

Project: High performance lightweight panels with a new optimized design for advanced aircraft.
Acronym HIPEAS

Ctr. 206/2012 Financed by State Budget - Contracted Authority Unit Executive for Funding Education Higher, Research Development and Innovation (UEFISCDI), Program PNCD II, Period 2012-2016

Coordinator: UPB-CCMA, **Partners:** STRAERO SA, Universitatea Politehnica Timșoara, INAS SA, Smart Mechanics SRL

Project presentation

HIPEAS project aims design of the aircraft sandwich panels, manufactured from metal and composite materials, plane and curves forms with high performances. This is an innovator project and has at his base the following ideas:

- identification of the new solutions of mechanical interconnection for sandwich panels using elements that are in the core of the sandwich panel, which are simple, more efficient and more cheaper than the usual used; interconnection of the upper an bottom face of the sandwich panel is more efficient than the rigid polymeric foam core that leads to decrease the delaminating risk and by that this procedure increase the stiffness and the strength of designed panels
- properties evaluation of sandwich panels with ultra light cores spatial folded; it is desired the increase of performances with maintaining the geometry of the honeycomb inside the panel
- proposal and evaluation of simple solutions for manufacturing orthotropic panels with high carrying capacities with two or three cores, with different properties and geometry
- proposal of hybrid assemblies (structural bonding and mechanic fixing) sandwich panels with faces and cores from different materials
- using the prefabricated and usual materials such that the design, manufacturing and homologation costs will be at minimal level
- developing the procedure for mechanical response simulation of the panels using the homogenization method for easy and efficient design of the process

The main objectives due to manufacturing process of the new panels, with better properties are presented as follow:

- study of the design approaches, materials and technologies for light panels, used in structural of the aircraft. New construction solutions are proposed
- static and cyclical testing of the joints for structural adhesive behavior characterization on ambient temperature, low and high temperature on -50° C to 150° C range
- parametric 3D simulation for linear and nonlinear analysis, in order to characterize the proposed sandwich panel for evaluation; optimizations
- parametric 3D simulation and analyzing for evaluation of the curved sandwich panels proposed for the study; optimizations

- manufacturing the experimental sandwich panels models, plane and curved and testing of them.

First stage consists in studies of constructive variants; materials, manufacturing technologies and testing methods of lightweight panels for aircraft structures; developing the methodology of calculus and propose of new constructive solutions. Following there are presented some models of honeycomb used in aircraft industry as well as an finite element model for sandwich panel simulation analysis.

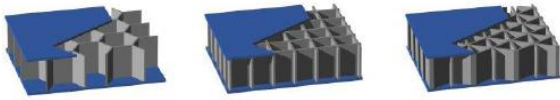


Fig. 1 Honeycomb models (hexagonal, square, triangular)

