International Journal of Heat and Mass Transfer 88 (2015) 508–515

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical investigation of pressure drop and heat transfer through reconstructed metal foams and comparison against experiments

Andrea Diani [a], Karthik K. Bodla [b], Luisa Rossetto [a,*], Suresh V. Garimella [c]

[a] Dipartimento di Ingegneria Industriale, Università degli Studi di Padova, Via Venezia 1, 35131 Padova, Italy
[b] Electronics Cooling Lab, General Electric Global Research Center, 1 Research Circle, Niskayuna, NY 12309, United States
[c] School of Mechanical Engineering, Purdue University, 585 Purdue Mall, West Lafayette, IN 47907-2088, United States

ARTICLE INFO

Article history:
Received 24 September 2014
Received in revised form 13 April 2015
Accepted 13 April 2015
Available online 15 May 2015

Keywords:
Microtomography
Metal foams
Pore scale simulation
Pressure drop
Heat transfer coefficient
Forced air convection
Electronics cooling

ABSTRACT

Direct numerical simulation of transport in foam materials can benefit from realistic representations of the porous-medium geometry generated by employing non-destructive 3D imaging techniques. X-ray microtomography employs computer-processed X-rays to produce tomographic images or slices of specific regions of the object under investigation, and is ideally suited for imaging opaque and intricate porous media. In this work, we employ micro-CT for numerical analysis of air flow and convection through four different high-porosity copper foams. All four foam samples exhibit approximately the same relative density (6.4–6.6% solid volume fraction), but have different pore densities (5, 10, 20, and 40 pores per inch, PPI). A commercial micro-computed tomography scanner is employed for scanning the 3D microstructure of the foams at a resolution of 20 μm, yielding stacks of two-dimensional images. These images are processed in order to reconstruct and mesh the real, random structure of the foams, upon which simulations are conducted of forced convection through the pore spaces of the foam samples. The pressure drop values from this μCT based CFD analysis are compared against prior experimental results; the computational interfacial heat transfer results are compared against the values predicted by an empirical correlation previously reported, revealing excellent agreement between the numerical and experimental/empirical hydraulic and thermal results, thus highlighting the efficacy of this novel approach.

1. Introduction

Open-celled metal foams are a network of randomly oriented ligaments, consisting of connected pores with nearly uniform size and shape. These foam structures have several beneficial heat transfer characteristics, such as a large surface area per unit volume, high thermal conductivity, and enhanced flow mixing capabilities [1,2]. These desirable, multi-functional characteristics make foams suitable extended surfaces for diverse applications such as air conditioning, refrigeration, and electronic cooling. Metal foams are primarily characterized by two parameters, *viz.*, the volumetric porosity denoted by ε, which is defined as the ratio of total void volume to cumulative volume occupied by the solid matrix and void space, and the number of pores per linear inch (PPI).

Much of the work in literature on heat transfer through metal foams has been focused on single phase flow. Similarly, a majority of the literature views foam structures as an alternative extended surface (i.e., fins) using air as the coolant. Examples of experimental studies on forced convection of air through metal foams include those by Younis and Viskanta [3], Hwang et al. [4], Hsieh et al. [5], Duckhan and Chen [6], Incera Garrido et al. [7], Mancin et al. [2,8], and Zhao [1]. Among these studies, Mancin et al. [2] reported heat transfer and pressure drop measurements performed for forced convection of air through multiple copper foam samples. The sample space investigated consisted of PPI values in the range 5–40, and porosity values in the range 0.905–0.934. The effect of different geometrical parameters of the foam on the global heat transfer coefficient, normalized mean wall temperature, pressure gradient, permeability, inertia coefficient, and drag coefficient were described. Zhao [1] provided a review on several thermal transport mechanisms in open-celled foams including conduction, forced convection, natural convection, thermal radiation, as well as pool boiling and flow boiling heat transfer. Very few researchers had focused on detailed heat transfer analysis at the pore scale, either by numerical or experimental approaches.

Traditional approaches of modeling fluid and thermal transport through metal foams approximate stochastic foams as periodic porous materials, and employ a single unit cell for analysis. Lu et al. [9] developed a simple analytical model to evaluate the utility

* Corresponding author. Tel.: +39 049 8276869; fax: +39 049 8276896.
E-mail address: luisa.rossetto@unipd.it (L. Rossetto).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.04.038
0017-9310/

Nomenclature

a_{sv}	surface area per unit of volume [$m^2\ m^{-3}$]
c_p	specific heat at constant pressure [$J\ kg^{-1}\ K^{-1}$]
e_A	absolute deviation [%]
e_R	relative deviation [%]
G	mass velocity [$kg\ m^{-2}\ s^{-1}$]
H	foam core height [m]
m	parameter defined in Eq. (8) [m^{-1}]
l	fiber length [m]
L	parameter defined in Eq. (9) [m]
p	pressure [Pa]
Pr	Prandtl number [–]
q''	heat flux [$W\ m^{-2}$]
Re	Reynolds number [–]
t	fiber thickness [m]
T	temperature [K]
$\overline{T}_{air}$	mean air temperature [K]
$\overline{T}_{wall}$	mean wall temperature [K]
u_i	velocity in the i-direction [$m\ s^{-1}$]
u_j	velocity in the j-direction [$m\ s^{-1}$]
x_i	coordinate in the i-direction [m]
x_y	coordinate in the y-direction [m]

