NONLINEAR BEHAVIOR OF FOAM MATERIAL

Ing. Martin Weis
Institute of Aerospace Engineering
Technická 2 Brno
Czech Republic
weis@fme.vutbr.cz

Abstract. Nowadays, composite materials are often used in aircraft structures. The most common construction is sandwich structure. Very important part of sandwich structure is core material. For core material can be used honeycomb or foam. In our tests herex foam material was used. The following article presents the results of experimental tests to determine nonlinear behavior of herex foam. Compression tests were done for finding module of elasticity and compression strength. Shear tests were done for finding shear modulus and ultimate shear stress. Consequently was performed verification finite elements analysis. This data can be used for design of sandwich structure and its load capability.

Keywords: foam, stress, nonlinear, sandwich structure, FEM

1 Introduction

This article describes properties of foam material and is based on tests which were performed at Institute of Aerospace Engineering, Brno University of Technology. The motivations for conduction of these tests were previous tests of sandwich structures where low load capability leaded to failure of all structure.

Sandwich structures are widely used for their low weight and high stiffness. Typical using of sandwich material is fuselage of light airplanes, wing skin and wing web. Sandwich structures are usually composed from two face sheets and core. Face sheet is composed from carbon or glass/epoxy laminate and core is from foam material.

Design is generally focused on stress analysis of laminate face sheet and core analysis is little bit neglected. The reason is quite simple. Geometrically complicated parts like fuselage are usually analyzed by FEM with using 2D elements. It is very effectively method for design layers of face sheets, but usually we don't know stress through thickness. This neglect of through thickness stress lead to failure of whole construction. In Figure 1 is landing gear which was break in drop test. Glass laminate was designed correctly but core crushing leaded to failure. The figure shows damage of blue foam core. The landing gear was made again but with different core material with much higher compression strength. Another drop test demonstrated importance of foam core for load caring capacity. The second example is VUT 061 TURBO fuselage (Figure 2). Whole fuselage is laminate sandwich structure with foam core and carbon/epoxy face sheet. It was tested and break occurred in central section. One of the main reasons was insufficient compression load capacity of foam core.

For quality and appropriate stress analysis is necessary to used combination of 2D elements at face sheet and 3D elements for foam core. The reason is to gain through thickness stress in foam core.

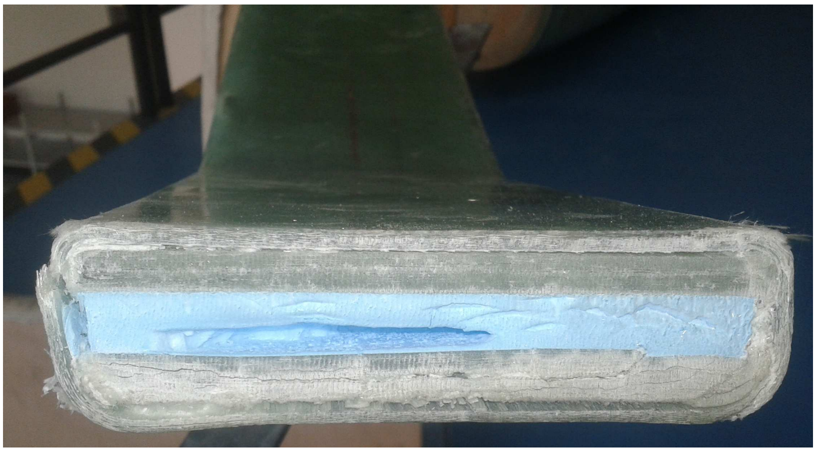

Figure 1: Landing gear core crushing.

Figure 2: Damage area of VUT 061 Turbo fuselage.

Experience from previous tests was the motivation for further exploring issues of foam behavior. It was necessary to perform tests of foam materials and its validation with FEM analysis. Pre and post processor was used MSC Patran and MD Nastran like a solver. In analysis were used different types of material models from the simplest to the most complex. Geometry from previous test was readmitted and load was applied with corresponding boundary condition to obtain the same condition in analysis and tests. Good agreement was achieved with comparison of analysis and tests.

