Materials Science & Engineering A 599 (2014) 125–133

Contents lists available at ScienceDirect

Materials Science & Engineering A

journal homepage: www.elsevier.com/locate/msea

Finite element analysis of indentation of aluminium foam and sandwich panels with aluminium foam core

A. Xu [a,b,*], T. Vodenitcharova [a], K. Kabir [a], E.A. Flores-Johnson [c], M. Hoffman [a]

[a] School of Materials Science and Engineering, The University of New South Wales, Sydney, NSW 2052, Australia
[b] Department of Materials, University of Oxford, Parks Road, Oxford OX1 3PH, UK
[c] School of Civil Engineering, The University of Sydney, Sydney, NSW 2006, Australia

ARTICLE INFO

Article history:
Received 24 July 2013
Received in revised form
27 January 2014
Accepted 27 January 2014
Available online 1 February 2014

Keywords:
Aluminium foam
Indentation
Sandwich panels
Finite element analysis

ABSTRACT

This paper presents modelling techniques for achieving convergent and accurate solutions in simulating quasi-static indentation of closed-cell aluminium foams and sandwich panels with aluminium foam core using the commercial finite-element software ABAQUS/Standard. Various indenters are utilised: hemispherical indenters of 10, 15 and 20 mm diameter, a 16-mm cylindrical flat punch, and a long flat punch. The material parameters are established by comparison with the previous experimental results in uniaxial compression. Experimentally-observed tearing of the cell walls is accounted for in the simulations by introducing element failure and deletion criteria. Stress and strain analysis of the results reveals that foam failure occurs where cell tearing takes place. This occurs at critical tear energy and shear strength, independent of indenter size. The resulting load–displacement curves correlate closely with experimental results. The quasi-static indentation of sandwich panels of aluminium foam cladded with aluminium skins is also simulated for hemispherical indenters of diameter 5–20 mm. Cell tearing does not occur until the point of failure and thus, is not accounted for in the simulations. The effect of the skin strength on the stiffness, strength and energy absorption of the panels is elucidated. A particular emphasis is placed on the material formulation for the foam core which would ensure convergence and reliable predictions.

Crown Copyright © 2014 Published by Elsevier B.V. All rights reserved.

1. Introduction

Aluminium foam is a relatively new material with many desirable properties. It is light and capable of absorbing very large deformations [1], making it suitable for energy absorbing applications involving high compression loading (e.g. car chassis and body panels). Because aluminium foam exhibits both metallic and foam-like properties, it has unique deformation behaviour. In compression, there are three stages of deformation (with increasing compressive strain): quasi-elastic, yield plateau and densification; the second stage being the main contributor to the large energy absorption capacity. However, aluminium foams exhibit weaker strength in tension than in compression and undergo irreversible volumetric deformation, thus the von Mises failure criterion is visualised as an ellipsoid [2]. Deshpande and Fleck [2] developed the fundamental theory for predicting metal foam behaviour which forms the basis of the *CRUSHABLE FOAM and *CRUSHABLE FOAM HARDENING material model in the finite element (FE) code ABAQUS (ABAQUS v6.9 User Manual).

The cell walls within the aluminium foam deform by a combination of bending/buckling or tearing [2,3]. The latter is dominant in areas where high shear stresses are experienced. Cell wall tearing is undesirable because it limits deformation to a localised area and produces a smaller hardening response [4]. Cladding aluminium foam with metal sheets to form sandwich panels minimises the effect of cell tearing because the face sheet provides an extra shear strength and distributes the load over a greater area [5].

ALPORAS aluminium foam has been a desired candidate material for metal foam characterisation. It has a small cell size of 3.5–4.5 mm and a relatively homogeneous cellular structure [6,7]. Thus, consistent results can be achieved on wide range of indenter sizes. It was found that in indentation of ALPORAS foam with flat end punches the cell walls at the periphery of the indenter tear while the cells beneath the indenter crush in a manner similar to uniaxial compression; the tearing energy has been reported to be 7.45 kJ m^{-2} [8]. In indentation with hemispherical and conical indenters, however, the cells at the blunt faces shear (instead of tear) whereas the cells beneath the indenter crush; the shear strength is reported as 1 MPa but it can be as high as 1.54 MPa due to cell size effects [7,9].

* Corresponding author at: Department of Materials, University of Oxford, Parks Road, Oxford OX1 3PH, UK. Tel.: +44 1865 273 694; fax: +44 1865 273789.
E-mail address: alan.xu@materials.ox.ac.uk (A. Xu).

0921-5093/$ - see front matter Crown Copyright © 2014 Published by Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.msea.2014.01.080

Cladding ALPORAS foam to form a sandwich panel increases the resistance to indentation deformation. The load–displacement behaviour is typically linear and the slope is positively correlated with skin strength and toughness [6]. Foam indentation behaviour generally resumes once the cladding is fractured. The behaviour discussed thus far is concerned solely with quasi-static indentation rather than dynamic indentation trend. Dynamic indentation is beyond the scope of this study although further information can be found in Refs. [10,11].

Finite element (FE) modelling has been a useful tool for validating the experimental trends observed in aluminium and polymeric foams and sandwich panel uniaxial compression, indentation and bending experiments [12–16]. Bart-Smith and Rizov simulated three- and four-point bending of sandwich panels to validate failure modes observed in the experiment [13,16]. The foam uniaxial compression data and the foam loading modulus were used for the material model. Their material model achieved a reasonable level of correspondence with their experimental data at shallow vertical displacement of 1–4 mm. However, at large deformations there was poor correlation with experimental results [13,16].

