8th International Conference on Porous Metals and Metallic Foams, Metfoam 2013

Numerical modeling and simulation of uniaxial compression of aluminum foams using FEM and 3D-CT images


8th International Conference on Porous Metals and Metallic Foams, Metfoam 2013

Abstract

The aim of this paper is to evaluate the elasto-plastic deformation behavior of open cell aluminum foams samples simulated under compressive loads and verified experimentally. Aluminum foams were fabricated from AlSiMg alloy by infiltration process with vacuum pressure. The samples geometry was obtained via 3D reconstruction using micro-computer tomography images. Solid representation of the samples was used for the numerical model based on Finite Element Method (FEM) where mechanical behavior was established. The material parameters were calibrated by experimental results and were used for setting up simulations. The compression test was carried out at room temperature and strain rate of 0.5 mm.s⁻¹. The average plateau stress and energy absorption per unit volume were evaluated from the measured stress-strain curves. The 3D FEM models were used to simulate the mechanical behavior of open cell aluminum foam and establish its stress-strain state. The experimental responses from the compression tests were in agreement with the obtained from the 3D numerical models. It was possible to determine by numerical models and experimental stress-strain curves, the mechanical behavior of the aluminum foams under study: the plateau stress and energy absorption increase with decreasing pore size and increasing density.

© 2014 Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/3.0/).

Peer-review under responsibility of Scientific Committee of North Carolina State University

Keywords: Aluminum foams; FEM; Compression test; Modeling; CT images.

* Corresponding author. Tel.: +574-448-8388; fax: +574-448-8388 ext.10901.
E-mail address: patricia.fernandez@upb.edu.co

2211-8128 © 2014 Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/3.0/).

Peer-review under responsibility of Scientific Committee of North Carolina State University
doi:10.1016/j.mspro.2014.07.609
1. Introduction

Cellular metals have recently attracted considerable attention due to their excellent combination of mechanical and physical properties together their exceptional low densities. Metal foams are special materials that exhibit unusual properties with a great potential for a wide variety of applications, such as energy absorption appliances, core materials for sandwich structures, ultra-lightweight structural components (Banhart (2001), Ashby et al. (2000)).

Evaluate the elasto-plastic deformation behavior of metal foams is not a new thing (Gibson and Ashby (1997), Cao et al. (2006), Andrews et al. (1999), Gibson (2000), Kitazono et al. (2003)), and using numerical models based on Finite Element Method (FEM) (Youssef et al. (2005), Maguid et al. (2002), Kim et al. (2006)), either. In general, each cellular metal structure is different in function of the manufacture process, and in this sense, properties data from this kind of materials are always different and not well understood. The aim of this work is to evaluate the elasto-plastic deformation behavior using simulations under compressive loads and verified experimentally, of open cell aluminum foams fabricated from AlSiMg alloy by infiltration process. In this case, the samples geometry was obtained by 3D reconstruction images using micro-computer tomography.

2. Materials and methods

2.1. Sample Preparation

The aluminum foams have been obtained using a preform of salt particles sieved in three sizes: 0.5 mm, 1.2 mm and 2.0 mm. The manufacture method employed is named infiltration process, and to infiltrate the NaCl preform was used melted AlSiMg aluminum alloy (Fernández (2010)). Finally, the preform was removed with water to obtain the open pore structure. Then, three samples for each pore size (0.5, 1.0 and 2.0 mm) with cylindrical geometries were cut. The porosity was determined using a pycnometer device, obtaining around to ~65% porosity in average.

2.2. Compression tests

The specimens for the compression tests were 20 mm in diameter and 20 mm in height. Uniaxial compression tests of the specimens have been performed on Instron 5582 testing universal machine at a cross-head speed of 0.5 mm.s⁻¹. It was used three samples for each pore size.

2.3. Geometric Reconstruction

Microtomografic (μCT) images for each sample were taken using magnification, which is especially useful when there are small details being inspected ((Banhart (2001)). The samples topology was obtained via 3D reconstruction. A set of 25 μCT images from the middle of each sample was taken. That zone can be considered as representative for the mechanical behavior analysis for each sample, because the edge effects and distortions that may be in the areas subjected to the cutting process are avoided. A set of microtomographic images are necessary for 3D geometric reconstruction. Due to the size of the sample and the pores it was necessary to choose a partial component from the μCT study, which is represented inside the rectangle as it is showed in Fig. 1. Partial reconstruction has 4mmx4mmx2mm.

Using a trial version of MIMICS a reconstruction process generates a 3D geometry, which in that point represents a solid (Fig. 2), but this solid is not useful for numerical modeling by finite element method (FEM) due to the small edges at solid reconstruction.

