

H.M.Silva, C.D.Carvalho, N.R.Peixinho

**ELASTO-PLASTIC BEHAVIOR OF ALUMINUM FOAMS SUBJECTED TO COMPRESSION LOADING**

*Department of Mechanical Engineering, University of Minho, Campus of Azurém, 4800-058 Guimarães, Portugal; email:hugolopesilva@gmail.com*

**Abstract.** The non-linear behavior of uniform-size cellular foams made of aluminum is investigated when subjected to compressive loads while comparing numerical results obtained in the Finite Element Method software (FEM) ANSYS workbench and ANSYS Mechanical APDL (ANSYS Parametric Design Language). The numerical model is built on AUTODESK INVENTOR, being imported into ANSYS and solved by the Newton – Raphson iterative method. The most similar conditions were used in ANSYS mechanical and ANSYS workbench, as possible. The obtained numerical results and the differences between the two programs are presented and discussed.

**Key words:** plasticity, finite element method, cellular material, non-linear analysis, aluminium foam.

**1. Introduction.**

Engineers make efforts to reduce the pores from casted parts, welded joints, or coatings, because there is a common idea that pores always act as a defect. With this common thinking, it is hard to accept that metallic structures that include pores can be quite effective for structural applications. Cellular structures are structures that have pores. Some structures don't have a regular pattern (Eg. Foams), but some do have a regular pattern (Eg. Honeycombs, Hollow Sphere agglomerates). Cellular materials have advantages, such as the high specific strength, and the associated reduction of cost that is inherent to its use. In fact, and as it is logical, a cellular material has a fraction of its weight in comparison with the bulk base cell. One can then achieve a reduction of material cost. The mechanical properties and the mechanical behavior of cellular materials depend significantly on the porosity [4]. The influence of geometric variables related to the pores on the mechanical behavior is not fully understood at present. Some types of regular cellular solids, such as spherical inclusions, have already been studied by [8]. The authors studied the influence of parameters related to the pores on the global behavior of those solids. The authors proved that the studied variables are relevant for the mechanical behavior and determined the most relevant variables to be used on optimization processes [8]. Gibson and Ashby have found that, on cellular solids, the main factor governing the mechanical behavior in comparison to the bulk solid is the relative density [5]. Nieh et al. investigated the mechanical properties of open-cell 6101 aluminum foams with different densities (5 – 10%) and morphologies (4 – 16 cells  $\text{cm}^{-1}$ ) subjected to compression loadings. The authors concluded that the relative density is the main variable controlling the modulus and yield strength of foams. The cell size appears to have no significant effect on the strength of foams at a fixed density. However, it is found that the cell shape has effect on the strength of foams [6]. Rakow and Wass [7] investigated size effects on the shear response of aluminium foams having manufactured foam samples and subjected them to static and quasi-static low-rate shear deformation using a loading fixture designed for the produced materials. The authors have found that for the range of densities examined, the shear

modulus shows a linear dependence on relative density, while the ultimate strength follows a non-linear power law. [7] The effect of strut geometry on the yielding behavior of open-cell foams has been studied by Xie and Chan. The authors performed a micromechanical analysis with the aim to investigate the effect of strut geometry on the Yielding behavior of open-cell foams. The authors tested different cross-sections: rectangular, circular, and equilateral triangular shapes. The authors found that the shape of the plastic-yield surface is dependent on the relative density and the cross-sectional shape of the structure. The authors also concluded, by means of numerical results, that open-cell foams with asymmetrical sectional struts have different tensile and compressive collapse strength [9]. Andrews et. Al [1] tested the compressive and tensile behavior of Al foams having compared the uniaxial compressive modulus, tensile modulus and strength of several aluminum foams with models for cellular solids. The authors have found that the foam experimental results regarding open cell have good correlation with the model. The closed cell foams have modulus and strengths that are quite lower than the models predicted values. The authors concluded that the discrepancy is due to micro defects which cause bending instead of stretching of the cell walls [1]. The effects of cell size on compressive properties of aluminum foams were studied by Cao et al [2]. The authors studied the effects of cell size on the quasi-static and dynamic compressive properties of open cell aluminum foams. The authors found that the elastic moduli and compressive strengths of the studied aluminum foam are not only dependent on the relative density but also dependent on the cell size of the foam under both quasi-static loading and dynamic loading [2].

