International Journal of Heat and Mass Transfer 123 (2018) 787–804

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Homogenized and pore-scale analyses of forced convection through open cell foams

Dig Vijay [*], Pitt Goetze, Rhena Wulf, Ulrich Gross

Institute of Thermal Engineering, TU Bergakademie Freiberg, Gustav-Zeuner-Str. 7, Freiberg 09596, Germany

ARTICLE INFO

Article history:
Received 15 October 2017
Received in revised form 12 February 2018
Accepted 2 March 2018

Keywords:
Porous media
Interstitial convection
Thermal dispersion
Thermal conduction
Open cell foams

ABSTRACT

Open cell foams have desirable geometrical characteristics that make them a suitable choice in various heat exchanger applications. The objective of this study is to determine such volume-averaged key parameters that can characterize the complex thermal transportation process through open cell foams. These key parameters are represented by effective thermal conductivity, k_e, volumetric heat transfer coefficient, h_v, and dispersion conductivity, k_d. In order to determine these parameters, detailed pore-scale simulations through the representative element volumes (REVs) of the actual foam structures are performed. Moreover, knowing the fact that the successful implementation of simplified foam structures as a suitable representative of the actual foam structures can simplify the complexity of the problem, it is also investigated. In the presented work, various microscopic pore-scale models are implemented for both simplified and actual foam structures to determine the key parameters. Subsequently, these key parameters are implemented into two different homogenized macroscopic models to predict the temperature fields of large-scale steady-state and transient forced convection processes. The numerical outcomes of homogenized macroscopic models are validated with the experimental data, which is available for a set of ceramic foams having different pore size (10–30 PPI) and porosity (79–87%). As a consequence of the validation process, the findings of this study reveal that the proposed methodology successfully predicts the values of the concerned key parameters. Further, it is observed that simplified foam structures cannot represent the actual foam structures, as the tortuous shape of open cell foams bound to enhance the advection and dissipation of heat due to recirculation and eddy formation.

1. Introduction

The open cell foams are continuous porous structures having high open porosity. They have desirable geometrical characteristics that make them suitable for various existing and forthcoming thermal management applications. Some existing applications of open cell foams are electronic cooling systems, flow diffusers, heat exchangers, cryogenic tanks, energy absorbers, filters for metal purification, substrates for catalysts, combustion incinerators and catalytic diffusion burners [1–6]. The reason that makes open cell foams so effective is their highly complex and tortuous shape, which gives them excellent geometrical characteristics such as high specific surface area, low density and high porosity [7]. Nevertheless, for designing any open cell foam based application, it is required to determine the functional values of all the key parameters that can characterize the complex thermal transportation processes through open cell foams. One physical process that commonly occurs in most of the above mentioned applications is the forced convection of gases/liquids through open cell foams, which is a complex phenomenon divided into various modes of heat transfer. These modes of heat transfer are thermal conduction, interstitial convection, fluid-phase thermal dispersion and thermal radiation. The key parameters that characterizes these concerned modes of heat transfer are solid and fluid phase effective thermal conductivities, k_{se} and k_{fe}, respectively, volumetric heat transfer coefficient, h_v, dispersion conductivity, k_d and radiative thermal conductivity, k_r. In order to determine these key parameters there are two types of numerical models available in the literature [7] namely microscopic and macroscopic models.

1.1. Pore-scale microscopic models

The microscopic numerical models are implemented to perform pore-scale analysis of thermal transportation processes through open cell foams [8]. In some cases, to reduce the computational efforts without losing the essential geometrical features, the full-scale geometry of the open cell foam is represented by a

---

[*] Corresponding author.
E-mail address: digvijay.1431987@gmail.com (D. Vijay).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.03.008
0017-9310/© 2018 Elsevier Ltd. All rights reserved.

Nomenclature

Latin symbols

a_s	specific surface area, m^2/m^3
d	pore diameter, m
d_{str}	strut diameter, m
h_v	volumetric heat transfer coefficient, W/m^3 K
I	turbulence intensity, %
$k_{d,x}$	longitudinal dispersion conductivity, W/m K
$k_{d,y}$	lateral dispersion conductivity, W/m K
k_e	effective thermal conductivity, W/m K
L_r	strut ratio
L_{str}	strut length, m
Nu	average base plate Nusselt number
Pr_{fe}	fluid-phase Prandtl number
Re	strut diameter Reynolds number
Re_d	pore diameter Reynolds number
t_m	actual transient process time, s
T_∞	ambient air temperature, K

Greek symbols

ε	foam porosity
θ	dimensionless temperature (steady state)
σ	root mean square deviation
ϕ	dimensionless temperature (transient)

Subscripts

b	base plate
f	fluid
i	inlet
o	outlet
s	solid
TO	truncated octahedron
w	wall

small-scale representative element volume (REV). One such example of a REV is shown in Fig. 1(a). Moreover, to simplify the problem and to avoid dealing with the complex 3D models generated through computed tomography (CT), various researchers have implemented the substitutes of the actual foam structures in terms of simplified foam structures [9–16]. One such example, in terms of a Kelvin cell model, is shown in Fig. 1(c).

