A comparative study evaluating the hybrid composite-metal aviation structure

Daniela BARAN*,[2], Romeo MARIN[1], Mihaela PETRE[2], Radu BOGATEANU[1]

*Corresponding author
[1]INCAS – National Institute for Aerospace Research "Elie Carafoli",
B-dul Iuliu Maniu 220, Bucharest 061126, Romania,
marin.romeo@incas.ro, bogateanu.radu@incas.ro
[2]AEROSPACE Consulting,
B-dul Iuliu Maniu 220, Bucharest 061126, Romania,
baran.daniela@incas.ro*, petre.mihaela@incas.ro

DOI: 10.13111/2066-8201.2021.13.2.1

Received: 26 February 2021/ Accepted: 31 March 2021/ Published: June 2021

Abstract: *The main purpose of this paper is to develop some numerical experiences based on mechanical tests performed on a hybrid composite metal aeronautical structure using finite element commercial codes (here NASTRAN). The results of the numerical simulations are consistent with the laboratory tests and encourage us to continue to improve the models using NASTRAN capabilities to obtain a realistic simulation of aeronautical structures made of such composites, taking into account their special properties.*

Key Works: *composite structures, mechanical tests, finite element simulations*

1. INTRODUCTION

In this paper we consider numerical finite element simulations on some hybrid composite-metal aeronautical structure comparing them with a similar metal structure. The results of the numerical simulations are compared with laboratory tests in order to understand how these hybrid structures can successfully replace the classical metal structures.

The numerical simulations are performed using the finite element analysis with PATRAN/NASTRAN and compare the results of these simulations, both composite and classical, with laboratory tests. In this paper we consider only experimental values of material constantans.

2. FINITE ELEMENT SIMULATIONS

2.1 CAD Models of an aeronautical structure

To compare the hybrid structure with the classical structure we consider the normal modes using the finite element analysis, and compare them again with the experimental values. Figure 1 describes a geometrical representation of a hybrid panel.

In this analysis we use data expressed in N, mm, MPa. Panel dimensions are L=820mm, l=620mm and with a mean radius of 286mm made by 7 plyes arranged 0-90-0, with aluminium

2024 stiffening with: E=73100Mpa, ν=0.33, ρ=2.78Kg/m^3. The composite plate is modeled using PATRAN/NASTRAN as in figure 2 and the evaluation is performed as usual in the aircraft design [1, 2, 3, 4].