## 2. Numerical procedure.

A cellular solid, shown in the fig. 1 was tested in compression. A metallic foam of uniform cell size is characterized by two parameters: the distance between two adjacent hole centers, with the value of 1,38 mm, and by the radius of the larger holes, with the value of 7,5 mm [3]. The cellular solid has a regular pattern and has a wall around it having a thickness of 0,4 mm. The outer diameter of the model is 16,36 mm. The CAD model for the studied geometry with wall and the real geometry are presented in the fig. 1.

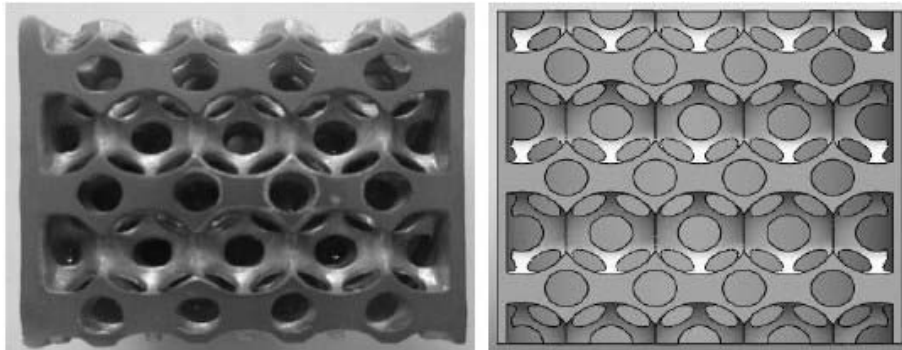


Fig. 1. Studied geometry: real (left) and CAD model (right) [3].

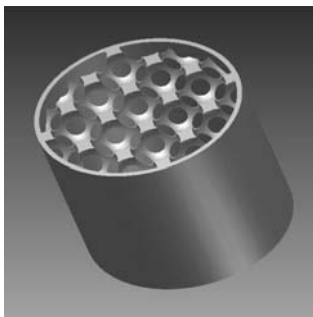


Fig. 2. Geometry of the cellular solid, taken from Autodesk Inventor [3].

In this work, the model was solved used the Newton – Raphson iterative method. On the «Newton – Raphson» method, the load is subdivided into a series of load increments. The load increments can be applied over several load steps. Each load step can have several substeps. The load is incremented on each substep. The aim is to obtain convergence. Convergence may be obtained, when the solution satisfies the convergence norm, which has several parameters, being the tolerance an important parameter for convergence. The lower the tolerance, the more accurate the solution is, but usually more iterations are needed in order to achieve a converged solution. The chosen element in ANSYS APDL is Solid Tetragonal with 4 nodes, known as

SOLID285. Both cylinders and cellular solid are meshed with the same parameters. In ANSYS workbench, the solid elements used are SOLID186 for the foam and walls and SOLID187 for the cylinders. Other elements were used in ANSYS Workbench, such as the contact element CONTA174. In order to model contact, the element TARGE170 was used to assign the contact surfaces. The element SURF154 was also applied to allow related surface effects. It is applied a distributed compressive load on the top area of the top cylinder, and DOF constraints on the bottom area of the bottom cylinder. All 3 translation DOF and 3 rotation DOF were constrained. The applied pressure is:1,17 MPa. The FEM model is composed by a cellular solid and by two cylinders. The cylinders have 50 mm diameter and 10 mm height, and are located at the bottom and top of the model. Fig. 3 shows the mesh of the FEM model. The cellular solid is represented in red, while the cylinders are shown in green. The mesh is a tetragonal free mesh, and has an average element length of 0,0012 m.