2 Basic tests of foam material

To ensure the quality of the sandwich structure design is necessary to performed basic test. Two basic tests were selected for its simplicity and efficiency in design. The first of them is compression tests (Figure 2.1) and second is shear test (Figure 2.2). Tests were performed at universal test machine with 20kN load cell. For displacement measuring was used external displacement meter during compression test respectively extensometer for shear test. Measured data were used for verification of FEM analysis.

2.1 Compression tests

The first test of foam material was select compression test. Two different materials were tested. Was used yellow Herex C70.55 and green Herex C70.75. Five square specimens of each material were cut from plate with a thickness of 10mm. In center of specimen was drilled hole for displacement measurement. The specimen was places between two plates. Upper was fixed to load cell and bottom plate was compensated on a spherical surface. The test arrangement is shown in Figure 3. The resulting perfectly plastic behavior of foam material is evident in Figure 4.

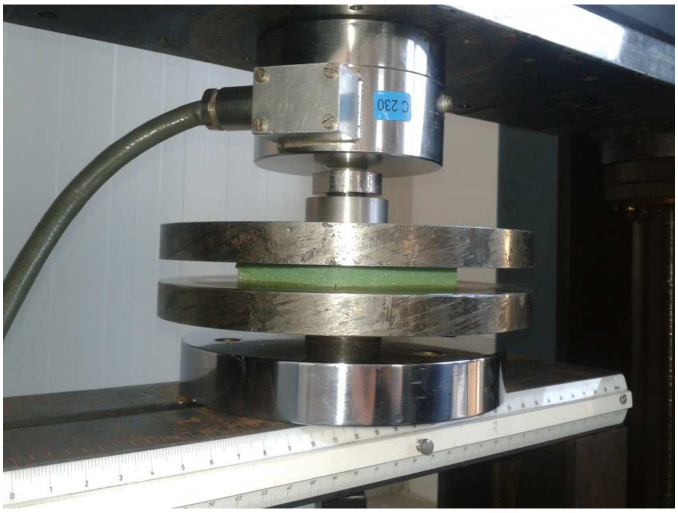

Figure 3: Compression test of “green” herex

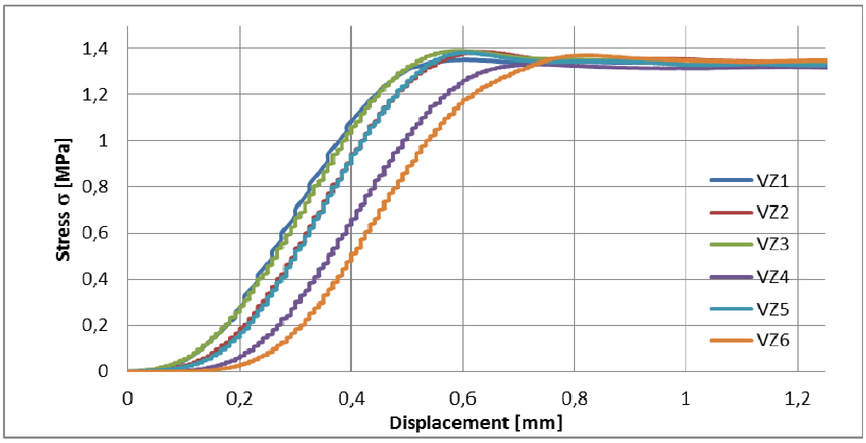

Figure 4: Results of compression tests

2.2 Shear tests

The second test of foam material was select shear test. This test was performed according to ASTM C 273. Two different materials were tested. Green Herex C70.75 was used. Five rectangle specimens of each material were cut from plate with a thickness of 10mm. Each specimen was glued to two load carriers. The specimen was clamped to test machine and loaded to failure. Displacement was measured by extensometer and displacement meter. The test arrangement is shown in Figure 5. The resulting shear behavior of foam material is evident in Figure 6. Blue curves are data from extensometer and purple curves are from displacement meter.