FE simulations of indentation of ALPORAS foam have been performed using the ABAQUS and LS-DYNA explicit solvers. Mohan and Hansseen et al. simulated flat end indentation to a depth of 50% strain in their ALPORAS sample [17,18]. The simulation results achieved did not match the experimental results primarily because they were using explicit solvers to simulate a quasi-static indentation. The stress propagation in an explicit solver is density dependent and the ALPORAS density increase with deformation was not accounted for in their calculations. Also, it needs to be noted that explicit analysis is only applicable to time dependent problems because the solution is sought in a time domain, and thus is incapable of modelling quasi-static indentation.

This paper explores techniques for simulating quasi-static deformations of ALPORAS foam and sandwich panels in ABAQUS/Standard. Whilst the simulation works of the aforementioned authors provide useful starting points; new techniques are needed to increase the degree of accuracy and convergence. This paper presents a standard method of achieving accurate degree of crushing and tearing in ALPORAS foam and highlights the limitations of the Deshpande–Fleck material model. The findings are verified using the experimental results in the literature [20].

2. Finite element simulations

2.1. Material model and material parameters

The *CRUSHABLE FOAM material model was used for foam with isotropic hardening. The material parameters used were density 0.27 g/cm^3, loading Young's modulus 60 MPa (E_s), plastic Poisson's ratio 0.015 and compression yield stress ratio 1.702 [4,19]. The compression yield stress ratio is calculated using $k=\sqrt{[3(1-2\upsilon)]}$ (ABAQUS Analysis User's Manual, Version 6.9, Simula; 2008). A yield stress of 1.2 MPa and the stress–strain curve to describe the foam hardening were based on uniaxial compression results of Idris et al. [19] as shown in Fig. 1(a) and (b). The solid black line represents the representative experimental result from which a simplified curve was generated (dashed line). The dash line of the stress–strain curve in Fig. 1(b) was input into *CRUSHABLE FOAM HARDENING materials model. The ABAQUS software uses the stress–strain data to generate a spheroidal 3D plot that serves as a reference point in predicting element deformation under complex loads.

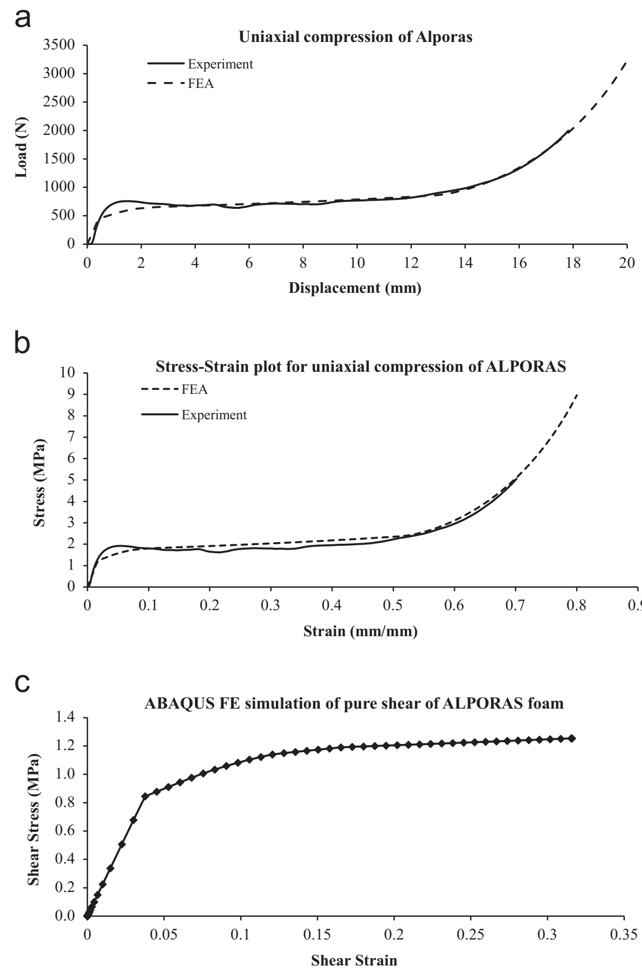

Fig. 1. (a) Experimental and FE simulation load–displacement plot of uniaxial compression of ALPORAS foam. (b) Experimental and FE simulation stress–strain plot of uniaxial compression of ALPORAS foam. (c) FE simulation of pure shear of ALPORAS foam (experimental results from Ref. [20]).

2.2. Numerical simulations of uniaxial compression and pure shear testing

Numerical simulations of uniaxial compression and pure shear of the foam material were performed to validate the material model and boundary conditions. The numerical simulations of uniaxial compression used a 2D plane strain model with a mesh of 40 × 20 elements. The model size was 25 mm high and 10 mm wide with a side edge constrained to have a planar symmetry. The bottom edge was constrained in the vertical direction only to allow lateral expansion (frictionless). The model was loaded at the top edge by a frictionless plate moving downward to a maximum displacement of 20 mm (80% compression).

To simulate metal foam shearing, the same model (2D, 4-node plane strain) mentioned above was used. A standard double-lap shear test was simulated, so all nodes were constrained in the vertical direction [1]. A horizontal displacement of 10 mm was applied at the top edge whilst the bottom edge was constrained horizontally and vertically.

2.3. Hemispherical punch indentation

Numerical simulations of hemispherical punch indentation on the foam were achieved using 2D, 4-node axisymmetric elements. The foam model was supported by a flat plate along the bottom edge which was defined as a fully-constrained rigid surface.

Download English Version:

https://daneshyari.com/en/article/1575403

Download Persian Version:

https://daneshyari.com/article/1575403

Daneshyari.com