International Journal of Heat and Mass Transfer 117 (2018) 527–537

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical simulation of natural convection in open-cells metal foams

D. Chiappini

University of Rome Niccolò Cusano, Dept. of Industrial Engineering, Via don Carlo Gnocchi 3, 00166 Rome, Italy

ARTICLE INFO

Article history:
Received 8 August 2017
Received in revised form 4 October 2017
Accepted 5 October 2017

Keywords:
Conjugate heat transfer
Open-cells metal foam
Lattice Boltzmann method
Finite volume method
Natural convection

ABSTRACT

The aim of this work is to present results obtained through a multi-physics solver used to numerically determine the thermal behaviour of an open-cells metal foam in the case of natural convection. Particular attention is addressed to the right geometry definition in order to capture the intrinsic foam characteristics, thus, the elementary cell used for describing the unitary metal foam one is the tetrakaidecahedron. In addition, in order to improve its isotropy, a random deformation on the basic structure has been introduced. This feature allows to locally deviate flow paths with obvious benefits in terms of heat exchange, with reducing preferential paths arising while packing the elementary cell over the volume. This mesh generation approach is coupled with a hybrid solver based on the lattice Boltzmann framework for the fluid-dynamics field reconstruction and on finite volume method for solving the energy equation so to retrieve temperature evolution. For the first time, high order differencing scheme are used in temperature discretization with obvious benefits in computational accuracy. Numerical results are compared with a set of experimental data available for a range of Rayleigh numbers and for different foam geometries. The agreement between numerical and experimental data is satisfactory with positive outcomes for future model developments.

1. Introduction

During recent past, metal foams are becoming everyday more attractive for heat transfer applications for their interesting properties, like low density, high surface area to volume ratio and capability to locally increase the turbulence. The latter is really important for thermal applications, because it is responsible of increasing the heat exchange efficiency. The always increasing attention to this kind of material is also striving the studies towards the understanding of natural convection mechanism which takes place in it. There are many industrial components based on the buoyancy force typical of natural convection processes, e.g. cooling of electronic devices. Open cell metal foams for industrial applications are usually characterized by a relatively high porosity value, which usually starts from to 80% and, for this kind of materials, accurate computational models for understanding the transport properties are still missing. Open-cell metal foams have been deeply analysed in terms of pressure drop and equivalent fluid-dynamics properties, or in forced convections configuration, for compact heat exchanger optimization. Many authors have highlighted the benefit of using such a material for heat exchange process both experimentally [1–4], or numerically [5], but they have also pointed out the drawbacks related to the high pressure drop characteristic of porous media. Moreover, different studies have been focused on the definition of thermal properties of metal foams [6–8]. A complete study aimed to determine thermal conductivity, permeability and inertial coefficient of different metal foams, characterized by different porosities and pores size, has been conducted in [9]. Here, a two-dimensional model based on hexagonal array has been validated with experiments with highlighting the correlation between the effective thermal conductivity with the porosity. Thermal modelling of metal foams has often been studied through the assumption of the Local Thermal Equilibrium (LTE) [1,10]. Nevertheless, in the recent past, an alternative approach, based on the Local Thermal Non-Equilibrium (LTNE), is starting to be used [11,12]. In 2005 [13], a numerical/experimental comparison has been presented for open cells metal foams with highlighting that natural convection strongly contributes to effective conductivity.

More in detail, the correct determination of the thermo-mechanical properties of an open-cells metal foam seems to be related to two different aspects. Firstly, the right topology has to be reconstructed by means of numerical techniques; secondly, a reliable multi-physic solver has to be developed and coupled with the complex mesh generated so far. Different approaches have been used to recover the physical behaviour of porous media through numerical simulations. For example, the porosity effect could be taken into account by using a simplified domain and