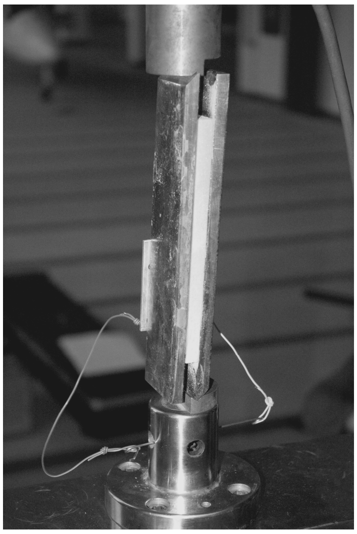

Figure 5: Shear test of “green” herex

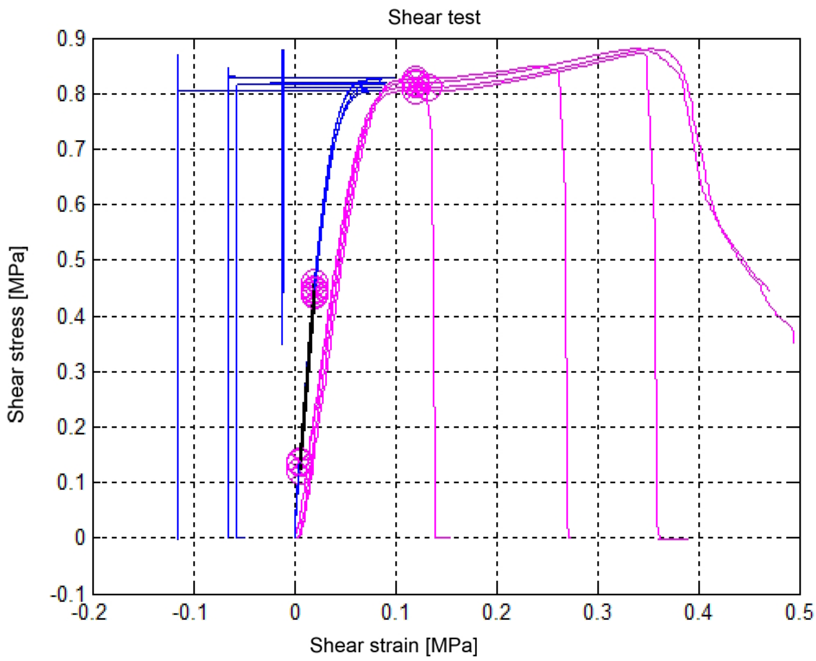

Figure 6: Results of shear test

3 FEM simulation

FEM simulation is widely used for design of strength of structure. In order to use the analysis of complex structures it is necessary to start from the simple specimen and continue to complete construction. The first step is definition of material behavior. Second step is verification of simulations and measured data for the material model. After this verification, we can proceed to the complex simulations of structures.

3.1 Material definition

Linear material

Foam core can be described by a lot of options. In FEM analysis exist a lot of ways, how to describe behavior of material. Some of them are described below. Very simple method is describing by isotropic behavior, which is interpreted by modulus of elasticity E and Poisson ratio μ. This linear definition is very simply and we can get correct results, but it's necessary to control results and verify usually maximal shear and compression stress, which can lead to failure of structure and indication of lower stress in face sheet. This material definition is usually used for preliminary design with 2D elements.

E	μ
[MPa]	[-]
54	0.071

Table 1 - Isotropic behavior of herex foam

Perfect-plastic material

Much better is using of perfect-plastic material. Advantage of using this material is bi-linear perfect-plastic behavior which is sufficiently accurate for the most application. Bi-linear material is described by modulus of elasticity, Poisson ratio, hardening slope and yield point.

It is very simple to get material data from basic tests. For describing basic behavior of material must be performed shear and compression test and compute important characteristics.

E	μ	Yield point	Hardening slope
[MPa]	[-]	[MPa]	[MPa]
54	0.071	0.8	0

Table 2 -Bi-linear behavior of herex foam

Hyper-elastic material

Previous material definition is very simple and efficient, but it's not absolutely physically correct. Isotropic and elastoplastic material are determined for incompressible materials.

The most complex and physically accurate describe of foam material is hyper-elastic model. Description of this material is much more complex than previous method and requires number of inputs. For definition of hyper-elastic material is necessary to performed uniaxial, biaxial, planar or volumetric tests. MSC Software contains utility which is able to compute constants of hyper-elastic material from measured test data.