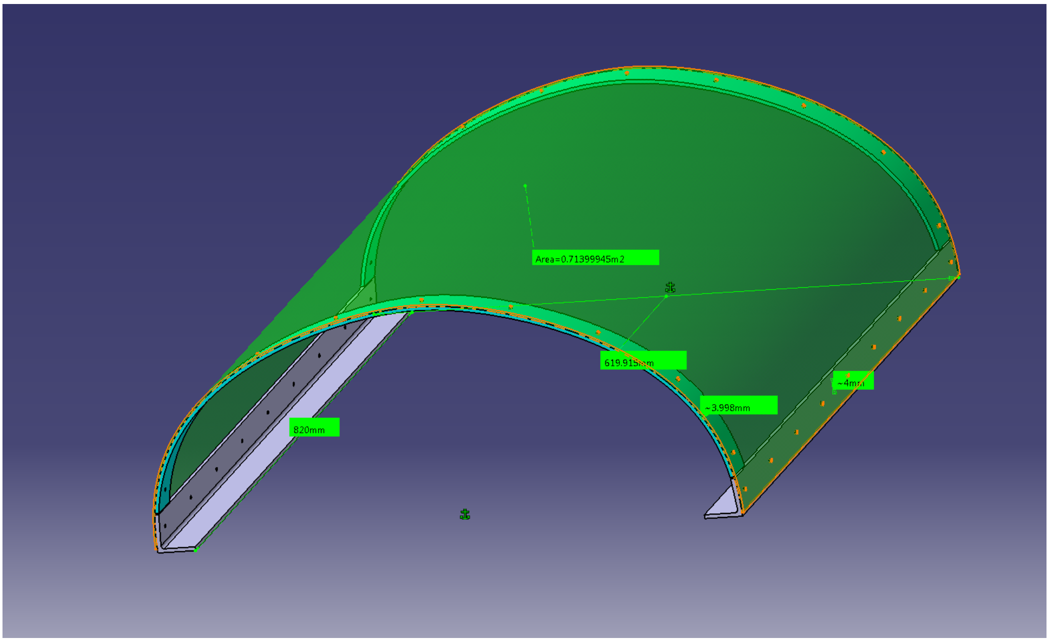
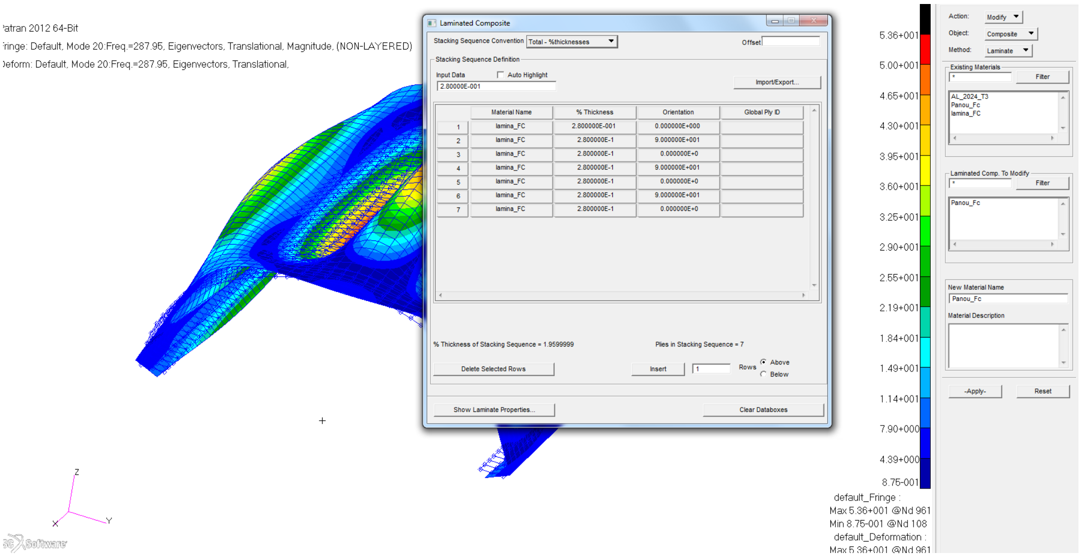


Figure 1. Geometrical representation of a hybrid panel


atran 2012 64-Bit
ringe: Default, Mode 20:Freq.=287.95, Eigenvectors, Translational, Magnitude, (NON-LAYERED)
eform: Default, Mode 20:Freq.=287.95, Eigenvectors, Translational,

Laminated Composite
Stacking Sequence Convention: Total - %thicknesses
Offset:
Stacking Sequence Definition
Input Data: 2.80000E-001
☐ Auto Highlight

	Material Name	% Thickness	Orientation	Global Ply ID
1	lamina_FC	2.800000E-001	0.000000E+000	
2	lamina_FC	2.800000E-1	9.000000E+001	
3	lamina_FC	2.800000E-1	0.000000E+0	
4	lamina_FC	2.800000E-1	9.000000E+001	
5	lamina_FC	2.800000E-1	0.000000E+0	
6	lamina_FC	2.800000E-1	9.000000E+001	
7	lamina_FC	2.800000E-1	0.000000E+0	

% Thickness of Stacking Sequence = 1.9599999
Plies in Stacking Sequence = 7
Insert 1 Rows ● Above ○ Below

Action: Modify
Object: Composite
Method: Laminate
Existing Materials: AL_2024_T3, Panou_Fc, lamina_FC
Laminated Comp. To Modify: Panou_Fc
New Material Name: Panou_Fc
Material Description:

default_Fringe :
Max 5.36+001 @Nd 961
Min 8.75-001 @Nd 108
default_Deformation :
Max 5.36+001 @Nd 961

Figure 2. The FEM model of the layers of the hybrid panel

3. MECHANICAL LABORATORY TESTS

The equipment used to test the hybrid structure are presented in figures, 3 and 4. The specimen was excited with frequencies ranging from 6 to 160Hz with a 0.1 step and a stabilizer time of 3 seconds. Some results of the experience are presented in figure 5.

