International Journal of Thermal Sciences 86 (2014) 104–114

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

# Forced convection inside metal foam: Simulation over a long domain and analytical validation

Ahmed S. Suleiman, Nihad Dukhan*

*Department of Mechanical Engineering, University of Detroit Mercy, 4001 W. McNichols Rd., Detroit, MI 48221, USA*

ARTICLE INFO

*Article history:*
Received 28 November 2013
Received in revised form
19 June 2014
Accepted 22 June 2014
Available online 26 July 2014

*Keywords:*
Metal foam
Convection
Unit cell
Simulation
Modeling

ABSTRACT

In this paper a long domain, made up of numerous idealized geometrical cells in the flow direction, is used to numerically simulate heat transfer in the foam. The idealized cells mimic the complex morphology of open-cell solid foam. The large number of cells avoided periodicity issues and ensured minimal or no size and entrance effects. The conjugate laminar flow and energy equations are solved directly at the pore level; and the temperature fields are obtained for various approach velocities using a commercial numerical package. The details of the geometrical modeling and simulation are given in this paper. The commercial foam that was simulated had 20 pores per inch and porosity of 91.5%. The simulation showed a thermal development region. To validate the simulation, direct comparisons to analytical local fluid temperatures from the solution of volume-average two-equation model in the thermally fully-developed region were also carried out. Good agreement between the simulation and analytical results were obtained. The results are encouraging and lend confidence to the geometrical modeling and simulation approach.



## 1. Introduction

Convection heat transfer in open-cell metal foam is gaining more interest compared to other potential applications of this kind of porous media. Key characteristics of the foam, from a heat transfer point of view, are the high conductivity of the solid phase and the large surface area per unit volume. This is in addition to the vigorous mixing of the flowing fluid and continuous destruction of boundary layers provided by the ligaments of the foam, which enhances convection between the solid and the fluid phases.

The internal morphology of open-cell foam is complex web-like and random. Exact solutions of transport equations, including all pertinent effects, are extremely difficult and impractical. Nonetheless, metal foam is a porous medium, and some analytical treatments of the heat transfer in porous media in general do exist, some of which will be briefly presented here. Lee and Vafai [1] presented an analytical model for the solid and the fluid temperatures for Darcy flow in porous media between two parallel plates. They assumed equal temperatures at the heated base, i.e., $T_s = T_f = T_w$. They identified three heat transfer regimes dominated by fluid conduction, solid conduction or convection between the solid and the fluid. Vafai and Kim [2] solved the local-thermal-equilibrium governing equations for a porous medium between two heated parallel plates.

Haji-Sheikh and Vafai [3] analyzed heat transfer in porous media imbedded inside ducts of different shapes assuming local thermal equilibrium and applying a constant-wall-temperature boundary condition. Minkowycz and Haji-Sheikh [4] solved the local-thermal-equilibrium equations for the case of parallel plates and circular porous passages including the effect of axial conduction.

Lu et al. [5] analyzed forced convection in a tube filled with a porous medium subjected to constant surface heat flux. . The two-equation model, which relaxes the thermal equilibrium assumption, was solved. A closed-form for the solid and fluid temperatures was presented. They applied the results to tubes filled with metal foams used as heat exchangers. In a follow-up study, Zhao et al. [6] presented an analytical solution for a tube-in-tube heat exchanger, for which the inner tube and the annulus were filled with metal foam. Both of these studies employed the Brinkman-extended Darcy momentum model.

Analytical solutions in porous media continue to be sought after due to their utility in practical engineering design, identifying trends of critical variables, parametric studies and for validating numerical models, see for example Xu et al. [7], Qu et al. [8] and Xu et al. [9].

* Corresponding author. Tel.: +1 313 993 3285; fax: +01 313 993 1187.
*E-mail address:* nihad.dukhan@udmercy.edu (N. Dukhan).

http://dx.doi.org/10.1016/j.ijthermalsci.2014.06.022
1290-0729/

Detailed numerical analysis at the pore (microscopic) level can assist in better understanding intricacies of the complex flow and temperature fields inside metal foam. In order to solve the governing equations at this small scale, an idealized geometrical model of the internal microstructure structure of the foam is crucial. Such model must be able to reasonably capture the morphological parameters of actual foam, i.e., shape and diameters of actual foams' cell and open windows, shape and thickness of foam's ligaments, specific surface area and porosity.

Numerous studies have attempted to model the geometry of open-cell metal foam by defining a representative elementary volume (rev) that captures the relevant characteristics of these materials in order to study transport of momentum and heat. See for example, du Plessis et al. [10] who represented the foams by a set of rectangular prisms and Boomsma et al. [11] who represented the cell of the foam by a tetrakaidecahedron (six 14-sided polyhedra having 12 pentagonal and two hexagonal faces, and two pentagonal dodecahedra), and used a 'periodic' unit that consisted of eight such cells.

Krishnan et al. [12] simulated flow and heat transfer in open-cell metal foam using a single ideal body center cube (bcc) unit cell. They assumed fully-developed flow and invoked periodicity. Karimian and Straatman [13] used a unit cell based on interconnected sphere-centered cubes to represent the internal structure of foam. For their computational domain, they used two such cells in the flow direction and assumed fully-developed laminar flow. Periodic boundary conditions were imposed at the inlet and outlet.