3.2 Comparison of measured results with FEM simulation

Previous tests of mechanical properties were simulated using FEM. Was achieved very good agreement between measurements and simulations. It was necessary to simulate the compression and shear tests of foam. After obtaining satisfactory results were forwarded to simulation of sandwich structure by four-point bending.

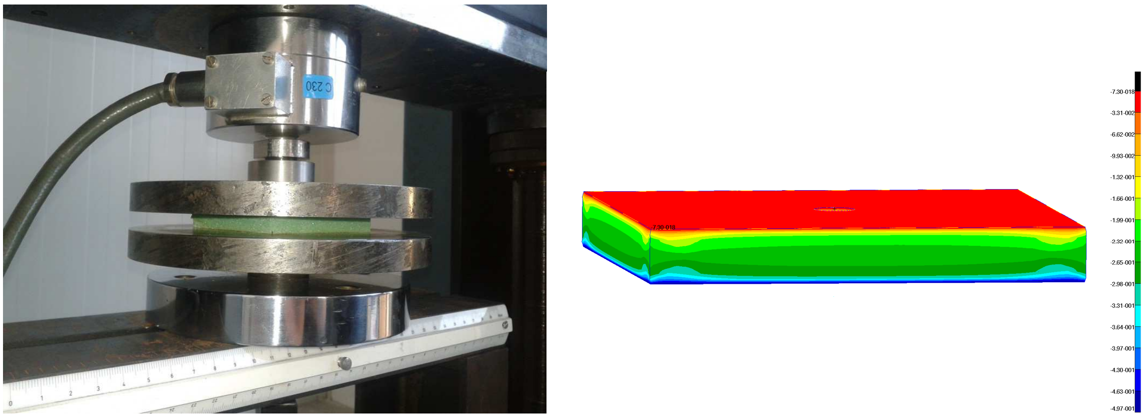

Figure 7: Compression test and FEM simulation

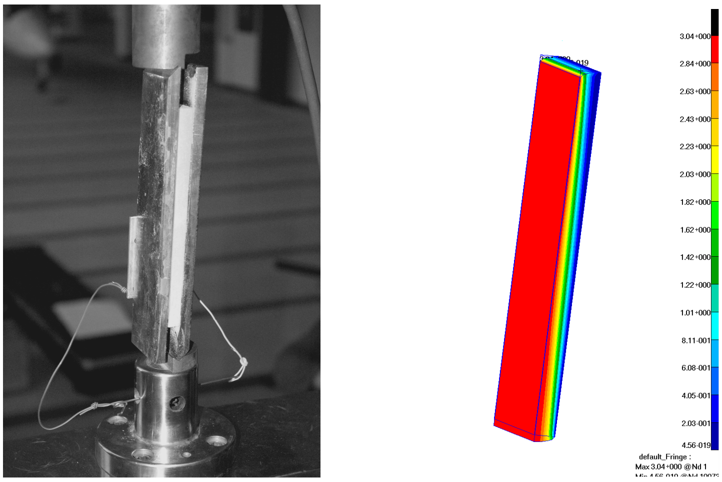

Figure 8: Shear test and FEM simulation

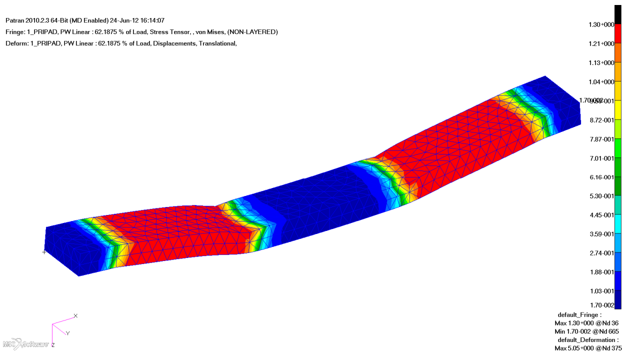

Figure 9: FEM simulation of bending test measurement

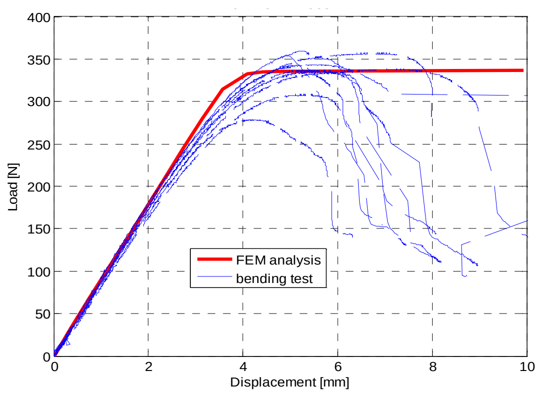

Figure 10: Graph comparing FEM and

3.3 Application previous knowledge on real construction

The following chapter is focused on FEM simulation of VUT 051 Turbo fuselage. For design of laminate layers was used complex FEM model of whole fuselage (Figure 11). 