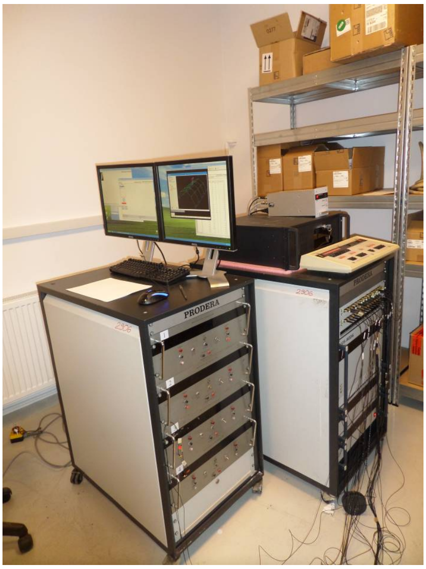


Figure 3. The experimental equipment PRODERA

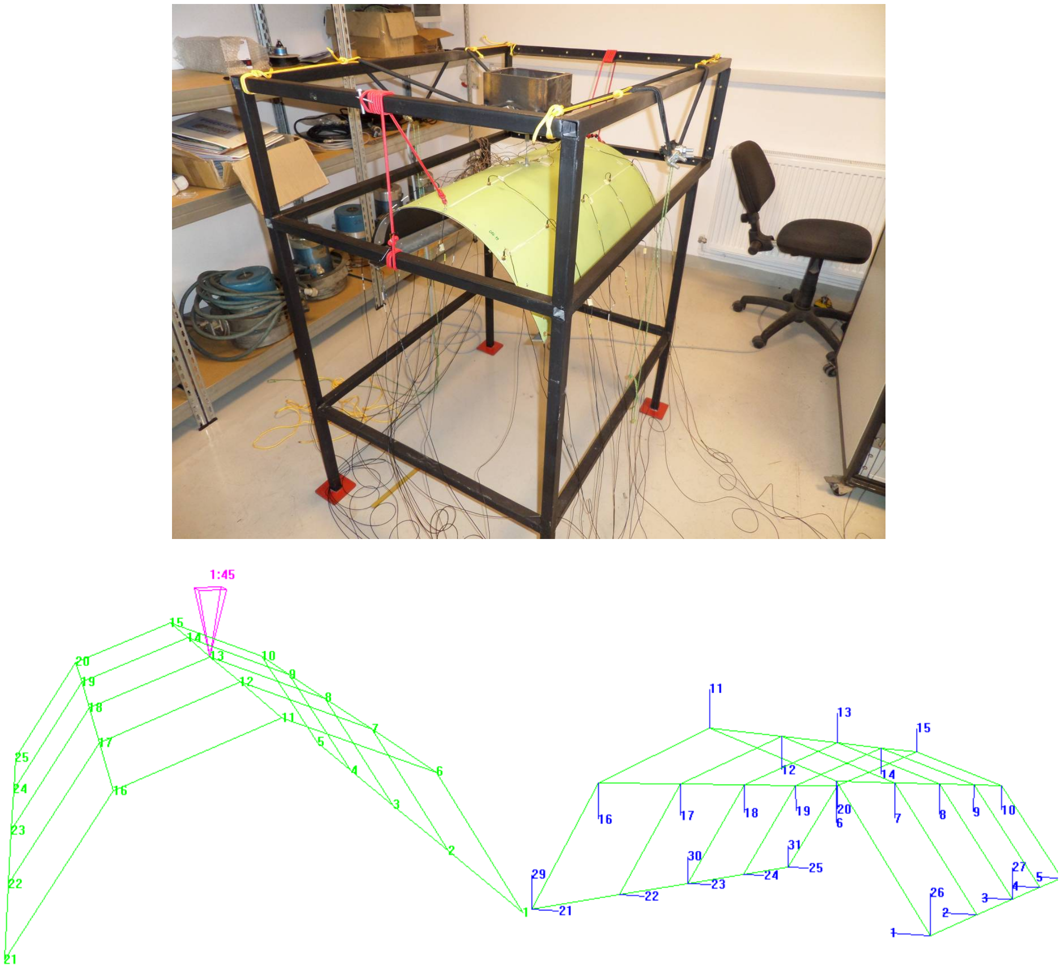
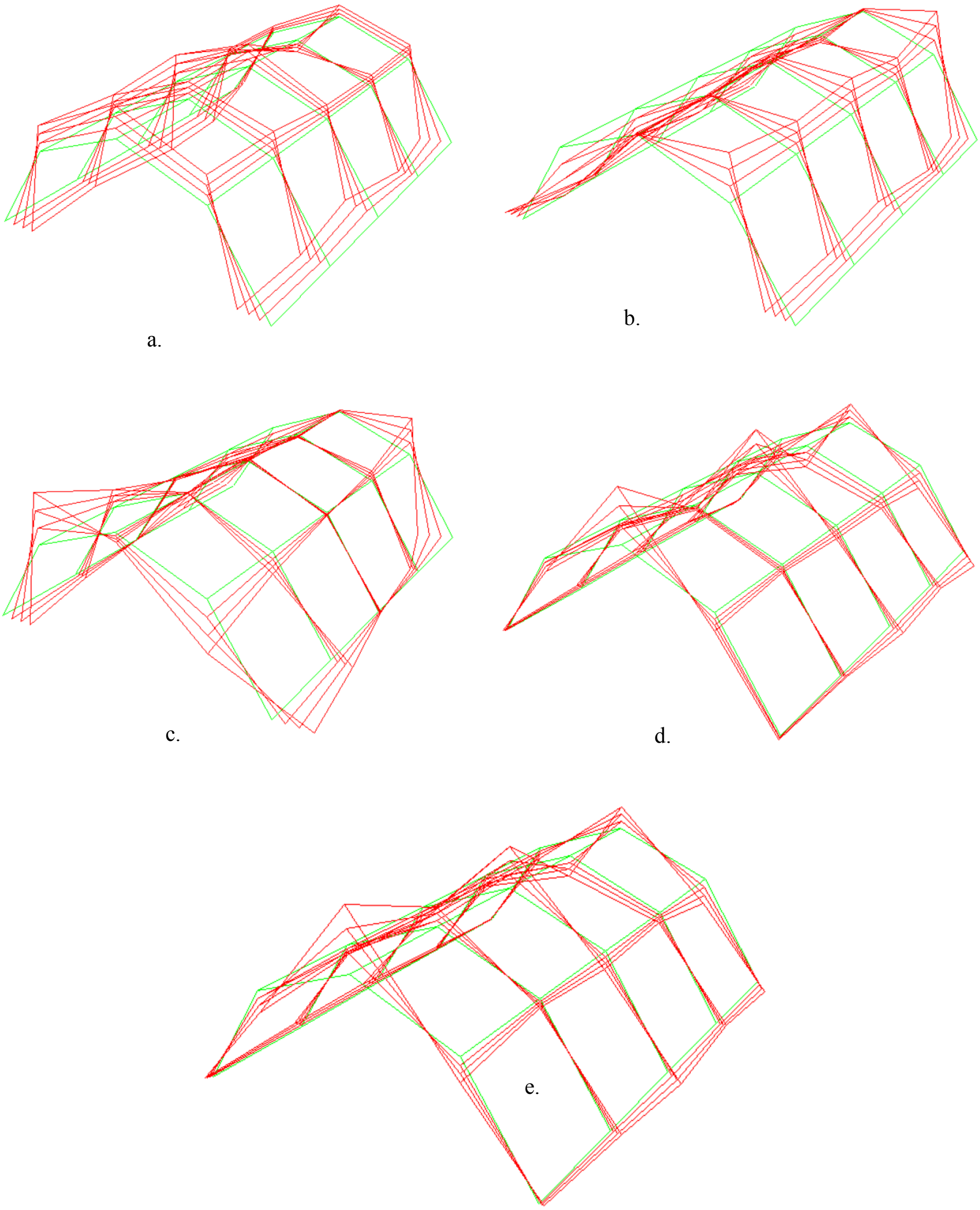