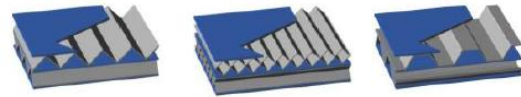


Fig. 2 Prismatic core models



Fig. 3 Lattice truss tetrahedral, pyramidal, 3D Kagome structures

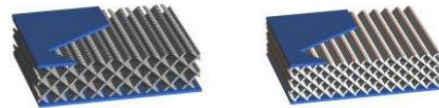


Fig. 4 Textile core structures

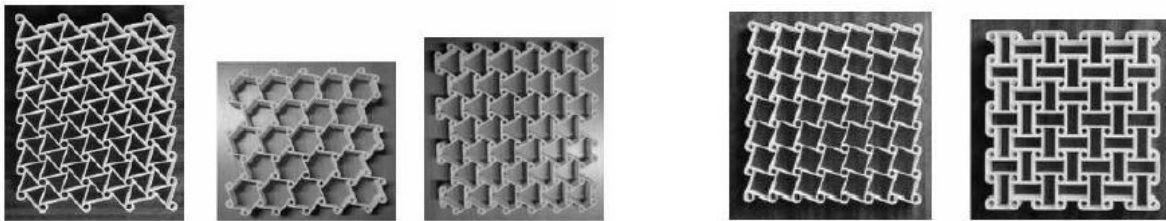


Fig. 5 Auxetic honeycombs

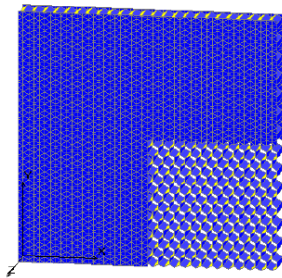


Fig. 6 Finite element model with Shells of the honeycomb panel

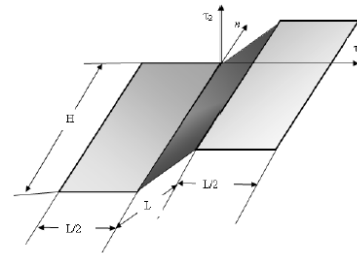


Fig. 7 Representative cell of the honeycomb model

The second stage consist in adhesive testing chosen for low temperature; characterization of static properties of composite materials from which the sandwich panel will be manufactured; numerical simulations of the tests: traction on double strap bonds; parametric simulation and finite element analysis of a sandwich panel with cellular core from polyurethane rigid foam.