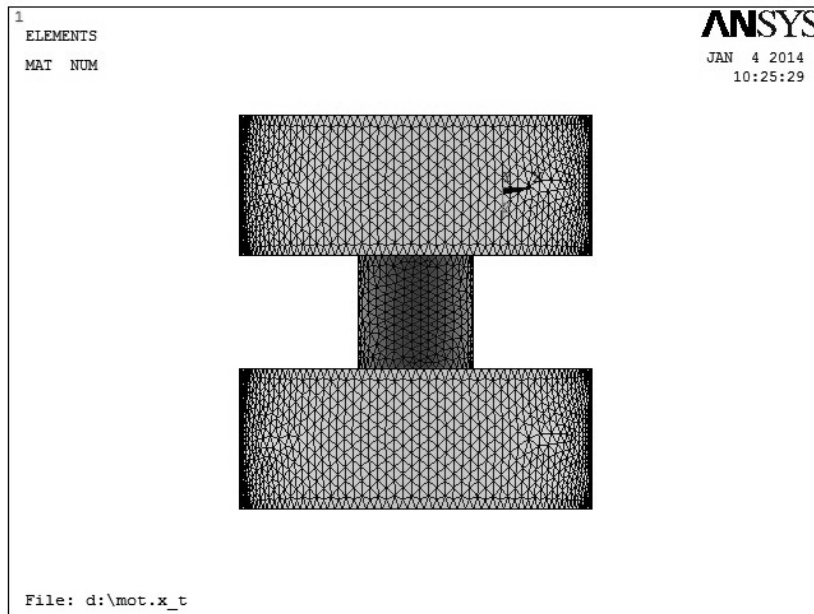


Fig. 3. Cellular solid between two cylinders

The used material for the foam is pure Aluminum. Steel was only used for the cylinders, in order to ensure the less influence possible of the cylindrical plates on the behavior of the cellular solid. The Young's modulus of the aluminum was determined by tensile test. Table 1 presents the aluminum and steel properties used on the ANSYS analysis. The material is assumed to be isotropic:

Table 1. Elastic properties of steel and Aluminum used in the FEM simulation.

Table 1

	Aluminum	Steel
Young Modulus	$57 \cdot 10^9$ [Pa]	$210 \cdot 10^9$ [Pa]
Poisson coefficient	0,33	0,29
Density	2700 [kg / m <sup>3</sup> ]	7800 [kg / m <sup>3</sup> ]

A multilinear isotropic hardening model was used, using Mises model, in order to simulate plasticity behavior of the cellular solid. The plasticity model was used only for aluminum. Figure 4 presents the stress-strain curve used in the software, which includes plasticity behaviour.