Figure 4. The panel with the setup of the excitation forces



a.

b.

c.

d.

e.

Figure 5. Experimental normal modes

4. COMPARATIVE ANALYSIS OF THE NUMERCAL SIMULATIONS AND THE LABORATORY TESTS

The panel is modeled with 2D shell elements (figure 6). For this simulation we did not consider the assembly elements (the connection between the panel and stiffener frame is simulated with a node on node link). The analysis is developed as mentioned before for a metal panel and a hybrid composite metal panel.

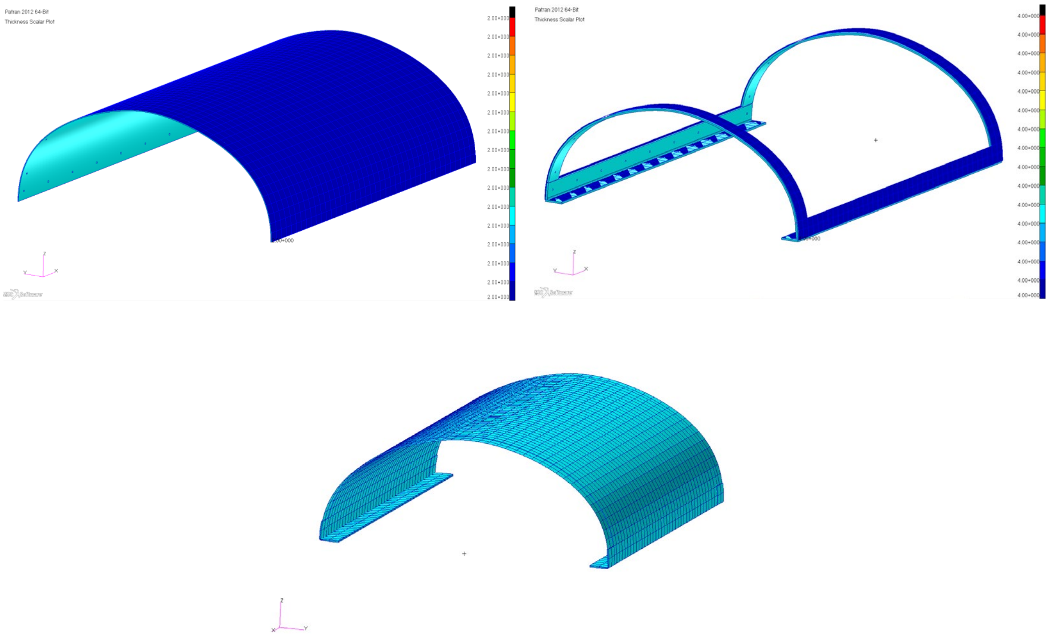


Figure 6. The finite element model panel and the metal reinforcement

The mass of the two panels is presented in figure 7.

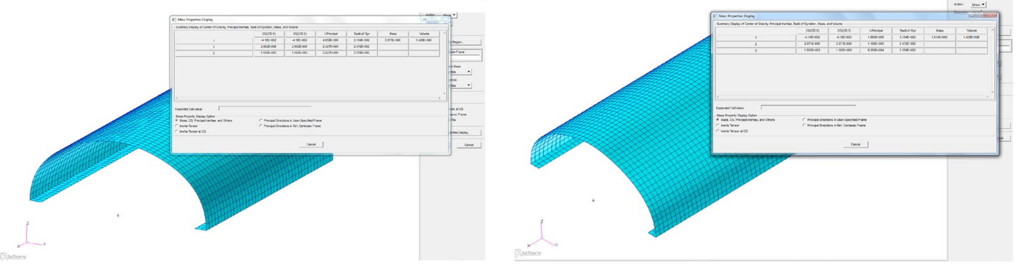


Figure 7. Mass of the two panels

Table 1. Mass comparison

Metal Panel	Modeled Composite	Experimental Composite
3.971 Kg	1.814	1.815

Comparatively, the first two elastic normal modes obtained with the finite element analysis for the metal panel are described in figures 8, 9, and those of the hybrid panel are described in figures 10, 11.