E-mail address: daniele.chiappini@unicusano.it

https://doi.org/10.1016/j.ijheatmasstransfer.2017.10.022
0017-9310/

introducing an additional term in the standard Navier-Stokes equations. However, this would introduce semi-empirical models which would generally require parameter tuning for each specific application. To overcome this potential issue, in this work, a detailed porous domain has been built by means of an automatic mesh generation. Fluid-dynamics effects are directly simulated, without modifying macroscopic equations and the algorithm is designed to match the metal foam properties, such as porosity and pores per inch (PPI) distribution. In the recent past, the random packed spheres approach [14,15], or the X-ray geometry reconstruction [16], have been extensively used in order to determine porous medium characteristics. The first method showed strong penalizations for thermal analysis despite being really intuitive, while the second one, despite being more realistic, needs a preliminary experimental campaign for porous medium geometry reconstruction. More recent advances have proposed the use of a more complex unitary cell, the Lord Kelvin one, named tetrakaidecahedron [5,17–19]. In this work, this mesh generation approach, modified in order to retrieve metal foam isotropy, has been coupled with a hybrid lattice Boltzmann method (LBM)/finite volume method (FVM) used to recover both Navier-Stokes and energy equations. In the recent past, the lattice Boltzmann method has been extensively used for the solution of complex flows and complex geometries due to its intrinsic simplicity in implementation [20,21] and it has been deeply used for a lot of applications involving multiphase flows oriented to engineering applications like sprays or injection analysis, [22,23], non-Newtonian fluids, [24,25], etc. Recent advances have also highlighted the possibility to use hybrid formulations which involve finite volume approach [26]. Thus, flow through porous media is a branch of computational fluid dynamic where LBM is a valid alternative to traditional numerical methods, as already demonstrated by recent advances [27,28]. In recent past, some authors have proposed the finite volume formulation for lattice Boltzmann framework both for isothermal [29,30], and thermal [31], flow through porous media. Apart from that, in its standard definition, LBM does not allow to recover energy equation for solving thermal field. For this reason, many models, which use an additional distribution function for temperature evolution, have been developed, focusing on natural or forced convection in standard media [32,33], or on determining thermal properties of fibrous porous ones [34–36]. Among these general issues, many authors focused on the correct definition of boundary conditions for thermal population [37,38]. In general terms, energy equation may be recovered by means of an additional population, using the same LB stencil, which allows tracing temperature as a passive scalar [39,40], with particular attention to external forcing implementation [41], or by means of FVM applied to the same grid of LB [42]. Recent advances have shown the possibility to deal with thermal lattice Boltzmann without distribution function evolution as well [43]. In order to perform natural convection for open-cells metal foams, this paper is mainly interested in conjugate heat transfer (CHT), which is a ubiquitous phenomenon in a large number of engineering applications [44,45]. The definition of the correct temperature at air/metal interface has interested many authors, who often used the approach proposed in [46]. In this work, a hybrid method is proposed in order to contemporary deal with fluid-dynamics and thermal problems, with always paying attention to critical aspects of CHT. In fact, as per its definition, it implies the interface between two different materials, here fluid and solid, and it has to fulfil two physical constraints. The first one is the temperature continuity at the interface, while the second is the normal flux continuity [47,48]. The developed model has been compared with a set of experimental data in order to validate its applicability to natural convection problems. The paper is organized as follows: in Section 2 a complete overview on mesh generation is provided, in Section 3 the hybrid algorithm is deeply described, with emphasising the two steps solution based on coupling LBM (Section 3.1) with FVM (Section 3.2), in Section 4 the experimental and the numerical set-ups are deeply analysed; in Section 5 results obtained from numerical simulations are compared with experiments, and, finally, in Section 6 some conclusions and future developments are shown.

2. Grid generation for porous medium

One of the critical aspects while analysing the open-cells metal foams is the possibility to deal with a reliable mesh generation approach, which would allow to match the real material properties, both in terms of fluid-dynamics performance and thermal behaviour. In the recent past, many approaches have been proposed, with highlighting pros and cons to be considered within the set-up definition. Some authors [14,15], use a random packed spheres approach in order to discretize a porous medium with the aim of building a numerical model of an open-cell metal foam where the pressure drop is the main observed property. Nevertheless, the method has a serious shortcoming for thermal simulations, the presence of isolated metal volumes which seem to "float". Obviously, these zones are useful in the evaluation of the pressure drop, but they are isolated from thermal point of view because they do not directly exchange heat with the other solid areas, being unconnected to each other.

For thermal simulations, an alternative mesh generation algorithm is proposed in order to overcome the limits of the method shortly presented above. This approach is based on the representation of a single elementary metal cell. Many strategies for this task have been already proposed in literature, here the tetrakaidecahedron [5,18,49], or Kelvin cell - regular volume with 14 faces (6 quadrilaterals and 8 hexagons), has been adopted. Starting from this elementary unity, the complete volume may be obtained with packing and repeating it along the three directions in space until reaching the desired metal foam volume.

Unfortunately, this volume reconstruction leads to preferential paths for fluid evolution, losing the natural isotropy of open-cell metal foams. Thus, in order to overcome this issue, a random perturbation is introduced, as proposed in [50].

Fig. 1(a) shows the unperturbed layout where stream-lines are aligned and organized following preferential paths, while Fig. 1(b) highlights the benefit of introducing random perturbations. It is worth nothing that, one of the main advantages in using metal foams for heat exchange processes is the possibility to locally increase mixing phenomena which would exploit thermal exchange mechanisms. Via comparing stream-lines orientations of Fig. 1(a) and (b), the advantages in perturbing tetrakaidecahedron topology appear quite evident with adding chaotic distortion to fluid paths.

Metal foam characteristics, such as PPI distribution and porosity may be obtained by modifying elementary cell dimension and connection struts diameters. It appears quite evident how the proposed method is computationally costly, requiring a high number of nodes in order to well represent the real geometry of the metal foam, nevertheless, it would assure metal continuity over all the domain with natural benefit for thermal simulations.

3. The numerical method

The approach used in the present work may be considered a kind of hybrid scheme, comparable with a fractional step approach. In fact, every time-step is split in two different sub-steps: in the first one the LBM solver is used to compute the fluid-dynamics field, while, in the second one, temperature is evaluated by using