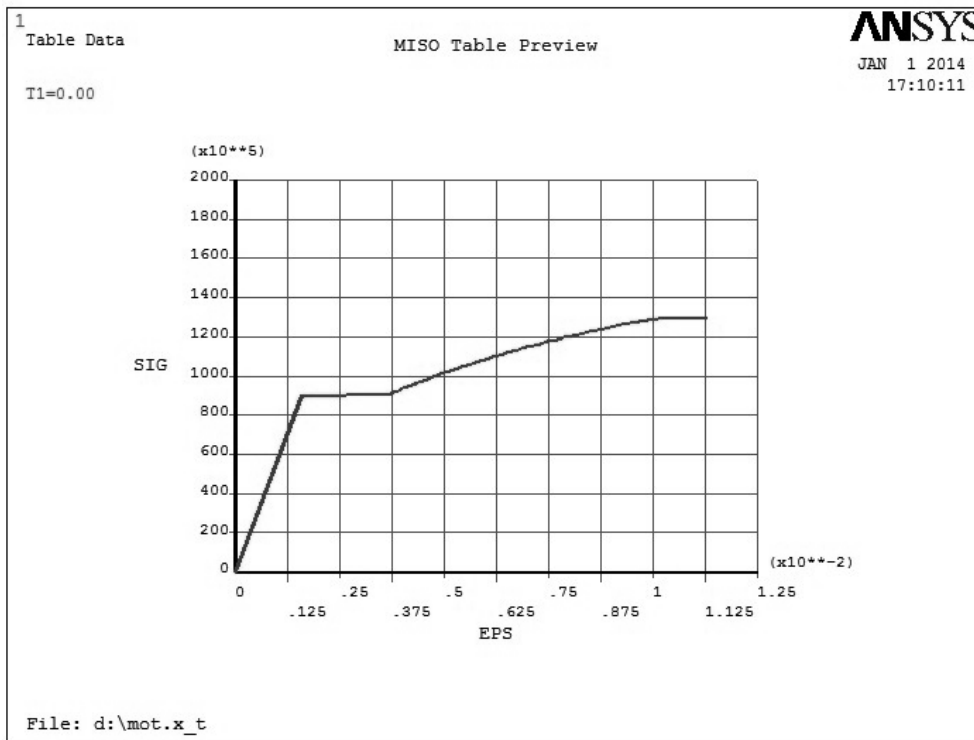


Fig. 4. Stress-Strain curve used for nonlinear material modeling.

Solution converges on every substep. The convergence criterion force based with convergence settings as presented in table 2:

Table 2. Force convergence criterion settings

Table 2

Parameter	Value
Reference value	Calculated
Tolerance	$1 \cdot 10^{-3}$
Norm	L2
Minimum reference value	$1 \cdot 10^{-5}$

The solution parameters are shown in the table 3:

Table 3. Nonlinear solution parameters

Table 3

Analysis type	Large displacement Static
Solution Method	Newton-Raphson
Number of load steps	1
Number of substeps	50
Automatic step time	OFF
Time at the end of the loadstep	1

### 3. Results.

Figure 5 shows the absolute convergence norm of the force (red), and the criterion (green) on all iterations, as presented by ANSYS APDL.

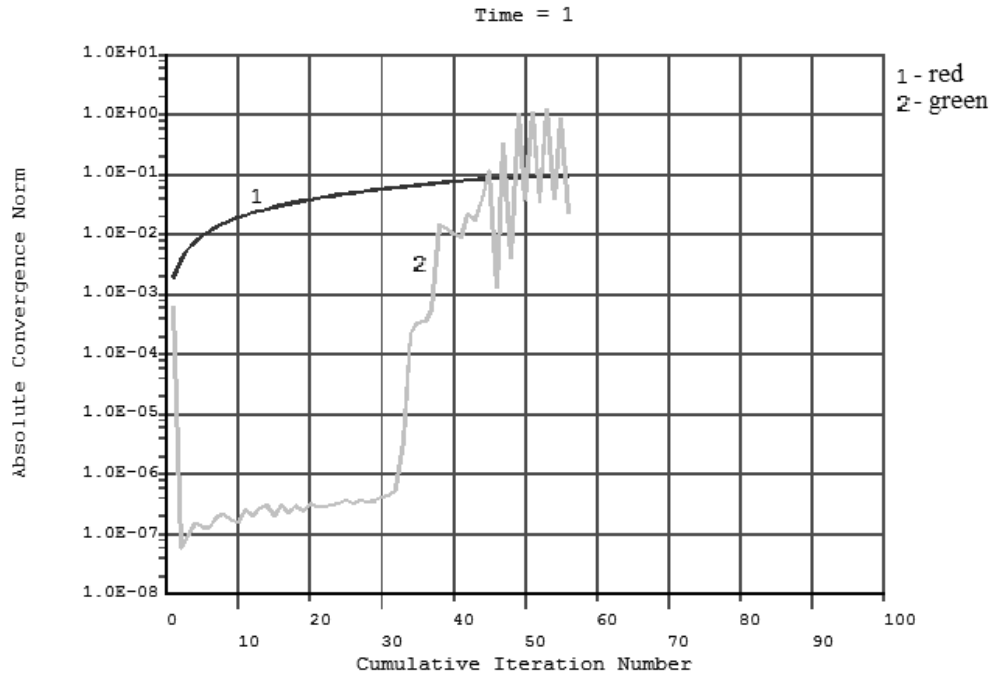


Fig. 5. Absolute convergence norm value on each iteration on ANSYS mechanical APDL.

