26th International Conference
ENGINEERING MECHANICS 2020
Brno, Czech Republic, November 24 – 25, 2020

FAILURE ANALYSIS OF CARBON FIBER LAMINATE AIRCRAFT STRUT MODEL

Vomáčko V.*, Zbončák R.**

Abstract: *The aim of this work is to verify functionality of an aircraft strut made of carbon fiber laminate. This goal was achieved in several steps. The aircraft strut model was manufactured and used for two types of mechanical tests. Prior to the tests, FEM simulations had been executed. The simulations predicted the failure mechanism correctly and offered good estimation of critical load. The functionality of the aircraft strut was verified.*

Keywords: Composite material, CFRP, Failure criteria, Numerical simulation, Failure testing.

1. Introduction

The use of continuous fiber composite materials for structural parts is favorable thanks to high strength-to-weight ratio (Laš, 2008). This feature was exploited during design of empennage of the plane which took place at VÚTS, a.s.. Numerical simulations were used for strength design of aircraft empennage and safety factor was monitored.

In case of orthotropic materials like continuous fiber composites we have to evaluate every orthotropic direction and load type separately. In the case of multilayer laminates occurs distribution of stress through layers according to orientation of individual layers. The safety evaluation of laminate structures is therefore demanding and important activity (Zbončák, 2018). Failure criteria were developed to enable evaluation of large structures like an aircraft. A number of general and specific failure criteria have been developed throughout history but all have limited area of validity (Ansys, 2019). In specific as well as critical cases, it is therefore necessary to verify the validity of the used failure criteria.

In some cases, such as the strut of this aircraft empennage, the results of the simulation differ significantly from the empirical experience of experts. Due to given reason arose the need to verify validity of failure criteria for continuous fibre composites for the case of the strut of the aircraft empennage.

2. Fabrication of empennage model

CAD model was designed in Solidworks 2018. Detail of the empennage is in the Fig. 2. The top part is a sandwich structure. There is polymer core AIREX T90.100 between two laminates. Two types of carbon fibres were used. Unidirectional fibres 125 g/m^2 and plain fabric 160 g/m^2. As a matrix was used epoxy resin LH 385 with hardener H 286. Each laminate skin consists of 4 layers [0F, 0, 0, 0F]. We will focus on the strut which consists of closing skin and part of bottom skin. Closing skin serve as a contact surface for connection to aircraft body and as an element that prevents from peeling between mid and bottom skin. Most of laminate parts are manufactured using vacuum infusion process. Bottom skin is made by wet layup. The empennage model is simplified compared to the original. The model is not V-shaped but is flat to avoid the production of expensive mold. Despite the simplification, the strut remains similar and thus the model is sufficient.

* Ing. Václav Vomáčko: [1] Institute of New Technologies and Applied Informatics, Technical University of Liberec, Studentská 1402/2; 461 17, Liberec; CZ, vaclav.vomacko@tul.cz, [2] Computations and Modelling Department, VÚTS, a.s., Svárovská 619; 460 01, Liberec; CZ, vaclav.vomacko@vuts.cz

** Ing. Radek Zbončák: Computations and Modelling Department, VÚTS, a.s., Svárovská 619; 460 01, Liberec; CZ, radek.zboncak@vuts.cz

doi: 10.21495/5896-3-516

Fig. 1: Left half of fabricated empennage model from carbon fiber laminate and detail of co-cured parts.

3. Empennage test

In the first mechanical test and numerical simulation there is imitation of lift force from operating conditions acting on the empennage. The test design can be seen in the Fig. 3. Empennage is screwed on steel cube which substitutes rigid aircraft body. Lift force is simulated by system of ropes and clamps which are raised by the lever.

