1.

Table 1. Properties of polymer materials used in FEA analysis (Durbacă, 2018)

Property	PLA	PLA+40% carbon fibre	ABS	PC	PMMA
Density ρ [g/cm^3]	1.25	1.29	1.1	1.2	1.18
Elastic modulus E [GPa]	2.5	2.2	2.2	1.96	2.7
Elongation [%]	6	2	2.6	4	3
Yield strength [MPa]	40	48	42	50	50
Tensile strength [MPa]	45	52	49	55	60
Shear modulus g [GPa]	0.92	0.81	0.81	0.72	0.99
Poisson's ratio υ	0.36	0.36	0.35	0.37	0.37

In the case of the MEF analysis of the 16 types of specimens, 5 types of materials were considered. Thus, the superior faces are made of synthetic glass (PMMA or plexiglass), and the cellular core of ABS, PLA, CF and PC, having thicknesses of 3 mm and 5 mm.

RESULTS AND DISCUSSIONS

The post-processing of results aims to analyse the yield stresses and maximum deformation forces for each type of specimen and corresponding to each type of test. These are shown in Figures 4 ÷ 6, for the PLA cellular polymer specimens, and the maximum stresses were analysed based on the stress gradient.

https://doi.org/10.24264/icams-2018.VII.1

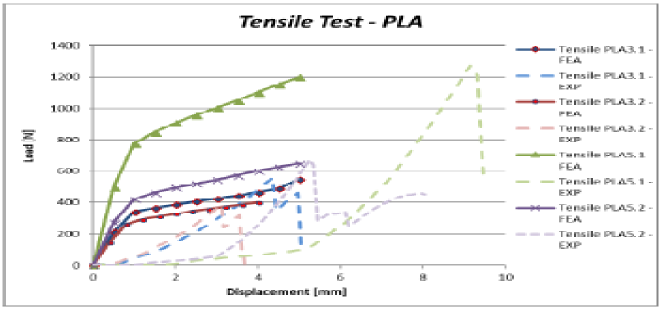


Figure 4. Deformation (elongation) force diagram for tensile stress simulations of composite material specimens having a PLA cellular polymer core of 3 mm thickness (PLA3.1, PLA3.2) and 5 mm (PLA5. 1; PLA5.2) (Durbacă, 2018)

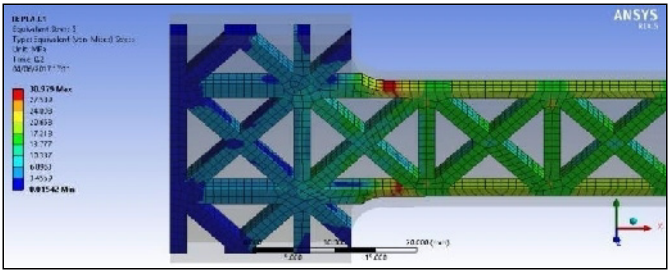


Figure 5. Maximum equivalent stresses in the PLA3.1 composite core [1]

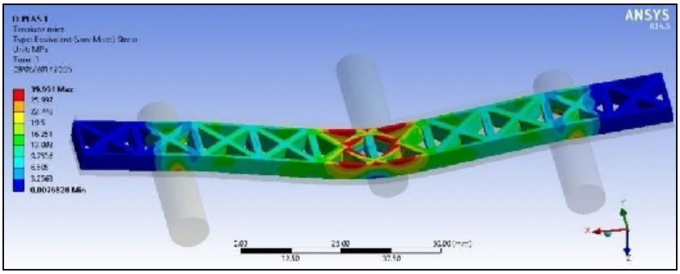


Figure 6. Maximum equivalent stresses in the composite core PLA5.1 (Durbacă, 2018)

For the other specimens having the ABS, CF and PC cellular core, the simulation results are analysed and presented in Durbacă, 2018.

CONCLUSIONS

The Finite Element method allows for a postcritical calculation of the structure in which damage has occurred, both in order to determine the load-bearing capacity of the structure and to observe how the material behaves under a certain damage (propagation of defects). The FEA analysis of mechanical system occupies an important place in the

https://doi.org/10.24264/icams-2018.VII.1

modern design process, being one of the ways to identify deformation and stress fields in the structures in question.

Analysing the applied force - maximum strain diagrams for each type of simulated specimen in both cases, traction and 3 points bending, the following were observed:

- the differences between applied force values relative to the maximum deformations for each core configuration (closed cells "1" and open cells "2") are visible, resulting in higher resistances of closed cell specimens;
- due to the finite element analysis in the plastic field, the maximum forces are the cause of a single failure mode, namely the most vulnerable to the failure component, namely the upper / lower faces, which is not always the case for laminated composites in which yield mode is complex, and the yield curve can also be described by the later yield of the core;
- the differences in applied force values relative to the maximum deformations between closed cell specimens "1" and "2" open cells are 10-50%.

Using the finite element method, you can study several modifications, namely: changing load conditions, boundary conditions, and how to apply them to the virtual model; In this way you can choose the optimal version, the dimensions and the elastic characteristics of the materials.

Thanks to the FEA, there are advantages to comparing and verifying the strength of a material, measuring the stresses, deformations that may exist in the material, and whether they are within normal limits, so as to save a lot of time, much faster, reduce costs, to each part of the structure.

REFERENCES

Durbacă, A.C. (2018), "Cercetări teoretice și experimentale privind evaluarea caracteristicilor fizico – mecanice ale plăcilor polimerice de tip sandwich cu miez compus din celule triunghiulare", Teză de Doctorat, Universitatea POLITEHNICA din București.

Durbacă, A.C. *et al*. (2017), "Experimental research on the triangular lattce type polymer based composite structure for sandwich panel construction București", *Revista de Materiale Plastice*, 54(4), 639 – 644.

Durbacă, I. and Durbacă, A.C. (2011), "Injection moulding simulation of polymer structures modeled with finite element", *The Scientific Bulletin of Valahia University Materials and Mechanics*, no. 6 (year 9), p. 156 - 159.

Itu, C. (2014), "Contribuții privind îmbunătățirea plăcilor circulare realizate din materiale compozite stratificate", Teză de Doctorat, Universitatea "Transilvania" din Brașov.

Sorohan, S. (2015), *Practica modelării cu elemente finite - Note de curs*, Universitatea Politehnica din București.

Syngellakis, S. (2016), *Composites: Advances in Manufacture and Characterisation*, Wessex, UK: Wessex Institute of Technology, UK, ISBN 978-1-78466-167-0.

https://doi.org/10.24264/icams-2018.VII.1