2D elements were used on the entire body. The structure was designed and tested. The test showed a lower load capacity of fuselage. The problem was a failure of foam in sandwich structure. This failure caused a return to the model, and troubleshooting.

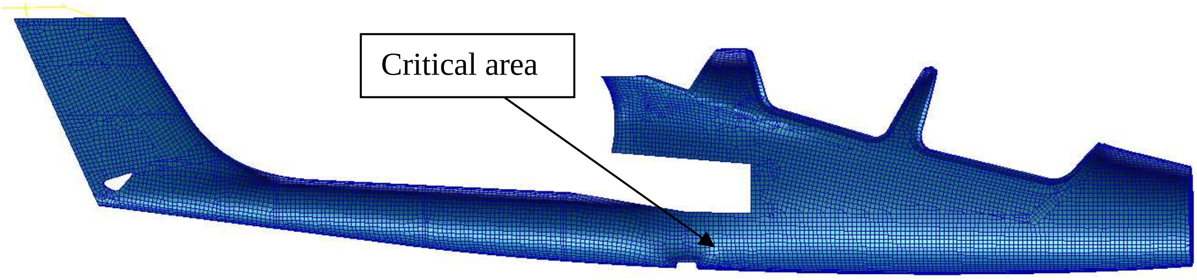

Figure 11: FEM model of fuselage

From global model was created smaller one. Condensation of model was used. Than was obtained new model of critical area. It was necessary to apply corresponding displacement boundary conditions. Subsequently was used division into smaller elements (Figure 12).

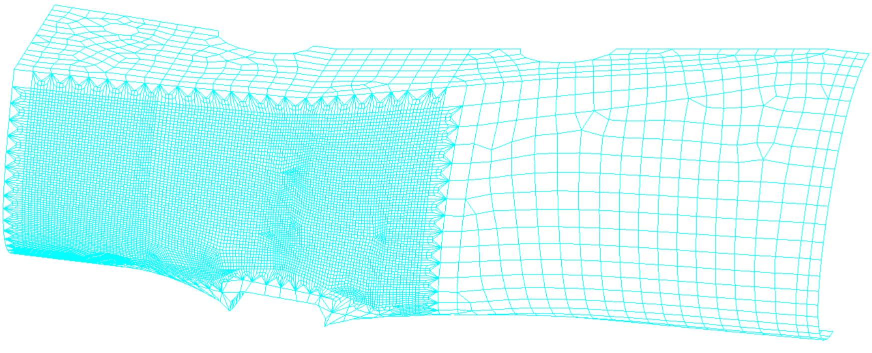

Figure 12: Condensed model of fuselage

This model with a finer mesh didn't show a problem in this area. That was the reason for the creation of much more complex model using 3D elements for foam core. For core material was used bilinear material model. The result of this analysis was crucial. In following (Figure 13) we can see completely different distribution of deformation.