Based on the fig. 5, one can say that the convergence criterion increased during the analysis and that convergence was observed on every substep. The program was able to ob-

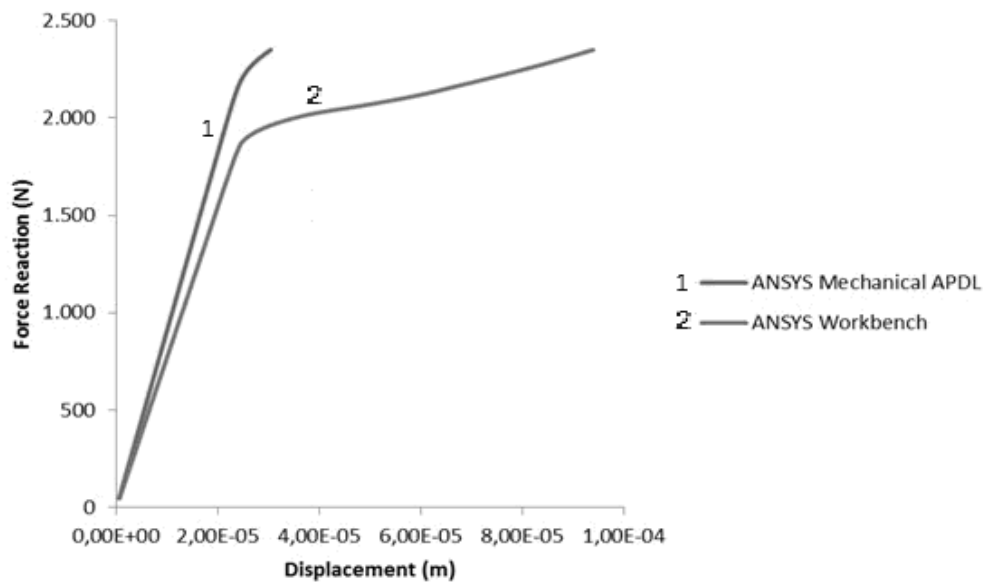


Fig. 6. Force reaction-Displacement curve for ANSYS workbench and ANSYS mechanical

tain a solution quite easily, given the simulation conditions. On 50 substeps, only 56 iterations were performed. The mechanical behavior of these geometries has been already studied [3], being presented in fig. 6 the Force-Displacement curve obtained for the same geometry and conditions with ANSYS workbench [3]. A simulation, using similar conditions was done on ANSYS Mechanical APDL in the present work. The maximum Z displacement result  $U_Z$  was taken from the top area, where the load is applied, and the sum of all FZ reaction forces were taken from the bottom area, where the model is constrained. The maximum absolute value of the displacement  $U_Z$  is plotted with the total force reaction in the Z direction, which is the sum of the force reaction values for all nodes. The chart of the fig. 6 shows the displacement-force reaction for every one of the 50 substeps.

The chart contains 50 points, being each point the result of the last iteration of each substep, when the solution converges. One can see that there are differences between the curves, mainly the slope of the elastic domain, which is higher for ANSYS mechanical and the strain energy which is clearly higher on ANSYS workbench.

The stiffness  $K$  can be defined from the linear relation between load ( $P$ ) and displacement ( $\delta$ ) within the elastic region:  $K = P/\delta$ . Another measure of the stiffness is the stiffness per mass unit ( $K_m$ ), which is given by:  $K_m = K/m$ , where  $m$  is the mass. In graphical terms, the stiffness can be determined by the slope of the elastic part of the Force-Displacement chart, which is a straight line. The Fig. 7 shows the graphical determination of stiffness for the obtained results. The corresponding values of stiffness and mass are presented in the table 4.