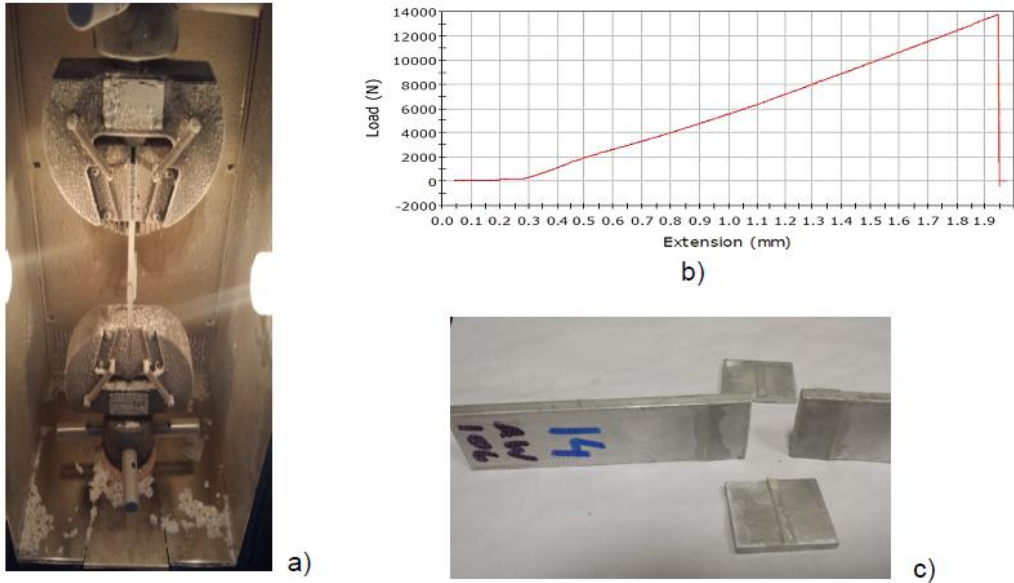


Fig. 8 Double strap bonded specimen with AW106 adhesive, tested at -40°C

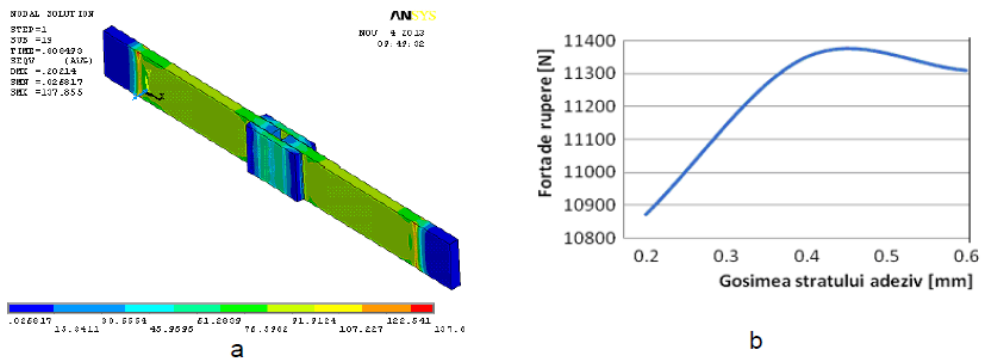


Fig. 9 Numerical simulation of shear testing for double strap bonded, a - equivalent assembly tensions, b - the influence of the adhesive thickness

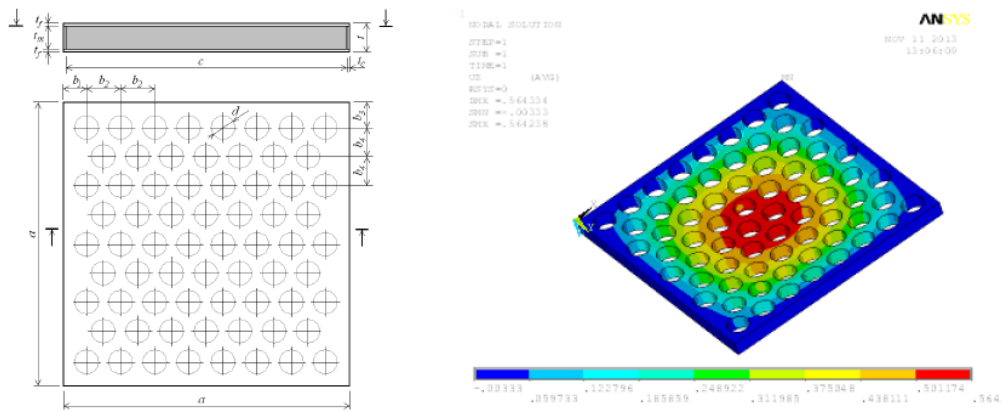


Fig. 10 The geometry of the panel and the resulting displacements map with a pressure load 0,07MPa

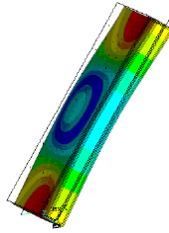


Fig. 11 First global buckling mode

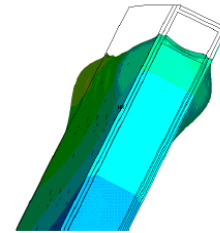


Fig. 12 Micro buckling of Al face

The third stage had the next targets: fatigue limit evaluation of cycling tests on single strap bonded; static analysis and evaluation of buckling risk for a plane panel (PSP1); fatigue analysis of plane panel (PSP3); fatigue analysis of cylindrical panel PSC1 subjected to pressured-depressurization cycles. In figure 13 and 14 are presented the fatigue limit evaluation subjected to cyclic loads for simple strap bonding.

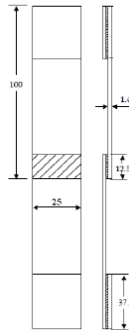


Fig. 13 Simple strap bonding specimen

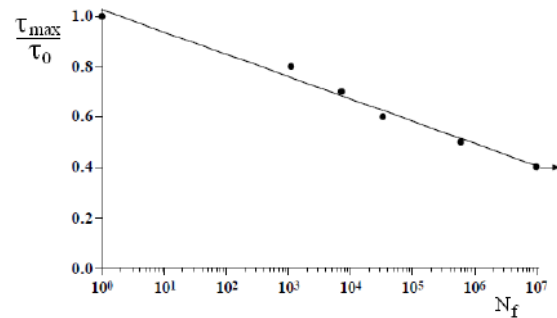


Fig. 14 Life cycle curve for adhesive tested on cyclic oscillator shear

Next follows statics analysis, stability and life cycle fatigue for plane and cylindrical sandwich panels.