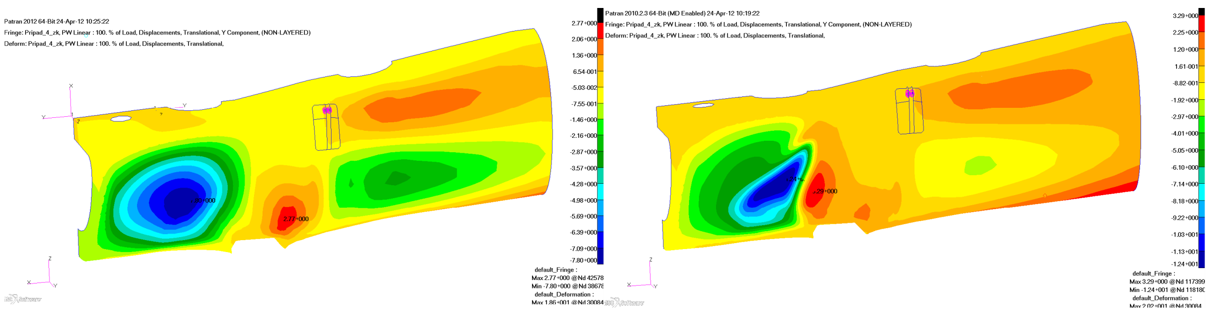

Figure 13: Condensed model of fuselage (buckling distribution)

On the left side of picture is model completely composed with 2D elements and on the right is model with 3D foam. In both models is face sheet formed by 2D elements.

Due to the buckling of sandwich structure and small compression strength of foam in this area occurred the failure of sandwich structure. Due to this failure of foam has been increase the stress in layers approximately twice from 300MPa to 600MPa. Distribution of stress is completely different (Figure 14).

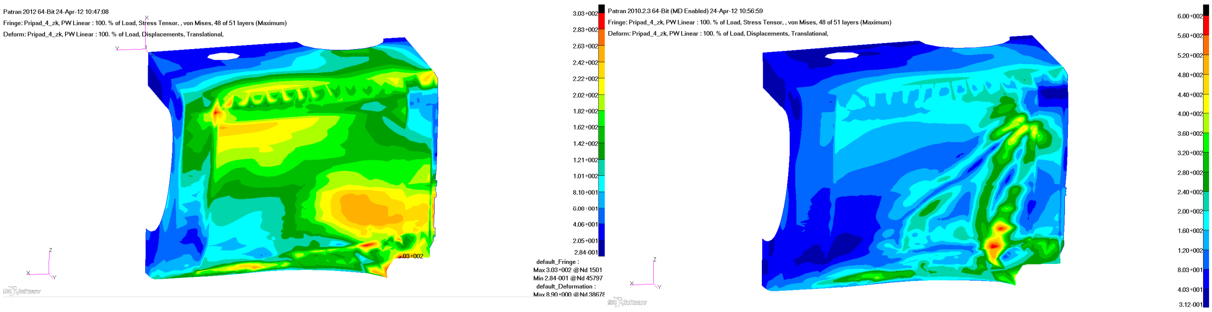

Figure 14: Condensed model of fuselage (stress distribution)

The analyses with 3D elements show important results. Until 65% of ultimate load is possible to observe one big wave on the skin and maximal stress is approximately 307MPa. During increasing of load about 10% of ultimate load occurred change location of buckling wave and stress increase to 473MPa. The reason for this situation was the creation of the next wave. This change of bucking wave and increasing of stress could lead to failure of structure. This change is well defined in following Figure 15.