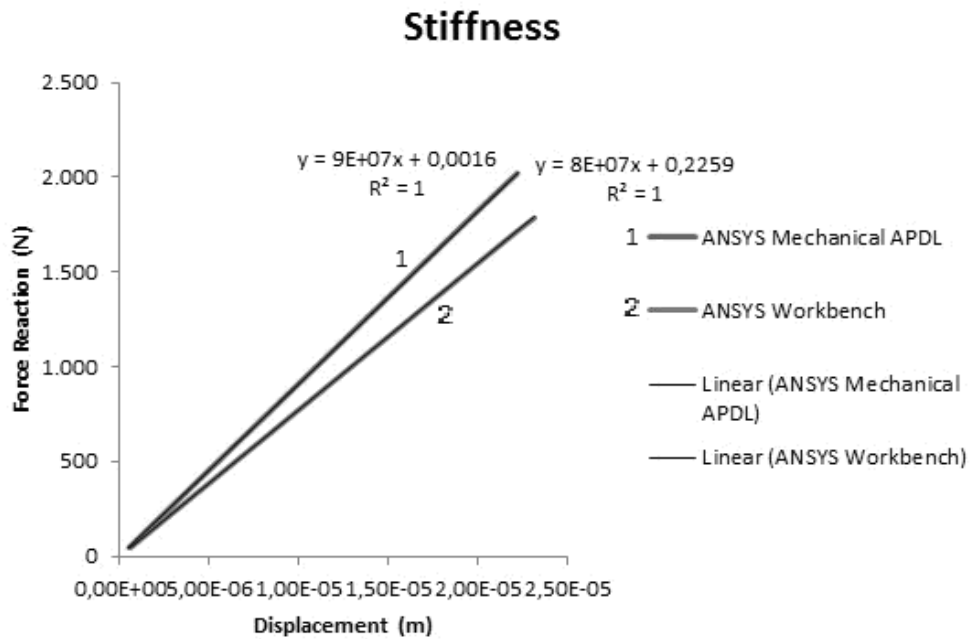


Fig. 7. Determination of the stiffness.

Table 4. Values of the stiffness and of the stiffness per mass unit.

Table 4

	Stiffness (N/m)	Mass (kg)	Specific Stiffness(N/m*kg)
Workbench	$8 \cdot 10^7$	$1,64 \cdot 10^{-3}$	$4,88 \cdot 10^{10}$
Mechanical	$9 \cdot 10^7$	$1,83 \cdot 10^{-3}$	$4,92 \cdot 10^{10}$

#### 4. Discussion of results.

From the analysis of load-displacement curves one can observe that there is a non-linear relation between force reaction and displacement after reaching yield point, therefore the applied load was high enough for the cellular solid to go beyond yield. One can then conclude that the simulation conditions, namely, mesh parameters, load intensity, and element type were reasonable and adequate for the studied problem. The converged force value strictly increases in absolute value with the displacement, as shown in the fig. 5 which concurs that the higher the displacement, the higher the load needed for the solution to converge. The Force-displacement curve has a typical behavior, having a linear part, for the lower displacements, and a non-linear part for the post-yielding (plastic) zone. The slope of the elastic part of the curve is higher in ANSYS workbench than in ANSYS mechanical. This can be explained by the fact that in ANSYS workbench it is possible to disable non-linear effects on the cylinders and other options that make the displacement on the cylinders equal to 0. This is not possible in ANSYS mechanical. It was made an effort to emulate as much as possible the simulation conditions on both ANSYS programs. Although the curves presented in figure 7 show differences in terms of slope:  $8 \cdot 10^{10}$  to  $9 \cdot 10^{10}$ , which are the values of the stiffness, the values of stiffness per mass unit are quite similar. The differences obtained in the stiffness values can be due to differences in internal recognition of the mass in ANSYS workbench and ANSYS mechanical APDL. In fact, and as the geometry was imported from Inventor to ANSYS mechanical, some details are hard to reproduce, such as the minimum mesh size, and the elements used in ANSYS workbench did not work well on ANSYS mechanical. It is found however, that the mesh used in ANSYS APDL was fine enough to obtain converged results. The difference between the definition of contacts, i.e., the contacts between the steel cylinders and the foam were defined on ANSYS workbench, while they were not defined in ANSYS mechanical APDL. On ANSYS workbench, the contact element CONTA174 was used. TARGE170 was also used to assign the contact surfaces to the associated contact element. The surface element SURF154 was used to allow associated surfact effects. The elements used: SOLID 186 and SOLID 187 in ANSYS workbench lead to difficulties in meshing in ANSYS APDL. As such, SOLID285 elements were used. Also quadrilateral mesh was used in ANSYS workbench for the steel cylinders, and due to that impossibility in ANSYS mechanical, tetragonal mesh was used. The difference between the elements and mesh is the main explanation for the observed differences in numerical results.

