Numerical and experimental analysis of composite nacelle behaviour under different boundary conditions

HAVLÍK Vojtěch [1,a*], KAFKA Václav [1,b], HRON Robin [1], MLCH Ivan [1]

[1] Aerospace Research and Test Establishment (VZLU), Strength of Structures Department, Beranovych 130, 199 05 Prague – Letnany, Czech Republic

[a] vojtech.havlik@vzlu.cz, [b] kafka@vzlu.cz

Abstract: In order to proof composite part design the compressive stiffness test of engine nacelle demonstrator was carried out in VZLU. The nacelle was made from monolithic glass epoxy laminate and sandwich glass epoxy composite with honeycomb core. The aim of this article is to harmonize numerical boundary conditions with the real boundary condition of the nacelle test fixture. Identifying of the real boundary conditions will help to increase the precision of numerical model prediction capability. Deflection of nacelle near the place of loading was used to compare how the finite element method analysis results match with the values measured from the test.

Keywords: composite, honeycomb, nacelle, boundary condition, deformation shape.

1 Introduction

One of the ESPOSA project aims was to obtain most suitable composite materials and processes for small gas turbine engines nacelle. Nacelle demonstrator was made of glass epoxy laminate combined with honeycomb sandwich. The test was designed to control quality and strength of the tested nacelle.

2 Test description

Test arrangement is shown on the figure1. The nacelle fixture was designed to correspond maximally with real joint conditions on the airplane. Nacelle was supported on all free edges. Test was divided into two sub-tests with different loading places - see Fig. 4. First loading was up to force 350N and loading was applied in the rear area where the nacelle is made from sandwich composite. Second loading was up to force 200 N and loading was applied in the front area where the nacelle is made from monolithic glass epoxy laminate.

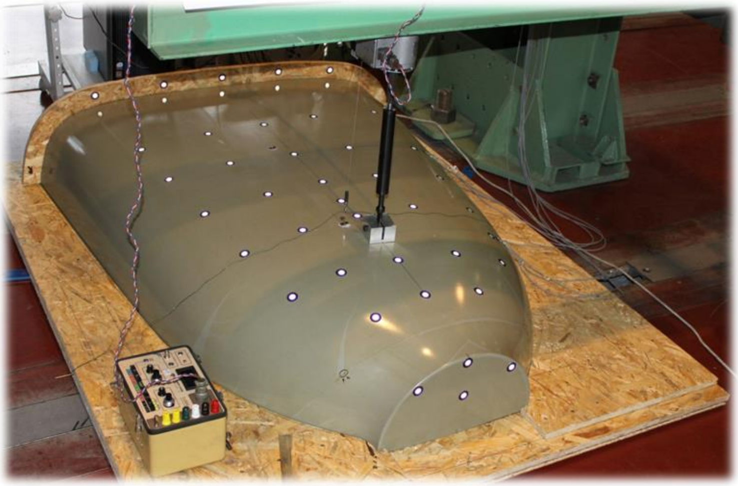

Fig. 1: Test arrangement

Fig. 2: Loading support foot

3 FEM model description

The aim of FE model is to harmonize numerical boundary conditions with the real boundary conditions during test. FE model was created in Patran. Due to assumption of geometrical nonlinearity the calculations in Nastran were carried out using nonlinear solver Sol 106. Material nonlinearities were not taken into account.

Loading in the place of monolithic glass epoxy laminate (Load 2) leads to bigger deflection and geometrical nonlinearities. Boundary condition influence on deformation shape is also bigger in this case. That’s why only results of the model for load 2 will be taken into account in the following chapters.

3.1 Materials

Material characteristics used for defining ACG glass-epoxy prepreg and for A1-48-5 Honeycomb are shown in the table1. Datas for ACG glass-epoxy were taken from [2]. Datas for Hexcel honeycomb were taken from [3]

Tab. 1: Material data.

	E_{11} [MPa]	E_{22} [MPa]	E_{33} [MPa]	G_{12} [MPa]	G_{23} [MPa]	G_{13} [MPa]	μ_{12} [-]
ACG glass-epoxy prepreg	23400	23400	8760	3700	1100	1100	0,2
HEXCEL A1-48-5 Honeycomb	25	40	140	-	-	-	-

Each of composite material is shown on the figure 3 with different color. Material angle 0° is oriented in direction of X axis on figure 3. Plane xz is symmetry plane of the nacelle. Axis Z is oriented vertically and in the same direction as loading force.