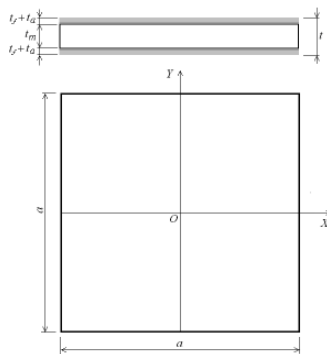


Fig. 15 General view of plane panel

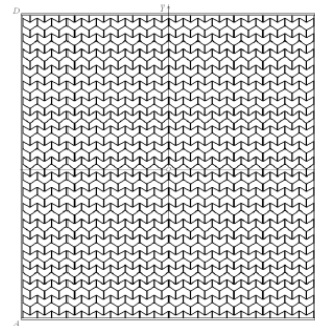


Fig. 16 The core geometry

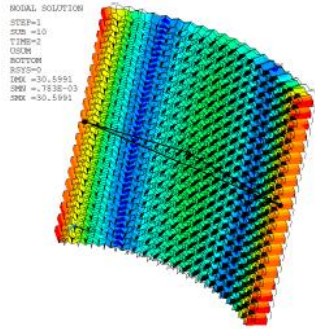


Fig. 17 Deformation on X direction calculus

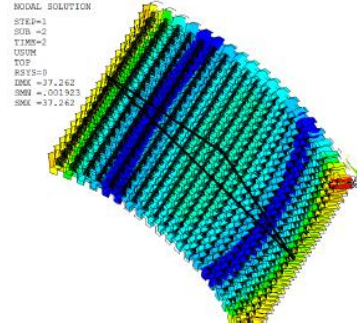


Fig. 18 Deformation on Y direction calculus

Different finite element analysis were performed for other two sandwich panels presented below in figures 19 to 22 for life cycle analysis and an comparative analysis between these two models.

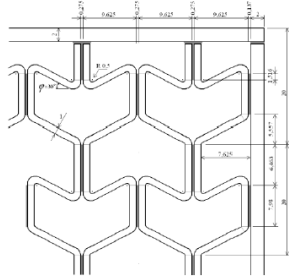


Fig. 19 SRE sandwich panel core - detail

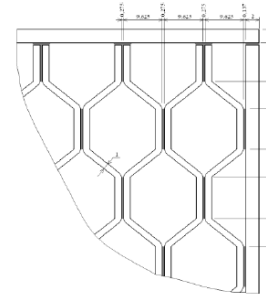


Fig. 20 Honeycomb sandwich core - detail

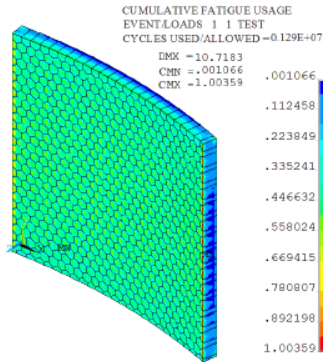


Fig. 21 Accumulated damage inside the faces of the sandwich panel after 1,290,000 pressurized-depressurization cycles

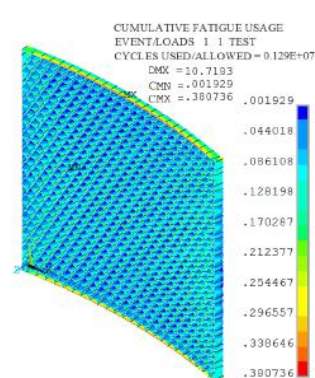


Fig. 22 Accumulated damage inside the core of the sandwich panel after 1,290,000 pressurized-depressurization cycles

Stage no. 4 consists in: finite element analysis of PSC 2a panel; design of experimental model (MEP2) and evaluation through finite element analysis of mechanical performances of this panel. The PSC 2a panel is modeled with finite elements, parameterized for possibility to change the thickness variables of different components of the panel for a sensitivity study of the thicknesses of the rigid PVC tubes wall, as well as determining the fatigue life of the panel for a convenient set of dimensional parameters to pulsing cycles.

