International Journal of Thermal Sciences 111 (2017) 234–245

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Simulation and analytical validation of forced convection inside open-cell metal foams

H. Liu [a], Q.N. Yu [a, *], Z.G. Qu [b], R.Z. Yang [a]

[a] National Key Laboratory of Human Factors Engineering, Astronaut Center of China, Beijing 100094, China
[b] Key Laboratory of Thermal Fluid Science and Engineering of Ministry of Education, School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an, Shaanxi 710049, China

ARTICLE INFO

Article history:
Received 27 February 2016
Received in revised form
2 July 2016
Accepted 4 September 2016

Keywords:
Metallic foams
Transfer matrix method
Heat transfer enhancement
Kelvin structure
Geometric model

ABSTRACT

Two ideal three-dimensional geometries are generated using cylinder-joint and sphere-subtraction methods, respectively. The former is firstly used to establish simple geometrical relations among average pore diameter, strut diameter, ligament diameter, porosity, and specific surface area. An analytical model of heat transfer in a foamed heat exchanger is then proposed using the transfer matrix method and one-equation model. The heat transfer predictions have been compared with available experimental data and favorable agreements have been obtained. Heat transfer coefficient and pressure drop in both two geometries with different porosities and cell dimensions are investigated using COMSOL Multiphysics. Pressure drop verifications among numerical results, semi-empirical models, and experimental data show good feasibility of Kelvin structure. The proposed Kelvin geometric model is shown to be self-consistency with respect to both its geometrical characteristics and thermal properties, which can be adopted for finer details of pressure drop phenomenon.

1. Introduction

Recent developments and commercial availability in the processing technology have led to a range of novel lightweight materials for structural, thermal, acoustic and other engineering applications [1]. Metal foams are such a new class of materials with unique properties consisting of randomly oriented cells with nearly uniform sizes and shapes. Morphologies exist for metallic foam materials are categorized into open cell and closed cell foams depending on the cell structure. When the pores are interconnected with each other, the foam is called "open-cell foam", whereas when most of the pores are isolated from one another, the resulting foam is known as "closed-cell foam" [2]. Thereinto, open cell foams have random interconnected voids that allow fluid flow through them. Hence, they can be ideally suited for use in heat exchanger applications to enhance heat transfer by conduction and convection through the continuous metal ligaments [3]. The motivation is attributed to the high surface area to volume ratio as well as enhanced flow mixing (convection) due to the tortuosity of metal foams [4]. Because these two features play a crucial role in the heat transfer enhancement, modeling of the thermal properties is of primary importance for the prediction and optimization of the capability and performance of these materials. Moreover, extreme randomness has largely restricted the study of metallic foams to experimental methods for optimization design [5].

Morphology models can be used to investigate the heat transfer in open cell foams both analytically and numerically. Analytical study for forced convective heat transfer in metallic foams or other porous media has gained much attention. Xu et al. [6] develop a benchmark-like analytical solution for velocity and temperature using the Brinkman-Darcy and two-equation models based on the volume-averaging method. Considering the microstructure of foams, Lu et al. [7] proposed simple cubic unit cells consisting of heated cylinders. The friction and heat transfer properties were analytically investigated and used to guide the design of optimum foam structures that would maximize heat transfer efficiency. However, the model oversimplifies the foam structure and its prediction of heat transfer leads to overestimates. Bai and Chung [8] develop a simplified analytical model based on diamond-shaped unit cells and an unit-cell CFD model based on a structure of sphere-centered open-cell tetrakaidecahedron. Because it is difficult to define proper boundary conditions for heat transfer simulation on a unit cell, so the heat transfer rate is predicted

* Corresponding author.
E-mail address: accpaper@outlook.com (Q.N. Yu).

http://dx.doi.org/10.1016/j.ijthermalsci.2016.09.006
1290-0729/

Nomenclature

A	cross sectional area, m^2
c_p	specific heat capacity of fluid, $J \cdot kg^{-1} \cdot K^{-1}$
d	strut diameter, m
d_f	ligament diameter, m
d_p	average pore diameter, m
h	local heat transfer coefficient, $J \cdot kg^{-2} \cdot K^{-1}$
$\overline{h}$	overall heat transfer coefficient, $W \cdot m^{-2} \cdot K^{-1}$
H	one half of the heat exchanger thickness, m
k_s, k_f	thermal conductivity ratios of solid and fluid, $W \cdot m^{-1} \cdot K^{-1}$
L,W	length and width of the sandwiched heat exchanger, m
l	cell wall length, mm
$\dot{m}$	mass flow rate, $kg \cdot s^{-1}$
Nu	Nusselt number
Δp	pressure drop, Pa
Pr	Prandtl number, $c_p \mu_f / k_s$
q	heat flux, $W \cdot m^{-2}$
Re	Renolds number
T	temperature
u	velocity, $m \cdot s^{-1}$
(x,y,z)	global Cartesian coordinates