Greek symbols

α	heat transfer coefficient [$W\ m^{-2}\ K^{-1}$]
$\alpha_{empirical}$	empirical heat transfer coefficient [$W\ m^{-2}\ K^{-1}$]
$\alpha_{numerical}$	average numerical interfacial heat transfer coefficient [$W\ m^{-2}\ K^{-1}$]
ε	porosity [–]
λ_{air}	air thermal conductivity [$W\ m^{-1}\ K^{-1}$]
λ_{mat}	foam material thermal conductivity [$W\ m^{-1}\ K^{-1}$]
μ	dynamic viscosity [Pa s]
ρ	density [$kg\ m^{-3}$]
σ_N	standard deviation [%]
Ω^*	foam finned surface efficiency [–]

of metal foams as compact heat exchangers. A cubic unit cell model consisting of slender cylinders as edges was developed to capture the most important trends of energy flow due to forced convection, and conduction through cell ligaments of the cellular foam.

Boomsma et al. [10] modeled the fluid flow through porous media with periodic unit cells. The energy minimization tool, Surface Evolver [11], was employed to obtain the microstructure of the foam. The pressure drops from the numerical simulations were compared against previous experimental results of Boomsma and Poulikakos [12]. Under identical conditions, it was reported that the pressure drop values predicted by the simulations were consistently approximately 25% lower than the experimental values, and this underestimation was attributed to the exclusion of the bounding wall effects which would increase the pressure drop.

Krishnan et al. [13] performed direct numerical simulation of thermal transport through open-celled foams using different periodic unit cell geometries. They used three packaging arrangements of spheres, *viz.*, body centered cubic, face centered cubic and A15 lattice, to model the structure of the foams. Important thermal and hydraulic properties such as effective thermal conductivity, pressure drop and Nusselt number were computed for aluminum foams with both air and water as the interstitial fluid, and the results were successfully compared against experimental values and semi-empirical models available in the literature.

Annapragada et al. [14] proposed a computational method to analyze fluid flow and heat transfer in compressed open-celled metal foams. Their unit cells were similar to those considered by Krishnan et al. [13]: body centered cubic, face centered cubic, and A15. They first validated the results for the A15 model by comparing the normalized permeability of compressed polyurethane foams against experimental results obtained from Dawson et al. [15]. After validation, the model was employed to predict permeability, friction factor, Nusselt number and effective thermal conductivity of aluminum foams, highlighting the effect of the compression on these parameters.

Bai and Chung [16] simulated the flow of air in a 10 PPI foam sample of 97% porosity, employing a sphere-centered tetrakaidecahedron unit cell to represent the actual structure. They considered two types of cells, an interior cell and a boundary cell. Numerical pressure drop results were compared against experimental data from Leong and Jin [17], and good agreement was shown. The wall boundary cells experienced approximately 5% higher pressure drop than those in the interior, attributable to the no-slip condition and the larger velocity gradients at the wall. Wu et al. [18] simulated the interfacial heat transfer through porous ceramic foams numerically, with air as the coolant. The ceramic foams were also represented by ideally packed tetrakaidecahedron structures, and the porosity was controlled by adjusting the curvature of the blending faces. Based on the numerical simulations, a correlation was developed for predicting local and volumetric heat transfer coefficients, covering a broad range of porosities, velocities, cell sizes and temperatures [18].

There has been growing interest in the use of X-ray microtomography techniques for a variety of applications such as material characterization and reverse engineering. For example, Fiedler et al. [19] numerically identified defects produced during manufacturing of an open-celled metal foam. They performed finite element calculations based on microcomputed tomography data of the samples. The effective Young's modulus and 0.2% offset yield strength were calculated, and an equivalent plastic strain was used to identify weakness within the material.

Micro computed tomography images may also be employed as the starting point for CFD analysis. Metal foams are inherently stochastic; thus, unit-cell based models only approximate the true microstructure and fail to capture the intricate details of fluid flow and heat transfer in such media. Recent advancements in computing architecture have led to increased processor speeds and memory, which enable tomography scans to be employed for mesh generation and subsequent, detailed fluid-thermal performance analysis of random porous materials such as metal foams.

Bodla et al. [20] adopted this approach to compute heat transfer and fluid flow parameters for aluminum foams of varying PPI, but with approximately the same relative density. The numerical results for thermal conductivity, permeability, friction factor and heat transfer coefficient were compared against experimental values and empirical correlations from the literature. The effective thermal conductivity was found insensitive to decreasing the pore size (as porosities were about the same), whereas the heat transfer coefficient and pressure drop were observed to increase as pore size was decreased.

Mendes et al. [21] numerically studied the effective thermal conductivity of open-celled, foam-like structures by considering different ordered structures and four real foams. They used Kelvin and cubic unit cells, and also took into account the distribution of solid phase between the struts and the nodes. Starting from the numerical results, they developed four correlations for the estimation of the effective thermal conductivity. In addition, the