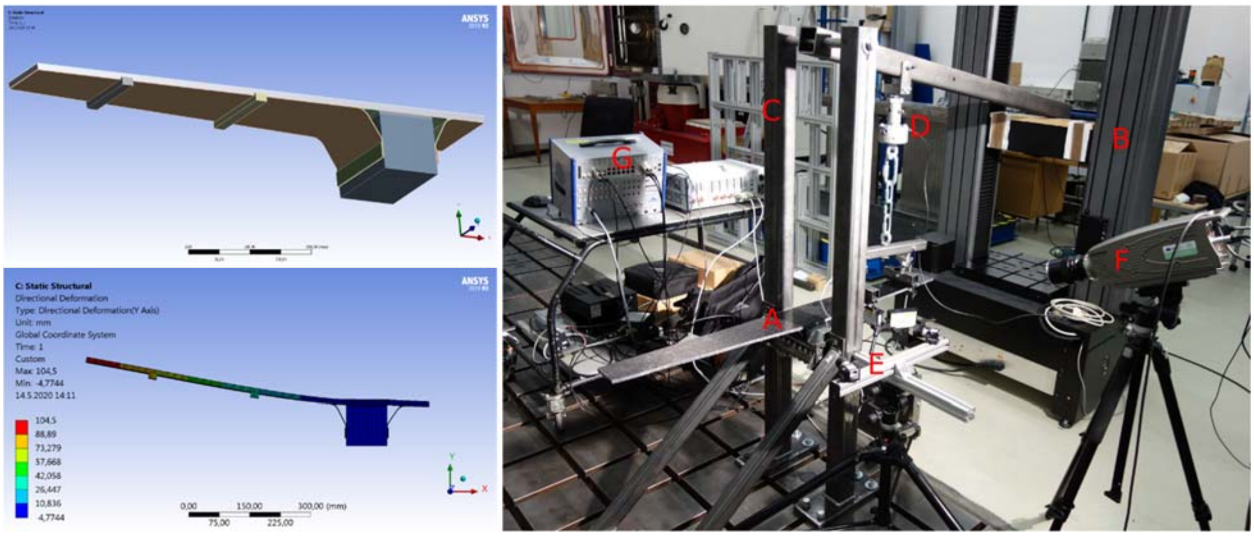


Fig. 2: On the top left geometry for simulation. Down on the left is displacement in the vertical axis. On the right is equipment for the empennage test: A – empennage, B – testing machine. C – test rig, D – load cell 3, E – optical measuring system, F – high speed camera, G – measuring analyzer.

3.1. Simulation of empennage test

Simulation models are made in ANSYS 2019 R3 software. For definition of mechanical properties of laminate parts was used module ACP. Simulation model is set to respect fixations and loads as in performed test. Fixed support is set between strut and steel cube which represents attachment by screws. Loading force is applied on the bottom surfaces of two clamps of loading system. Solid model type and quadratic finite elements are used and large deformations are enabled.

3.2. Empennage test

Test was performed by universal testing machine Instron 5982 100 kN. For measurement of displacements were used two laser sensors Micro-Epsilon optoNCDT model ILD2220 with range 100 and 200 mm. Force was measured in three points. Two force sensors HBM U2B 10kN were placed above clamps of loading system. Third force sensor was placed in the top of the loading system and therefore measured total applied force. Displacements and force were processed by measuring analyzer Dewetron.

3.3. Results and discussion of empennage test

Fig. 4 shows shear stress in the XY plane in the polymer core. The maximum value of shear stress in core 1.1 MPa at maximum loading 560 N is in the area above attachment to aircraft body. This value is higher than shear strength of core which is 0.8 MPa. The simulation thus predicts shear failure in the core. The critical value 0.8 MPa is reached with overall loading force 410 N. Maximum value of failure index according to all observed criteria (Max stress, Max strain, Tsai – Hill, Hoffman, Hashin) does not exceed critical value 1 in any area of the empennage. The laminate parts therefore should remain unbroken.

In the test the relationship between overall force from sensor 3 and displacement is linear up to a load of 400 N which corresponds to the critical value from simulation. With load increasing is the linearity disturbed which can be caused by nonlinear behaviour of the core. The total failure occurs at value 560 N. In the Fig. 4 is shown the crack in the core. The crack is at an angle 45° which points to shear failure as was indicated in the simulation.