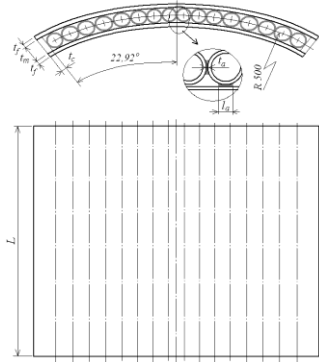


Fig.23 The geometry of the cylindrical panel $L=400$ mm, $t_a=0.2$ mm, $l_a=5$ mm

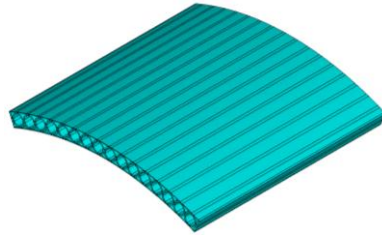


Fig.24 3D model of the panel

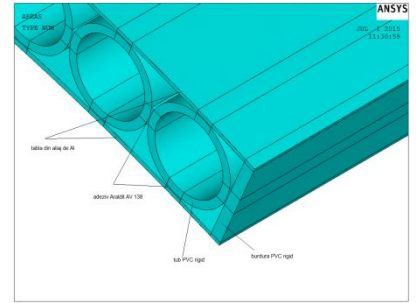


Fig.25 Detail of cylindrical panel

For the finite element model was used the isoparametric element SOLID45 with 8 nodes for structural solid model simulation with three degree of freedom per node: 3 translation after x, y and z directions.

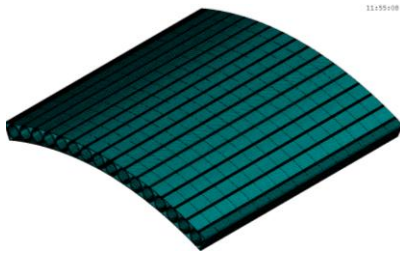


Fig.26 Finite element model of the cylindrical panel

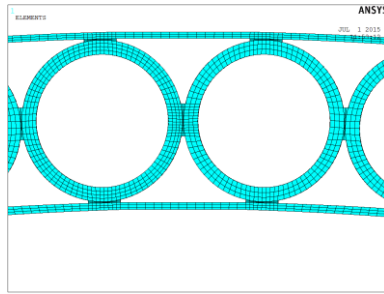


Fig.27 Finite element model of the cylindrical panel - detail

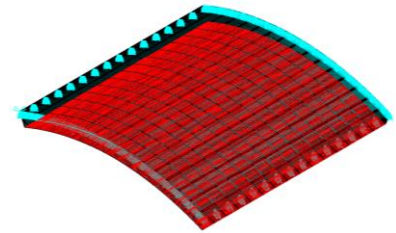


Fig.28 Boundary conditions (displacement and pressure)

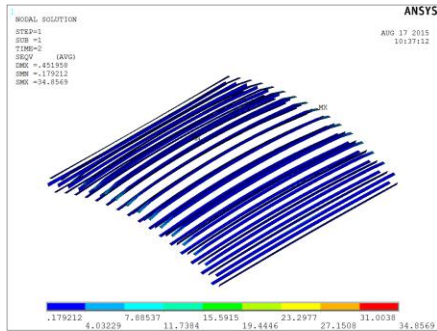


Fig.29 Equivalent tension map for adhesive

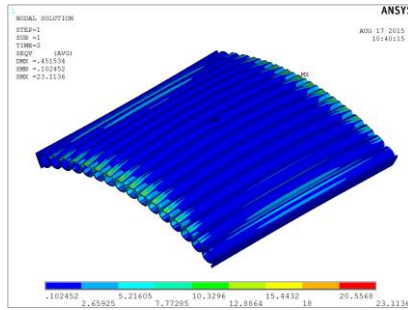


Fig.30 Equivalent tension map for the core of the panel (rigid PVC tubes)