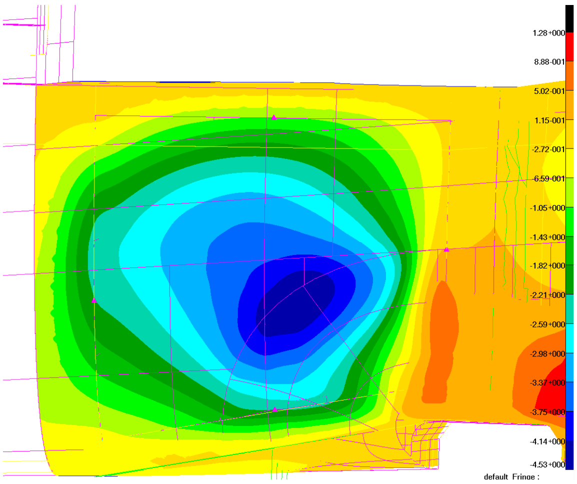

Figure 15: Buckling at 65% of ultimate load

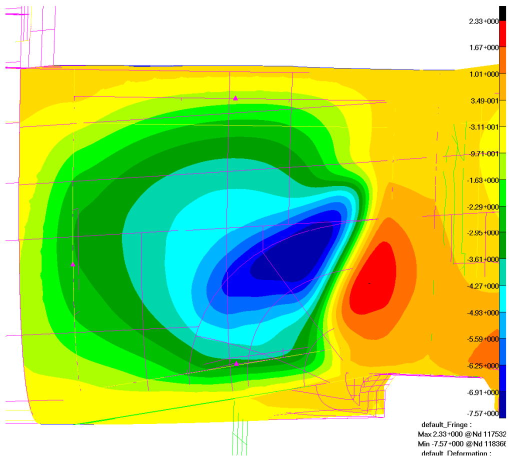

Figure 16: Buckling at 75% of ultimate load

The test was measured by acoustic emission and we can observe good agreement in FEM results with 3D elements and real measuring from test of fuselage. In figure 17 is shown cumulative frequency of acoustic events during test. Its localization is identical with simulation.

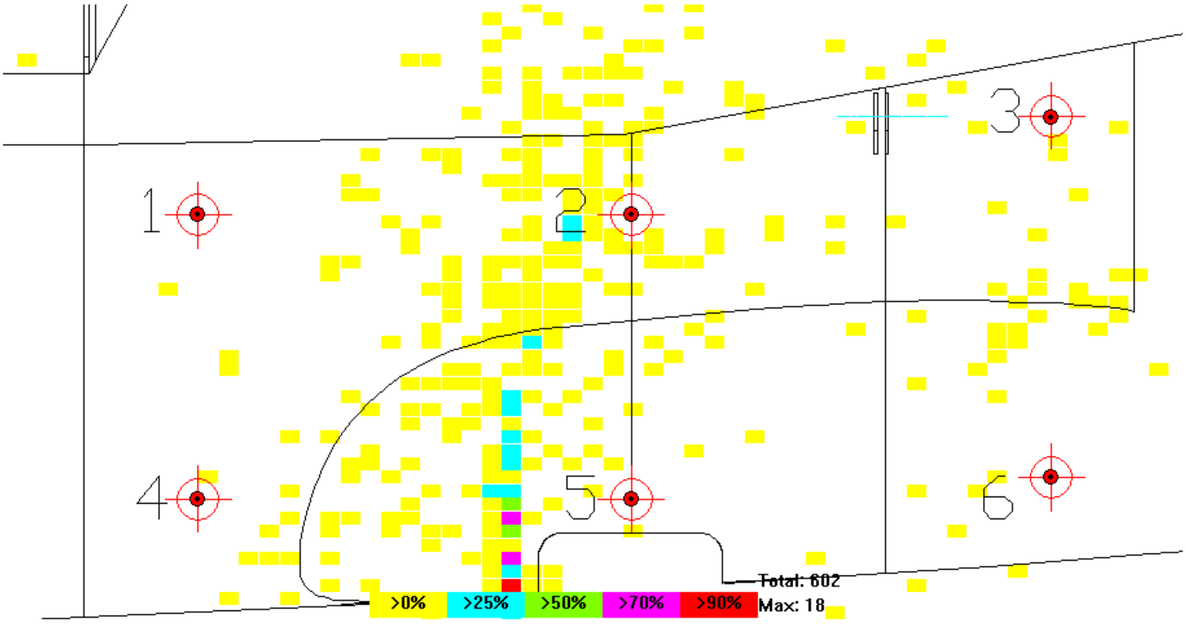

Figure 17: Localization of acoustic emission

4 Conclusion

FEM is a very powerful tool for the simulation of the structure. However, it is necessary to proceed from the simple to the complex simulation for its verification. The use of 2D elements for the simulation is very popular for the simplicity and speed of calculation. But we must be careful to neglect of through thickness stress. This through thickness stress can lead to failure of sandwich structure. Analysis of 2D elements is very suitable for the preliminary design of laminate layers. The result of this simulation is the identification of critical areas and deformation of structure. To determine the load capacity of the structure is necessary to use 3D elements for foam core. It is not possible because preparation of this simulation take a lot of time. Much better is using of condensation model from global model. We can change 2D elements of foam to 3D elements and get a very important through thickness stress.

5 References

[1] Weis M.: Tlaková zkouška zeleného herexu, Test report number: P12/2012, 2012
[2] Mališ M., Urik T. Čplíchal J. Cejpek J..: Zjištění modulu pružnosti ve smyku a meze pevnosti materiálu jádra sendviče, Test report number: P32/2011, 2011