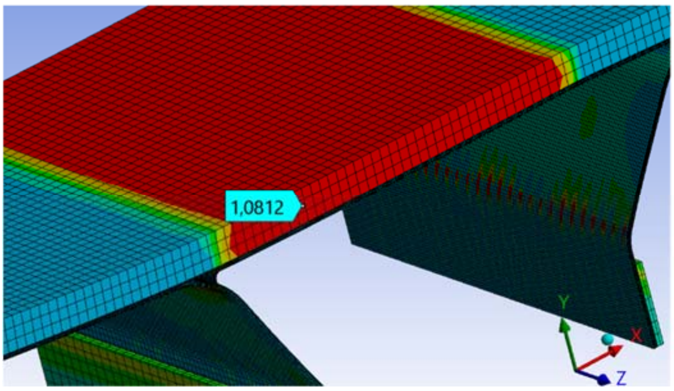
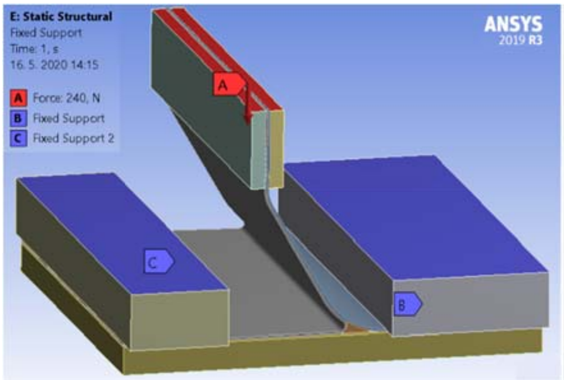


Fig. 3: The shear stress in the core in the simulation and the detail of the crack in the empennage test.

4. Strut test

The goal of the strut test is to determine load capacity of the strut. The equipment for the test is seen in the Fig. 5 right. The empennage is clamped to testing machine and the strut is bent by compressive loading.



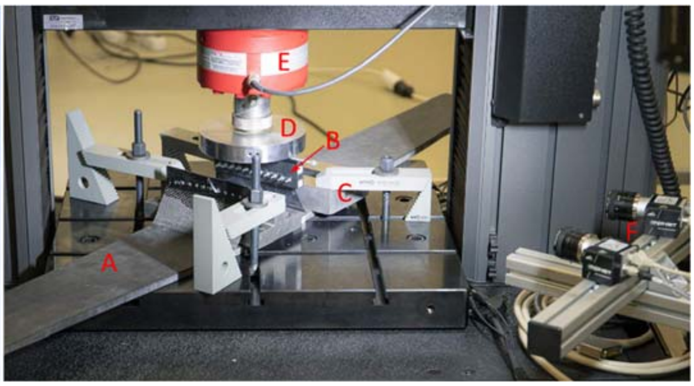


Fig. 4: Load and constraints in the simulation of the strut test and the equipment for the test: A – empennage, B – tool with bearings, C – clamping, D – pressure jaw, E – load cell, F – optical measuring system.

4.1. Simulation of the strut test

Modified geometry, loads and constraints in the strut test simulation are shown in the Fig. 5. On the blue surfaces was applied fixed support and on the red surfaces was applied force in the vertical direction with magnitude 240 N (10 N higher than maximum from test). It is considered only half of the empennage due to symmetry to decrease a size of the simulation model. Solid model type and quadratic finite elements are used and large deformations are enabled.

4.2. Strut test

The principle of the test is bending of the strut by compression loading of testing machine. When crossbar of testing machine moves downwards then the strut moves in vertical and also horizontal direction. Therefore, it is necessary to enable horizontal movement for the strut. It is done by tool with bearings which rolls on the compression jaw. The tool was fabricated on 3D printer Markforged Mark Two from Onyx material. Empennage was attached to T-groove table of the testing machine by two steel plates and clamps.