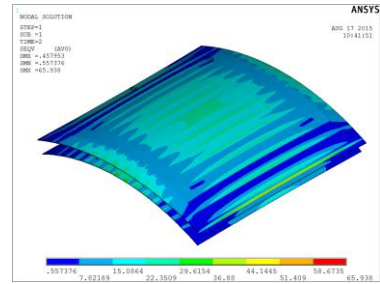


Fig.31 Equivalent tension map for the faces of the panel (Al alloy)

The results obtained for fatigue tests for PSC 2a cylindrical panel are presented below in figures 32 to 38.

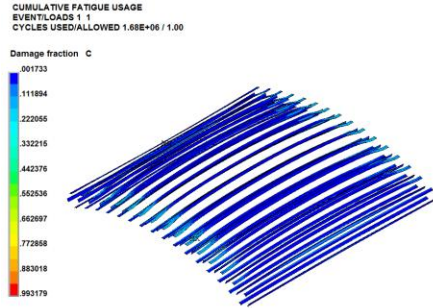


Fig.32 Deterioration ratio for 1.680.000 cycles - adhesive layer

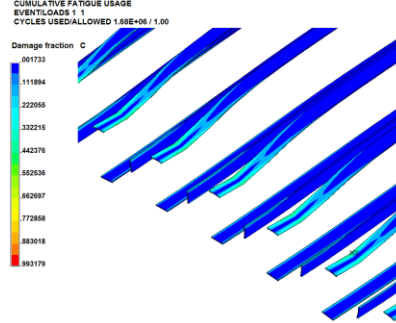


Fig.33 Deterioration ratio for 1.680.000 cycles - adhesive layer - detail

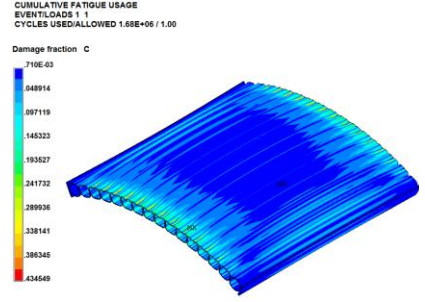


Fig.34 Deterioration ratio for 1.680.000 cycles - rigid PVC tubes

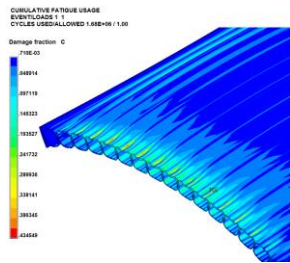


Fig.35 Deterioration ratio for 1.680.000 cycles - rigid PVC tubes - detail

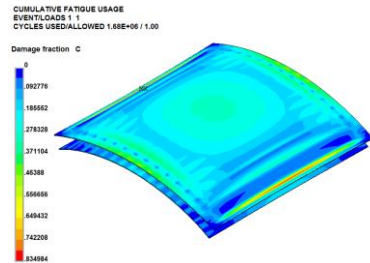


Fig.36 Deterioration ratio for 1.680.000 cycles - duralumin faces

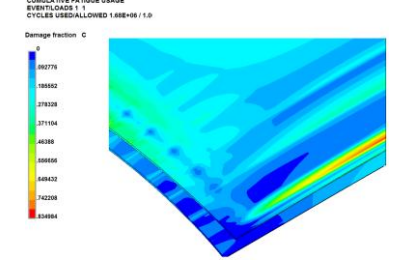


Fig.37 Deterioration ratio for 1.680.000 cycles - duralumin faces - detail

Design of the MEP 2 experimental model and finite element analysis for performances evaluation and obtained results are presented in the following figures:

