Advances in Engineering Software 87 (2015) 1–12

Contents lists available at ScienceDirect

Advances in Engineering Software

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

Diffuse interface method for fluid flow and heat transfer in cellular solids

J. Ettrich [a,*], B. Nestler [c,b]

[a] Offenburg University of Applied Sciences, Faculty of Mechanical and Process Engineering, Badstrasse 24, 77652 Offenburg, Germany
[b] Karlsruhe Institute of Technology, Institute of Applied Materials, Haid-und-Neu-Str. 7, 76131 Karlsruhe, Germany
[c] Karlsruhe University of Applied Sciences, Institute of Materials and Processes, Moltkestrasse 30, 76133 Karlsruhe, Germany

ARTICLE INFO

Article history:
Received 27 January 2015
Received in revised form 15 April 2015
Accepted 26 April 2015
Available online 16 May 2015

Keywords:
Phase-field
Lattice-Boltzmann
Segmented tensorial mobility
Heat-transfer
Cellular solids
Open cell metal foam

ABSTRACT

This work presents a contribution on the numerical modelling capabilities for the simulation of fluid flow and heat transfer in cellular solids – in particular we focus on open cell aluminium foams. Rather than applying one of the classical academical or commercial numerical finite volume (FV), finite difference (FD) or finite element (FE) interface tracking methods, we base our models on an interface capturing phase field method (Nestler, 2005). A coupled diffuse interface lattice Boltzmann fluid flow solver (Ettrich, 2014) and a diffuse interface heat transfer approach (Ettrich et al., 2014) are combined in view of dealing with even more convoluted geometries, incorporating the dynamics of interfaces and complex multiphysics applications. Numerical results for the combined fluid flow and heat transfer simulations in open cell metal foams are in very good agreement with experimental data (Ettrich and Martens, 2012; Ettrich et al., 2012).

1. Introduction

Due to the limitations of experimental measurements of fluid flow and heat transfer at the pore scale level of open cell foams, it appears likely to apply numerical methods. However, simulation does not come at zero cost. The task is not only to provide a feasible geometrical and topological model for the foam under consideration, but also in the numerical treatment of the complex structure. Today, computing power is falling in cost, which allows for the simulation of even bigger domains and more complex physics. However, the pre- and post-processing tasks are still challenging, and the rule of thumb of the early times of numerical modelling – which states that about 80% of the effort is spent on mesh generation, model specification and evaluation of the results – still holds.

Even though sophisticated mesh generation tools are available in the field of academic as well as commercial simulation tools, the discretisation of a complex cellular solid is still a challenging, time consuming and most interactive task. To this effect, the combination of an automated method for foam generation [6], within the context of an interface capturing phase field approach [1] with appropriate methods for fluid flow [2] and heat transfer [7] provides a promising alternative. Apart from that, to start with the geometry, a suitable representation of the foam structure is essential.

To the authors best knowledge, there are only few publications on the coupled fluid flow and heat transfer simulation in open cell metal foams. Actually, the greatest difficulty is the realistic modelling of the foam structure. For instance, this can be realised by conducting expensive and time consuming investigations using industrial computed tomography (CT) scanning. The CT technique provides voxel data that require careful post-treatment to reconstruct the three-dimensional foam structures, or that need the development of a simplified modelling approach.

A numerical model for fluid flow based on a fundamental periodic structure of eight unit cells is examined in the work of [8]. The foam geometry is spatially resolved by unstructured tetrahedral meshes with 2.2×10^5, 4.4×10^5 and 8.3×10^5 elements. Simulations are carried out using the flow solver CFD-ACE of CFDRC, whereas all pressure drop results are about 25% underestimated. The authors claim that the reason for the discrepancy is due to the lack of side wall effects.

Numerical analysis of the conduction heat transfer in high porosity foam structures is done in [9]. The effective thermal conductivity of porous structures is most commonly modelled by means of empirical or semi-empirical models, based on different assumptions regarding pore scale and unit cell topology, which reveals significant variations. Therefore, a finite volume method is applied for evaluation and assessment. Foams are generated from regular structures (unit cells) as well as from tomographic data. The results show, that the fraction of solid phase in the struts and in the lumps (intersections) is the key parameter for successful

* Corresponding author.
E-mail address: joerg.ettrich@hs-offenburg.de (J. Ettrich).
URL: http://www.hs-offenburg.de (J. Ettrich).

http://dx.doi.org/10.1016/j.advengsoft.2015.04.012
0965-9978/

modelling, whereas the shape of the cells and struts has much less impact.

A completely different approach is pursued in [10], where a so called mesh-based microstructure representation algorithm (MBMRA) is employed for the modelling of cellular solids. In doing so, a reasonable fine mesh spans the whole simulation domain including fluid and solid regions. Random placed seed points are used for a rule based unstructured mesh growing algorithm for the solid matrix. The residual cellular structure depends on the predefined rules, and facilitates the generation of porous or fibrous structures for instance. Finally, the mesh provides a sharp interface representation of the fluid and solid domain, whereas a coupled Navier–Stokes solver for the fluid and a heat conduction code for the solid is used to simulate the conjugate heat transfer problem in a porous structure. Despite the interesting MBMRA-approach, the presented results lack of a comprehensive validation.