Force was measured by load cell Instron 2580-100 kN. Displacements were measured by optical system Sobriety Monet.

4.3. Results and discussion of the strut test

In the simulation the critical area seems to be the one among mid, bottom and closing skin which is filled by unidirectional fibres and resin. The stress in the normal direction to the surface is 7.5 MPa almost all over the area. In the small area there is the peak of the normal stress reaching 12.6 MPa. It is value which is very close to strength of unidirectional fibres in the transversal direction which was measured 12.8 MPa. This value responds to the strength of the interface between fibre and resin which is exactly the case of the critical area. Normal stress above the limit 12.8 MPa would damage the interface among mid, bottom and closing skin and the delamination would occur.

In the test the failure was caused by delamination in already mentioned area among mid, bottom and closing skin. Maximum acting force was 230 N. Detail of the failure and comparison of the test and the simulation is shown in Fig. 5. The plot shows displacement in Y direction for the monitored point in the end of the strut.

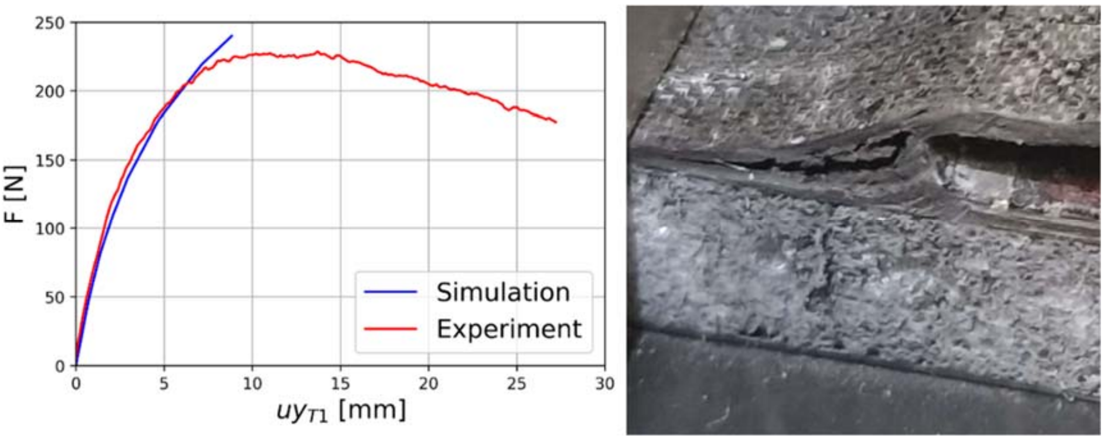


Fig. 5: Plot of the relationship between compressive force and displacement in Y axis of the point in the end of the strut- comparison of simulation and experiment. And detail of the delamination in the test.

5. Conclusion

Simulations focus on two critical areas of the empennage - sandwich structure in saddle area and root of the strut where three skins meet. The failure of laminate parts was not predicted by failure criteria in any of the simulation. The simulation results are then verified by two independent experiments. Simulations were compatible with experiments, predicted failure mode correctly and also offered good estimation of critical loads. Interest was focused on the strut, which remained unbroken and thus was verified its functionality.

Acknowledgement

This work was supported by the Student Grant Competition of the Technical University of Liberec under the project No. SGS-2019-3054. Materials and testing equipment were provided by VÚTS, a.s..

References

ANSYS, Inc. *Guide to Composite Failure Criteria*. 2019. Online, available at: https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v195/acp_ug/acp_compmodel_compfailure.html. Accessed: 2019-05-02.

Laš, V. (2008) Mechanics of composite materials. 2nd ed. Pilsen: Západočeská univerzita, ISBN 978-80-7043-689-9, (in Czech).

Zbončák, R. 920180 Strength criteria for composite materials.1st. ed. Liberec: VÚTS, a.s., ISBN 978-80-87184-82-0, (in Czech).