International Journal of Thermal Sciences 98 (2015) 381–394

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Forced convection through open cell foams based on homogenization approach: Steady state analysis

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 19 March 2015
Received in revised form
30 June 2015
Accepted 10 July 2015
Available online 14 August 2015

Keywords:
Convection
Porous media
Thermophysical properties
Heat transfer coefficient

ABSTRACT

To investigate thermal transportation through open cell foams there are various microscopic and macroscopic numerical models along with their limitations available in the literature. The purpose of this study is to investigate the limitations of macroscopic models and to propose some reliable ideas as conclusion that can be used to overcome the existing limitations. Therefore, a combined experimental and numerical study is presented. The experimental study comprises of steady state forced convection experiments which involve three different regimes of heat transfer. Further, in macroscopic models these three regimes of heat transfer namely conduction, thermal dispersion and interstitial convection are governed by stagnant effective thermal conductivity, k_e, dispersion conductivity, k_d and volumetric heat transfer coefficient, h_v respectively. Moreover, the complex structure of the open cell foams is simplified into a rather realistic Kelvin cell model for the determination of k_e. The influence of the geometrical parameters such as pore diameter, d and foam porosity, ε is investigated by examining 10, 20 and 30 PPI (pores per inch) alumina foams for a porosity range of 0.79–0.87. The findings of this study reveal that it is important to consider both local thermal non-equilibrium (LTNE) and thermal dispersion together for improved analysis. Further, it is revealed that although with the above consideration, it is possible to exhibit the effect of geometrical parameters on each regime of heat transfer but the accuracy of the results predicted through macroscopic models remain under question as there is no basis to validate the results.

1. Introduction

Open cell foams have desirable geometrical characteristics that make them suitable for various applications such as filters for purifying molten metal alloys, thermal insulations and advanced catalytic burners. Secondly, for high temperature heat-exchange applications, the ability of ceramic foams to withstand high temperatures makes them a suitable choice. Hence, due to significant importance of the above mentioned applications, it is important to design these systems with better efficiency. In case of a typical steady state application setup, where to represent the heat source, a constant heat flux is applied across the boundaries of the two-phase system and to represent the heat sink, a certain fluid flow is developed through the system. In such a forced convection application setup the parameters that represent different regimes of heat transfer are stagnant effective thermal conductivity, k_e, dispersion conductivity, k_d and volumetric heat transfer coefficient, h_v. Our literature review suggests that there are two approaches to analyze a forced convection problem through open cell foams. The first one is the microscopic approach [1] conducted by numerically simulating the periodic flow through a suitable representative of the actual two-phase system. The second one is the macroscopic approach conducted by considering the two-phase system as a homogenized effective medium represented by integrated transport equations over the entire volume of the system. The second approach is further classified into one-equation model and two-equation model. The one-equation model [2,3] as shown in Eq. (1) is based on the assumption of local thermal equilibrium (LTE), which means there is significantly high convective heat interaction between the two phases that ultimately brings them ideally to the same temperature. It further signifies that one-equation model is suitable for solving those problems that has thermal dispersion as the dominant mode of heat transfer. Based on the previous work [3] Eq. (2) proposes a model for dispersion conductivity, k_d that

* Corresponding author.
E-mail addresses: digvijay.1431987@gmail.com, digvijay_1431987@yahoo.co.in (D. Vijay).

http://dx.doi.org/10.1016/j.ijthermalsci.2015.07.017
1290-0729/

governs the thermal dispersion. On the other side, the two-equation model is based on the assumption of local thermal non-equilibrium (LTNE) between the two phases, which leads to continuous convective heat interaction with the consideration of significant temperature difference existing between them.

$$\rho_f c_{pf} u_P \frac{\partial T}{\partial x} = \frac{\partial}{\partial y}\left((k_e + k_d)\frac{\partial T}{\partial y}\right) \quad (1)$$

$$k_d = C_d \rho_f c_{pf} u_P \sqrt{K} \quad (2)$$

where u_p is the local physical velocity, x is the longitudinal direction of flow and y is the transverse direction of applied heat-flux.