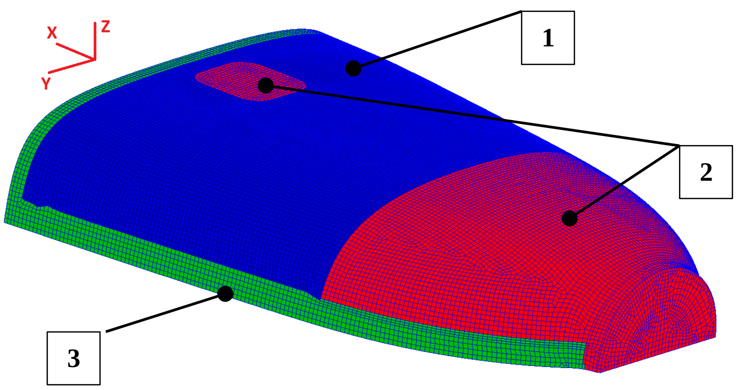

Fig. 3: Positions of monolithic laminate and sandwich composite

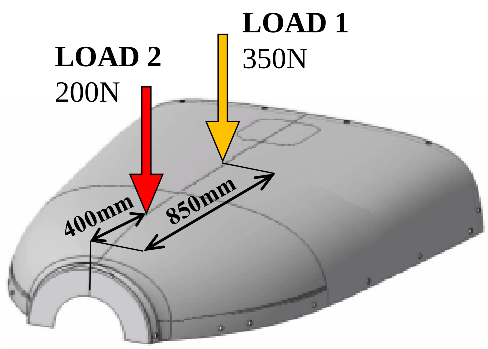

Fig. 4: Positions of loading

Sandwich composite structure made of glass epoxy-prepreg with honeycomb core is placed in area 1. Areas 2 and 3 represent monolithic laminate made of glass-epoxy prepreg. The orientation of each ply is written in the following list:

- 1 - sandwich composite - [45°/0°/Honeycomb/45°]
- 2 - monolithic laminate - [45°/0°/0°/45°]
- 3 - monolithic laminate - [45°/0°/-45°/0°/0°/-45°/0°/45°]

3.2 Boundary conditions and loading

There were two variants of modeling the boundary conditions. In the first case are all free edges of the nacelle constrained in all 6 degrees of freedom.

In the second case the gap elements were used on the free edges of the nacelle. In nonlinear solver gaps interact only in cases when they are in compressive stress state so in this way they substitute contact with nacelle fixture .Gaps enable translations of free edges only in the same directions as support fixture.

Due to avoid puncture of the material in place of loading the nacelle was tested with loading support foot 60x60mm.

4 Results comparison

Figures 7 and 8 show nacelle deformation shapes for different boundary condition. Each picture represents result for loading by vertical force 200N. To find the best boundary condition the deformation shapes from FEM analysis will be compared with the test deformation shape. Photo of the test deformation shape is shown on figure 9.

4.1 Deformation shapes

Figure 7 shows deformation shape of nacelle with free edges constrained in all 6 degrees of freedom. Figure 8 shows deformation shape of nacelle with gap elements on free edges.