Greek symbols

α_A	specific surface area, $m^2 m^{-3}$
δ	length of ligament from node center to node center, m
ε	porosity, 1-ρ
μ	dynamic viscosity, $kg \cdot m^{-1} \cdot s^{-1}$
ν	kinematic viscosity, $m^{-2} \cdot s^{-1}$
ρ_f	density of fluid, $kg \cdot m^{-3}$
ρ	relative density (solid fraction), 1-ε

Subscripts

f	fluid
s	solid
w	substrate wall
0	inlet of the channel

analytically while the pressure drop is predicted by CFD numerical methods, which are both compared with available experimental data, rather than validation with each other. As for numerical method, unlike many other modeling studies which considered a domain of only once cell [8,9], or few cells [10,11] and asserted that domain can be considered a repeating unit representing a large foam domain, Suleiman and Dukhan [12] employs a relatively large number of body center cube cells for simulation in order to insure fully-develop conditions and to avoid the issue of periodicity and size effect altogether. The simulation is validated by direct comparisons to analytical local fluid temperatures from the solution of volume-average two-equation model.

For open-cell metal foam morphology modeling, minimum surface energy cell geometry, variable cross-section ligaments and irregular shaped lump are the three dominant features. Kelvin's cell and Weairea-Phelan (W-P) are believed to have the minimum surface energy although the Weairea-Phelan's cell has a slightly lower surface area. The sphere-subtraction method produces the triangular shape of the cross-section of the ligaments and irregular shaped lump at the vertexes of the base geometry. Therefore, a combination of these two features is a feasible way to model the open cell foams. Bianco et al. [13] numerically investigate the heat transfer and pressure drop in open cell foams using both Kelvin and W-P foam models, whereas the solid phase is not taken into account. A comparison between the numerical results obtained from the two foam models shows that the Kelvin and W-P models can be used indifferently only for very high values of the porosity (>90%). However, these numerical researches for heat transfer are not well connected with the analytical models. This will lead to difficulty for further optimization design study of heat exchanger filled with metal foams using an analytical method.

In order to optimize the heat transfer performance of a heat exchanger filled with open-cell metallic foams, it would be desirable to be able to quantify the morphology model by characteristic parameters which would be directly measured. Generally, two quantities, the 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 inch (PPI) are used to describe the porous medium. The former is often considered by the previous analytical or numerical study, while the latter is not always directly linked to the morphology modeling and substituted by the average pore diameter.

In general, the available investigations, whatever it is experimental, theoretical, or simulational, more or less collide with each other. In other words, a more unified and self-consistency model is essentially needed. This is the final target pursed by the present study.

In the present study, both an analytical model and finite element simulation are developed to investigate the performance of a compact heat exchanger filled with open-cell foams. Firstly, a three-dimensional periodic unit cell with cylindrical struts is generated based on Kelvin structure to establish the geometrical relations for average pore diameter and specific surface area. An analytical model is proposed using the transfer matrix method to evaluate the heat transfer characteristics of a foamed heat exchanger. Secondly, to better describe the morphology of real foams, the open-cell foam is modeled by the idealized sphere-centered Kelvin by sphere-subtraction method using commercial software COMSOL Multiphysics. A direct simulation of the geometry used for analytical investigation is performed for validation. The comprehensive details of obtaining the two geometry models are presented. Considering the mutual verification, both the analytical and simulation models are linked to the morphology of actual foams with PPI and porosity. A long domain made up of several idealized geometrical cells is employed to investigate the convective heat transfer characteristics. Grid independence is investigated and simulation results are compared with present analytical predictions and available experimental data. The analytical predictions are also compared with experimental data of different Reynolds numbers.

2. Analytical heat transfer model

2.1. The physical problem description

The schematic diagram of a heat exchanger filled with open-cell foams cooled by forced convection is shown in Fig. 1. Two open-cell foam layers are symmetrically sintered on the upper and bottom substrate plates subjected to uniform heat flux. The cooling fluid is forced to flow across the channel of thickness $2H$ at the inlet ($x = 0$) and exits at the outlet (x=L). The module is thermally insulated on the left and right sides ($y = \pm W/2$). For simplicity, the length of the