In [11] a multiple relaxation time lattice Boltzmann method is employed to simulate the flow field in a metallic foam sample. The geometry of the sample is gained from CT-scans conducted for a single cubic NiCr-foam sample of about 12.8 mm edge length. Experimental pressure drop data is successfully recovered for a low to medium Reynolds number regime.

Foam structures based on a unit cell modelling are employed in [12] for the coupled simulation of fluid flow and heat transfer using an in-house solver for the Navier–Stokes and energy equations. Foam structures of 10 ppi and 40 ppi are generated from regular unit cells. Different unstructured computational grids with about 7×10^5, 1.5×10^6 and 2.8×10^6 tetrahedral elements are used, whereas the computational domain spans several pores in streamwise direction, in order to be representative. Pressure loss as well as heat transfer data is found to be reasonable with respect to experimental data. However, the authors in [12] consider, that the geometry and foam creation process is tedious and time consuming.

A commercial Navier–Stokes CFD-Software (Ansys FLUENT) is used in [13] for the simulation of pressure drop in foam structures of different pore densities but similar relative porosity. Similar to [11], the authors received two-dimensional images through computer tomographic X-ray measurements, which are then used to derive three-dimensional foam structures as a starting point for flow simulations. Computational domains of different sizes, covering a representative section of the foam with respect to the number of pores, are generated for the samples with pore densities 5 ppi, 10 ppi, 20 ppi and 40 ppi, ranging from about 3×10^6 to 27×10^6 mesh elements. The numerical results are in good agreement with measurement data, with deviations of about 5–15% in particular.

The solution of the Navier–Stokes equations is accompanied by several numerical difficulties, the treatment of nonlinear convective terms or the solution of the Poisson equation to evaluate the pressure [14]. In lattice Boltzmann methods, the pressure is obtained by the equation of state and the nonlinear convective terms reduce to an advection type problem [15].

In the present work we employ a coupled phase field, fluid flow and heat transfer method presented in [1–3,7] to real world foam structures, and compare the flow properties with experimental data [2,4,5]. Heat transfer and fluid flow performance is compared by means of macroscopic parameters such as pressure drop, pressure drop per unit length and Nusselt number. The fluid flow part within the combined approach is solved with a lattice Boltzmann method.

2. Open cell foam modelling

Making the complex geometric structures of cellular solids available to a numerical simulation requires the representation in a digitally, machine readable format. There are a couple of approaches for modelling foam structures, for example using representative elementary volumes (REV), ranging from simple cubic cell to regular dodecahedron [16] and even more complex tetrakaidecahedron [17,18], cf. Fig. 1. These models describe the pore structure by means of a representative unit cell, characterising the structural configuration of the foam. The unit cell is a combination of nodes which are interconnected by edges. An open-cell metal foam can thus be described as a combination of nodes and edges. Within these models, mass is distributed along struts and at the knots of the regular structures, which are then used for an analytical derivation of foam characteristics like porosity or structure performance data such as thermal conductivity.

In [16] the foam structure is modelled using a simple cubic elementary cell, which allows a very simple representative structure, and is used to derive a theoretical model for pressure loss and heat transfer. An even more simplified approach is used in [19], presuming one-dimensional heat transfer and modelling foam structure by means of a batch of cylinders. The cross-sectional shape of the interconnected edges is studied in [16] using the dodecahedron with twelve flat pentagonal faces. Elementary cells with prisms shaped and round edges are modelled and compared in terms of porosity and pressure loss. The tetrakaidecahedron is a polyhedron with six quadratic and eight hexagonal faces [20], which allows to account for the imbalance of face and edge shape in a real foam structure. Among others it is used in [17,18,21–23] to model porosity, pressure loss and heat transfer in open cell metal foams.

However, detailed numerical investigations at pore scale level are either lacking in realistic structures due to a simplified representation by the above mentioned unit cells, or depend on a geometry which needs to be captured by a costly tomography, cf. [8,11,12].

In the course of this work a new algorithm is developed and described in detail in [6], that is capable of synthetically creating three dimensional cellular solids, foam or fabric like structures, with either open or closed cells. The workflow substantially follows three steps:

- First, the computational domain is gradually filled with a compact packing of spheres, which controls the scattering of the pores, cf. Fig. 2(a).
- Next, the basic topology of the structure is derived from a Voronoi-decomposition of the spatial domain, see Fig. 2(b).
- Finally, the cellular structure is created from the boundaries of the Voronoi-diagram, see Fig. 2(c).

The struts of cellular structures are often different with regards to their cross-sectional shape. For example, aluminium foams often have a triangular cross-sectional shape and thickening in the vicinity of the knots. The latter can be re-sampled by relating the thickness of the struts to the distance to their knots. The triangular shape is re-sampled, by also considering the connections of two Voronoi-regions. Here, an additional criterion if material is set or not, is given by the distance to the nearest boundary region, see Fig. 2(d).

3. Diffuse interface approach

Historically, the phase field approach is developed to provide an alternate and computationally efficient method for solving free boundary problems. The elegance of the method lies in the fact that, compared to the classical front-tracking methods, the phase-field approach avoids the explicit treatment of phase boundaries, and the boundary conditions are implicitly applied at the