Having classified the microscopic models in Fig. 1, as far as the flow regime is concerned the fluid flow through open cell foams is classified into four categories i.e. Darcy, Forchheimer, post-Forchheimer and Fully turbulent flow [17–19]. Although, in most engineering applications, it is highly probable that one is dealing with the turbulent flow, but most of the available numerical research works that are focused on open cell foams are performed only for the laminar flow regime. For instance, Boomsma et al. [20] performed numerical simulations for the fully developed flow through a simplified foam structure. They observed that the pressure drop across the simplified foam increases as the lateral channel boundaries are changed from the periodic boundary conditions to the no-slip boundary conditions. Krishnan et al. [11,12] also performed DNS through simplified foam structures. They determined pressure drop data for relatively lower Reynolds number. In a similar case, Xu et al. [21] also implemented a simplified foam structure and observed that for $Re_d = 254$ the flow becomes unsteady and the vortices start forming behind the struts.

Based on the above mentioned works for laminar flow, the next logical step would be the modeling of turbulent flow through the simplified structures, as through such studies the suitability of simplified foam structures can be scrutinized. However, even for laminar flow, Iasiello et al. [8] found a discrepancy of more than 100% between the pressure drop data determined for the CT generated actual foams and the simplified foam structures. Therefore, some researchers decided to directly perform the fluid flow simulations through the actual foam structures. Such as, Petrasch et al. [22] and Haussener et al. [23] have determined the permeability of open cell foams by numerically determining the pressure drop data for a laminar flow. Diani et al. [24] also investigated the fluid flow and heat transfer through detailed models that were generated through image based techniques. But even their work is restricted to laminar flow. Therefore, to get an insight into the flow behavior at high Reynolds numbers, Hutter et al. [25] performed large eddy simulations (LES) through a simplified foam structure. They also manufactured the same structure by employing stereolithography. Subsequently, based on the experimental evidences, they found that the measured pressure drop and the turbulence statistics compare well with the simulated results. But not to forget that the foam implemented by Hutter et al. [25] is still a simplified structure, as in case of full-scale actual foam structures it would not be realistic to perform LES or DNS. Therefore, as in most cases of turbulence modeling, it is a regular practice to implement the

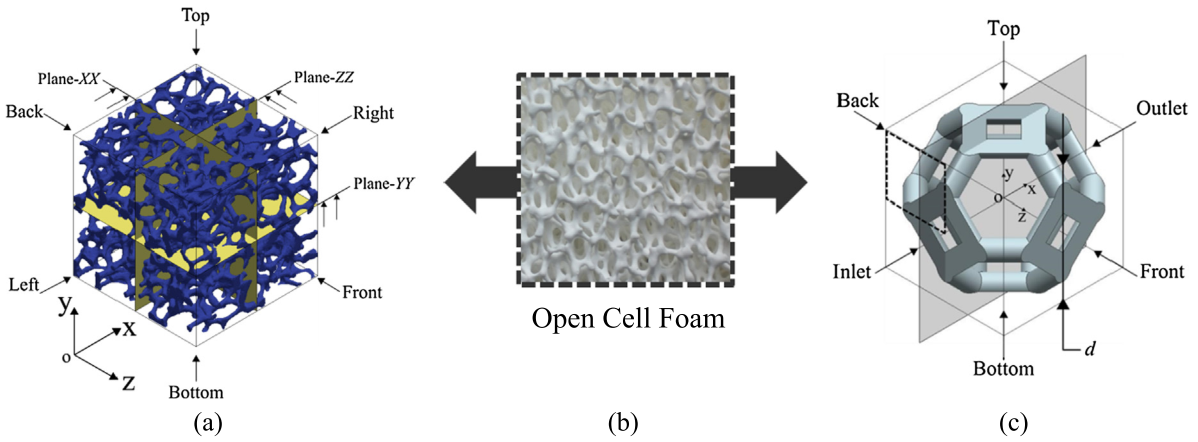

Fig. 1. Detailed and simplified representation of open cell foams.