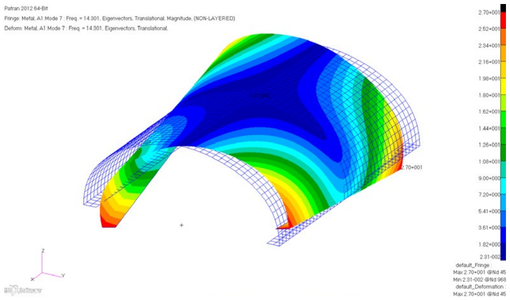


Figure 8. Elastic mode number 7 for the metalic panel 14.3Hz

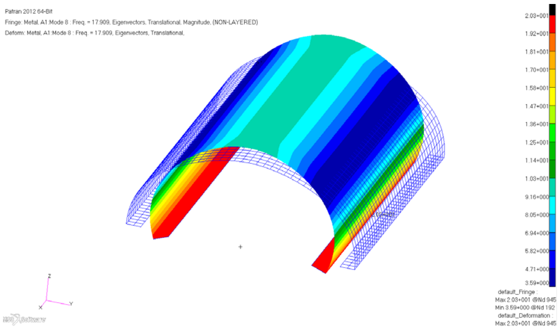


Figure 9. Bending mode number 8 for the metalic panel 17.90Hz

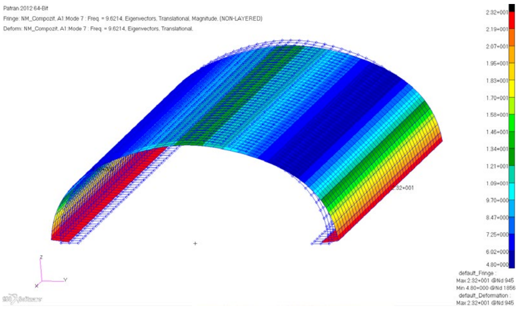


Figure 10. Elastic mode number 7 for the hybrid panel 9.62Hz

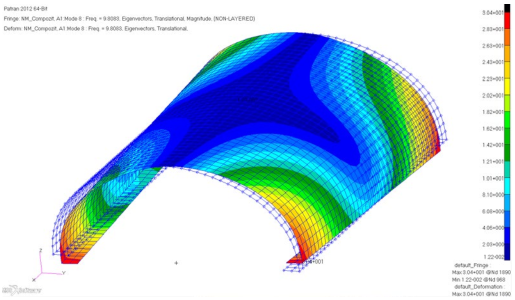


Figure 11. Bending mode number 8 for the Hybrid panel 9.80 Hz

The following table contains both the results of numerical simulations and laboratory experiments.

Table 2. Comparison between numerical results and experimantal results

	Metal panel	*Composite panel 7L*	*Experiment for Composite panel 7L*	*Remarks*
MODES	*Frequency [Hz]*	*Frequency [Hz]*	*Frequency [Hz]*	
Mode1	0. 0000	0. 0000	-	These are rigid body modes used only for FEM validation.
Mode2	0. 0000	0. 0000	-	
Mode3	0. 0000	0. 0000	-	
Mode4	0. 0000	0. 0000	-	
Mode5	0. 0000	0. 0000	-	
Mode6	0. 0000	0. 0000	-	

Mode7	14.30	9.62	7.7	
Mode8	17.90	9.80	-	The frequencies are too close to be determined by testing, two set-up excitation forces are needed
Mode9	37.33	23.73	21.5	The differences are produced by the clamps
Mode10	55.035	31.38	37.6	The differences are produced by the clamps
Mode11	90.11	54.24	47.8	The differences are produced by the clamps
Mode12	112.44	67.53	50.2	The differences are produced by the clamps
Mode13	166.25	100.45		

NOTE: Experimental results are only for bending modes.

5. CONCLUSIONS

From table Table 2 we draw the following conclusions:

- In this paper we present a comparison study between numerical simmulations and experimental tests for vibrations of a hybrid structure.
- The rigid body natural frequencies can not be determined by laboratory experiences,. The support structures influence the resonance frequencies from 0.01Hz to 10Hz.
- The six rigid body frequencies determined by the finite ekement analysis (3 translations and 3 rotations) prove the consistency of the finite element model.
- The low frequencies (below 100 Hz) are well determined by the laboratory tests.
- The high frequencies are influenced by the slenderness of the panel.
- The differences between the metallic sturcture and the hybrid composite structure are given by the the fact that the composite panel is lighter and they have different elasticity modulus. This may be evidence of the advantage of using hybrid structures in some aircraft engineering projects.
- The numerical simulation result and the experimental results are in good concordance for the hybrid structure and we are encouraged to continue to improve the models using NASTRAN capabilities to obtain a realistic simulation of aeronautical structures made of such composites. This analysis may be developed using new configurations of connections in numerical analysis and new configurations of tests.

REFERENCES

[1] E. F. Bruhn, *Analysis and Design of Flight Vehicle Structures*, Tri-State Offset Co, Cincinaty, ISBN-13: 978-0961523404, 1973.

[2] M. Niu, *Airframe Stress Analysis and Sizing,* Conmilet Press Ltd, 2nd Edition, New York, ISBN13: 978-9627128083, 2005.

[3] R. J. Roark & W. C. Young, *Formulas for Stress & Strains,* MCGRAW-HILL International, 7th Edition, New York, ISBN**:** 978-0-07-072542-3, 2001.

[4] A. Petre, *Calculul structurilor de aviatie,* Editura Tehnica, Bucuresti, 1984.