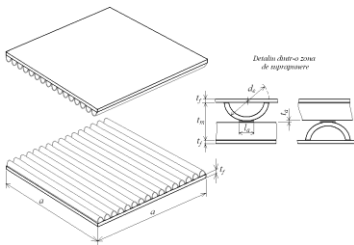


Fig.38 Preliminary sketch of the experimental MEP 2 model

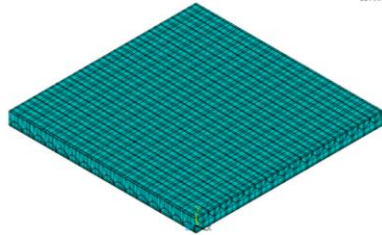


Fig.39 3D Model of the panel

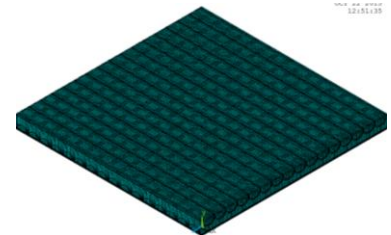


Fig.40 FEM model of the panel

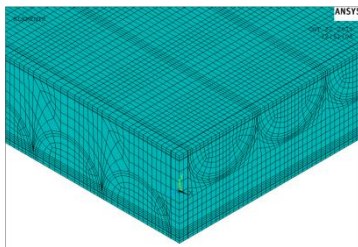


Fig.41 FEM model of the panel - detail

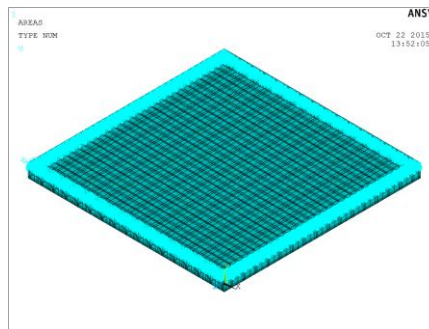


Fig.42 Boundary conditions on edges

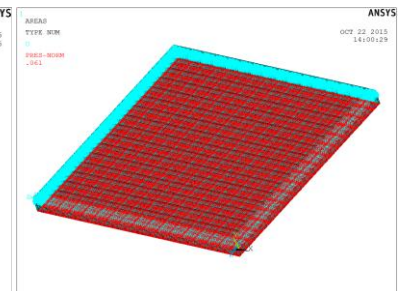


Fig.43 Boundary conditions on edges and pressure

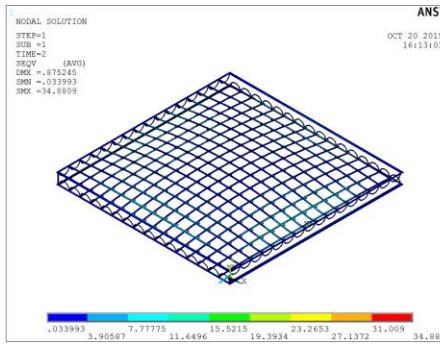


Fig.44 Equivalent tension map for adhesive

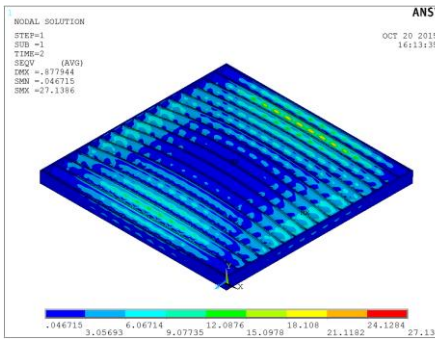


Fig.45 Equivalent tension map for the core of the panel (rigid PVC tubes)

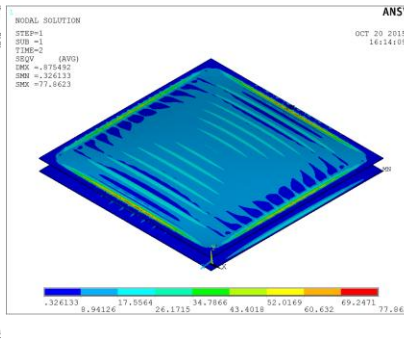


Fig.46 Equivalent tension map for the faces of the panel (Al alloy)