#### 5. Conclusions.

It is possible to conclude that it was easy to achieve convergence for this particular problem. This may suggest that the simulation conditions were adequate. The load was high enough to pass yield point and enter the plastic domain. Some differences in software options are present between the two programs used which limits the aim of reproducing exact simulation conditions, despite the best efforts. The most significant differences were associated to element type and mesh details. Also the internal recognition of geometry from CAD import might explain some result variation. Despite that fact, a reasonable agreement was established for the modeling conditions, and consequently, also for the stiffness values.

**РЕЗЮМЕ.** Досліджено нелінійну поведінку однорідних кліткоподібних пін, зроблених з алюмінію і підданих стиску. При цьому порівнюються числові результати, отримані двома програмними пакетами: Finite Element Method software (FEM) ANSYS workbench і ANSYS Mechanical APDL (ANSYS Parametric Design Language). Числова модель побудована в AUTODESK INVENTOR і далі імпортується в ANSYS і розв'язується методом Ньютона – Рафсона. Використано якомога ближчі умови у вказаних пакетах. Представлено і прокоментовано отримані числові результати і спостережені відмінності між двома пакетами.

1. *Andrews E., Sanders W., Gibson L.J.* Compressive and tensile behaviour of aluminum foams // *Materials Science and Engineering*. – 1999. – A270. – P. 113 – 124.
2. *Cao X.Q., Wang Z.H., Ma H.W., Zhao L.M., Yang G.T.* Effects of cell size on compressive properties of aluminum foam // *Trans. Nonferrous Met. SOC. China*. – 2006. – **16**. – P. 351 – 356.
3. *Carvalho C.D.* Simulação numérica e verificação experimental do comportamento elástoplástico de espumas metálicas, MSc. Thesis, University of Minho (2012) [In Portuguese].
4. *Degischer H.P., Kriszt B.* Handbook of Cellular Metals: Production, Processing, Applications, Wiley-VCH Verlag GmbH & Co. KGaA (2002).
5. *Gibson L.J., Ashby M.F.* Cellular Solids. Structure and Properties. 2<sup>nd</sup> edition. – Cam.: Cambridge University Press, 1997. – 532p.
6. *Nieh T.G., Higashi K., Wadsworth J.* Effect of cell morphology on the compressive properties of open-cell aluminum foams // *Materials Science and Engineering*. – 2000. – A283. – P. 105 – 110.
7. *Rakow J.F., Waas A.M.* Size effects and the shear response of aluminum foam // *Mechanics of Materials*. – 2005. – **37**. – P. 69 – 82.
8. *Silva H.M., Peixinho N.R., Silva F.S.* Numerical study on the sensitivity analysis for strength and stiffness of regular cellular solids // *Int. J. of Applied Research in Mechanical Engineering*. – 2013. – **3**, № 1. – P. 93 – 98.
9. *Xie L.S., Chan K.C.* The effect of strut geometry on the yielding behavior of open-cell foams // *Int. J. of Mech. Sci.* – 2006. – **48**. – P. 249 – 255.

---

From the Editorial Board: The article corresponds completely to submitted manuscript.

Поступила 29.04.2015

Утверждена в печать 14.03.2017