The reconstruction process under MIMICS consists on defining a density threshold, which chooses all the pixels that are included in that range defined by the threshold. Additionally, a manual complementary edition process is required to improve the quality of the solid. In this process, specific parts that can be classified as islands are deleted or integrated to the rest of the solid to obtain a continuous part. Once the solid has been finished, using 3-Matic Materialise, which is a complement of MIMICS, a planar mesh that represents the exterior part of the solid can be generated (Fig. 3), but the initial meshing results are too low in quality then an improving or remeshing process is
needed and applied. That process generates a high quality superficial mesh that can be used for numerical models based on FEM.

Importing in ABAQUS 6.12-3, the planar mesh from 3-Matic and using edit mesh tools the superficial mesh is converted into 3D mesh that can be used for the numerical model. In a numerical model related to the FEM and specifically in models like this one where the element size, required or imposed by the geometry, is pretty low, it is very important to verify the mesh quality. The element size of each foam can be divided in two types, big elements and small elements; the average element size for each type is 0.1473mm and 0.0400mm (pore size 0.5mm), 0.1773mm and 0.0370mm (pore size 1.0mm) and 0.2229mm and 0.0323mm (pore size 2.0mm). Using the diagnosis tool in ABAQUS 6.12-3 the mesh warnings were identified and solved with the edit mesh tool.

2.4. Material Properties and Boundary Conditions

The material properties were assigned according to experimental data for base material used for foam fabrication. A linear elastic and perfect plastic behavior were assigned using a Young’s Modulus of E=68GPa and Poisson ratio of $\nu=0.3$. For perfect plastic part the yield stress was 250MPa and 0.0 plastic strain.

Using the 3D geometrical representation of the foam sample, a complete numerical model was developed in order to represent a compressive test. The geometry associated to the foam was complemented by the use of two rigid plates used as a base and compressive component.

General contact option in ABAQUS 6.12-3 was used, applying hard contact in normal direction and frictionless in tangential direction. Those contact conditions avoid the foam penetration into the rigid plates and vice versa. A displacement restriction over all degrees of freedom was applied in the lower free surface of one rigid plate. Finally, a boundary condition of displacement was applied over the other plate, and its value was equivalent to 50% of the foam thickness. The complex geometry of the foam requires the use of 2.185.151 C3D4 tetrahedral elements, which represents a high computational cost.

3. Results

3.1. Compressive tests results

The Al foams under study have a very homogeneous behavior in the stress-strain curve for each pore size samples. The curves are characterized by an initial elastic region, followed by a deformation plateau region with a nearly constant flow stress until around to 50 - 60% strain, and finally a transition where the flow stress rapidly increased because of sample densification.
It was observed that the stress at the macroscopic yield point and the subsequent plateau stress at a certain strain, increase with increasing the relative density.

Moreover, it should be noted that the foams with the smaller pore size of 0.5 mm showed a higher flow stress than that with the greater pore size of 2.0 mm. Fig. 4 shows the experimental stress-strain curves obtained.

3.2. Numerical modeling and simulation results

The boundary condition of displacement was applied gradually, which implies a progressive contact between the plates and the metallic foam. Once an actual contact area of the foam is under the action of the rigid plate more homogenous stress – strain state is present at the foam. In Fig. 5, it is possible to identify that due to the deformation; the foam pores are closing continuously, which is according to experimental procedure and actual behavior.

Making a specific query over a set conformed by those nodes which are in contact to the rigid plates at the beginning of the model the reactive forces and displacements in compression direction can be extracted for the total time of the model. Using those data, combined with the apparent area of the foam sample and its height, the stress-strain curve was established. This curve and the experimental can be seen at Fig. 6.

In Fig. 6 it is possible to identify that the ABAQUS curve has the same behavior that the experimental has, this means that the boundary conditions, material properties, and geometry foam were replicated successful, and it is possible to simulate and to obtain the same stress – strain curve to characterize the foam without destroy the physical foam. However, it is seen that the numerical model based on the FEM did not finish all the deformation process because many elements were distorted, which caused a numerical error.

To reach the end of the deformation process it will be necessary to increase the computational capability, which allows refining the mesh and avoiding elements with low aspect ratio. However the numerical and experimental results matches, a convergence process is recommended, for ensuring the numerical results.
4. Conclusions

It was found that these aluminum foams showed a good mechanical behavior and the stress-strain relation was in agreement with others cellular Al.

The numerical model based on FEM seems to be appropriated for modeling foams. According to the results, the mechanical response represented by stress-strain state in numerical model and experimental data matches, not only on the curve pattern but also on its values.

Acknowledgements

The present authors wish to thank to CIDI-UPB for the financial support provided by Researching Project Rad. 910A-01/12-18.

References