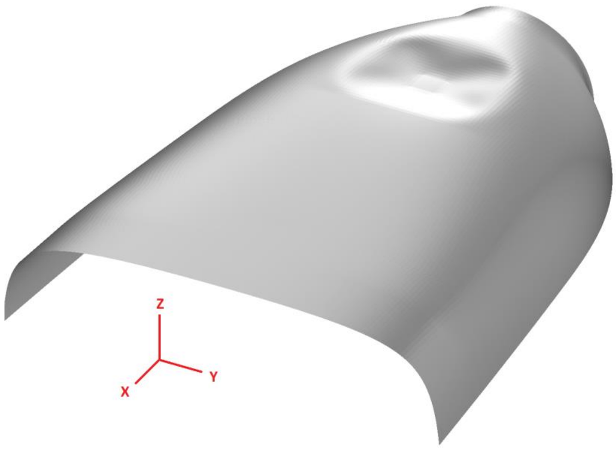

Fig. 7: Nacelle with constrained all 6 degrees of freedom on free edges

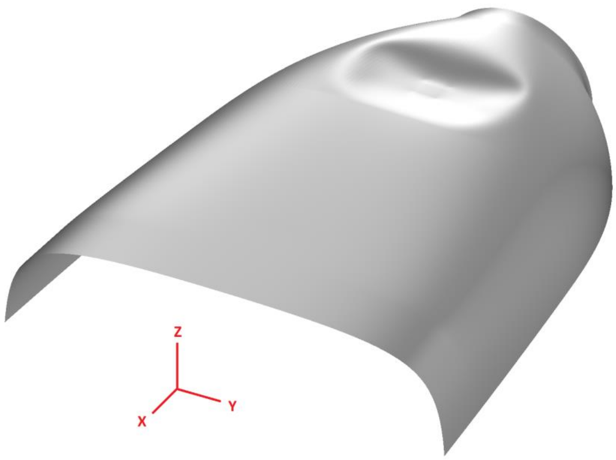

Fig. 8: Deformation shape of nacelle with gap elements on free edges

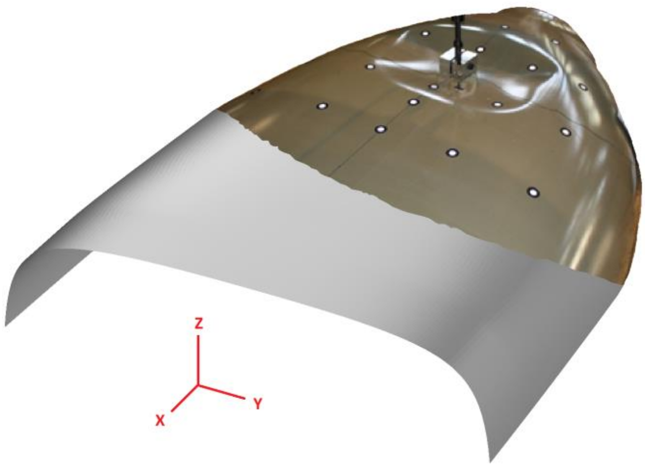

Fig. 9: Deformation shape of nacelle during test

Comparing results with the photo from the test on figure 9 the best fit in the deformation shape is with figure 8. That’s why the boundary condition with gap elements on free edges was chosen as the best way to model the nacelle test from deformation shape point of view.

4.2 Deflection of the nacelle

To compare FEM results with some measurable parameter the deflection of nacelle was chosen. The deflection was measured near the place where the load was applied. Comparison of deflection from FEM and test is shown on figure 10. The deflection in the monolithic structure for 200 N was during test 29,43 mm. The FE model shows in the same place deflection 22 mm. Difference between test results and FEM analysis may be caused by manufacturing of the nacelle. That’s why the thickness of nacelle can be in some areas slightly different from the thickness which was modeled in FEM model.

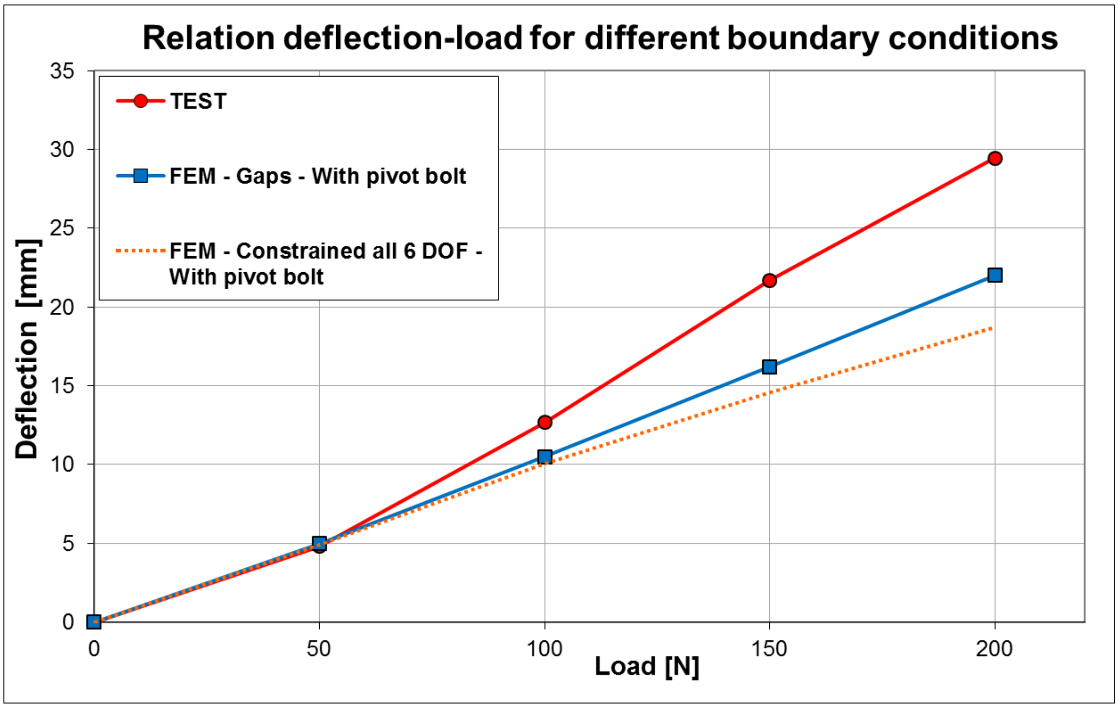

Fig. 10: Comparison of nacelle deflection for different boundary conditions

5 Conclusion

As it was shown in this article correct simulation of the boundary conditions has significant influence on the deformation shape. In the case of nacelle compressive test using of boundary conditions modeled by gap elements correspond better with reality then simple constraining of nacelle free edges. Identifying of the real boundary conditions helps to increase the precision of numerical model prediction

Acknowledgement

The research has received funding from the European Union’s 7FRPfor Research, Technological Development and Demonstration within ESPOSA (Integrated approach to efficient propulsion and related aircraft systems for small-size aircraft) project.

References

[1] V. Kafka, Report on test run on sub-scale component BE1, ESPOSA Deliverable WP6.5/D65.25 (2015)

[2] D. Szeląg, A. Osmęda, Detailed FEM analysis BE1, ESPOSA Deliverable D6.5.7 ILOT (2014)

[3] Retrieved from: http://www.hexcel.com/Resources/DataSheets/Honeycomb-Data-Sheets/A1A10_eu.pdf