Bai and Chung [14] used a sphere-center Kelvin structure to simulate fluid flow in metal foam by studying two kinds of cells: an interior cell (away from a confining wall) and a boundary cell in order to assess interactions with a solid boundary. In each case, a single unit cell was used in the simulation. Horneber et al. [15] used a single and multiple Kelvin cells to simulate flow in a metal-foam reactor. Constant inlet velocity was imposed and constant outlet pressure was prescribed. Results showed that the single cell case always produced higher pressure as compared to the case of multiple cells. The authors confirmed the presence of entrance and exit effects that could not be captured by simulating a single cell.

Kopanidis et al. [16] carried out numerical simulations for the conjugate flow and heat transfer in open-cell metal foam. They used a cell geometry that was obtained by minimizing surface energy. The computational domain consisted of 10 cells. Both laminar and turbulent flows were investigated for 10- and 20-pore-per-inch (ppi) high-porosity metal foam. Boundary conditions included fixed inlet velocity and fixed inlet–outlet pressure difference, noting that the former represented the entrance region of a larger foam block, while the latter represented fully-developed flow conditions.

The current study will briefly present the details of obtaining a bcc cell, and how the cell is linked to the morphology of actual metal foam. This unit cell is similar to Kelvin's cell and was believed to have minimum surface area-to-volume ratio compared to other space filling geometries [12,17], until 1994, when Weairea and Phelan [18] provided a counter example having a slightly lower surface area. The bcc geometry is relatively simple and easier in terms of meshing compared to others used in the literature [11,16].

Unlike many other modeling studies which considered a domain of only one cell [12,14], or few cells [11,13] and asserted that domain can be considered a repeating unit representing a large foam domain, the current study employs a relatively large number of cells in order to insure fully-developed conditions and to avoid the issue of periodicity and size effect altogether. Another improvement provided by the current study is the direct and meaningful validation of the simulation results using the volume-average analytical solution.

## 2. Idealized unit cell

The internal structure of typical open-cell aluminum foam produced by casting is shown in Fig. 1. The structure is formed by connecting solid ligaments (or fibers) to form void spaces (cells) and open windows-each referred to as a pore. In reality, this structure varies, but on average, this basic structure can be assumed homogeneous and repeatable.

For simulation of fluid flow and heat transfer inside the foam, this actual geometry is idealized as follows. In step 1 of Fig. 2, consider a cube of side length $a_1$. To obtain the idealized geometry of the foam, 15 equal spheres having a radius $R$ (equal to $a_1/2$) are subtracted from the cube as shown in the other steps of Fig. 2. In step 2, six polar spheres are subtracted from the cube. In step 3, a central sphere is subtracted from the cube. In step 4, six spheres at the vertices are subtracted from the cube, which results in the idealized cell geometry of the foam. The distance between the centers of the central sphere and a polar sphere (with overlapping) is adjusted to be $a_1$. The distance $a_1$ can be related to actual foam's morphology, i.e., the number of pores per inch (ppi), which is a common industrial designation. Open-cell 10-ppi commercial aluminum foam on average has 5 central cells (or central spheres) in one linear inch and 40-ppi foam would have 20 spheres per inch, and so on.

The steps of Fig. 2 for obtaining the idealized cell geometry can be easily performed in a Computer-Aided-Design (CAD) package. This geometry along with the sphere-subtraction method produces a cell diameter and a pore diameter within 3.3% and 15.0% of averaged measured values for commercial aluminum foam [19], respectively. In addition, the sphere-subtraction method produces the triangular shape of the cross-section of the ligaments, as can be found in actual high-porosity foam.

## 3. Simulation

Convection heat transfer simulations were carried out in the commercial software package COMSOL. A foam (solid and fluid) domain having a length of 83.8 mm in the flow direction and a height of 38.1 (perpendicular to the flow direction) was simulated. It is worth noting that due to symmetry, this domain represents flow and heat transfer between heated parallel plates separated by a distance twice the domain's height (76.2 mm). There were a total of 495 cells representing 20-ppi foam; 33 cells in the flow direction and 15 cells perpendicular to the flow direction. This domain also contained a one-mm-thick wall (to apply a heat flux mimicking what happens in practice, e.g., cooling of a heat generating device). This solid wall is shown at the bottom of the insert in Fig. 3. The calculated porosity of the idealized domain was 94.9%. To save on computational cost exploiting symmetry, the thickness of the

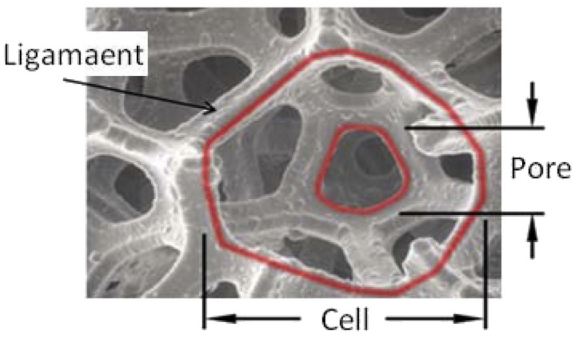


**Fig. 1.** Actual morphology of open-cell metal foam made by casting.