Having already mentioned the three regimes of heat transfer in case of a forced convection application setup, thermal dispersion is the first regime of heat transfer. The effect of thermal dispersion in case of a porous medium can be considered as an additional contribution of heat transfer through the fluid-phase more predominantly at high Reynolds numbers. As the fluid moves through the complex structure of the foam it disperses around the obstacles and remixes at the downstream while forming eddies [3]. In the presence of a temperature gradient at first this dispersion effect enhances the convective heat transfer from one phase to another phase and secondly, it enhances the overall molecular diffusion and bulk convection of heat through the fluid-phase in comparison with conduction through the same fluid-phase in stagnant conditions. While solving a forced convection problem by implementing a homogenized model, the enhanced convective heat transfer due to dispersion effect is a factor that is considered in terms of volumetric heat transfer coefficient, h_v but the enhanced molecular diffusion and bulk convection through the fluid-phase has to be considered separately. An additional factor that takes care of that is known as dispersion conductivity, k_d. Further, it is important to understand that dispersion conductivity is a fictitious quantity that appears in the homogenized models as a result of volume averaging of the actual microscopic momentum and energy equations. Considering that the effect of thermal dispersion is implicitly defined in a microscopic model as these models solve the actual velocity fields that consider the energy dissipation due to hydraulic mixing and eddy formation. Now as already mentioned that thermal dispersion in case of low porosity porous medium can be investigated by implementing homogenized one-equation model [4–9] and the analytical solutions are also available in the literature [6,10]. But in case of high porosity open cell foams to investigate the combined effect of all three regimes of heat transfer, one-equation model cannot be used as it does not include convective heat exchange between the two phases.

Subsequently, the second regime of heat transfer is conduction through the two-phase system. For solving thermal transport equations, it is important to analyze conduction through the two-phase system for determining stagnant effective thermal conductivity of each phase in the vicinity of the other phase. Most of the studies in the past whether experimental [11–13] or mathematical [14–16] have focused on the determination of stagnant effective thermal conductivity, k_e of the combined two-phase system. But none of the above mentioned studies have focused on determining the separate component of effective thermal conductivity of each phase. For studying forced convection through porous media some researchers [17–19] have implemented the effective thermal conductivity of each phase as shown in Eqs. (3) and (4), commonly known as parallel model. But this consideration has no geometrical basis and in some cases where there is significant difference between the bulk thermal conductivities k_s and k_f of the two phases, it does not predict the correct results [1]. Hence, knowing that the parallel model does not account for geometrical and flow characteristics of the two-phase system, Calmidi et al. [14,20] implemented an analytical model based on simplified two-dimensional hexagonal structure. Their analytical model predicts reasonable results for low temperature applications but still together with its requirements of validation through experimental data, it is solved for a rather simplified structure and it is not a true representative of the actual foam. Hence, it is important either to use the actual structures regenerated through CT scans or to use more realistic periodic structures such as Kelvin cell or Weaire-Phelan cell to represent the complex geometry of open cell foams.

$$k_{se} = (1-\varepsilon)k_s \quad (3)$$

$$k_{fe} = (\varepsilon)k_f \quad (4)$$

Finally, in conjunction with first two regimes of heat transfer, the third regime of heat transfer is interstitial convection between the two phases. Now from the application point of view various researchers such as Carpenter et al. [21], Mancin et al. [22] and Guarino et al. [23] experimentally investigated the combined effect of these three regimes of heat transfer. And to numerically investigate this coupled problem of conduction, convection and thermal dispersion, two-equation model was proposed. The two-equation model [24–26] is shown in Eqs. (5) and (6), where Eq. (5) represents the solid-phase energy equation and Eq. (6) represents the fluid-phase energy equation. Further, it is already mentioned that this model is based on the assumption of local thermal non-equilibrium (LTNE). This is a realistic assumption for high porosity open cell foams in comparison with low porosity packed bed porous media [20].

$$\frac{\partial}{\partial x}\left(k_{se}\frac{\partial T_s}{\partial x}\right) + \frac{\partial}{\partial y}\left(k_{se}\frac{\partial T_s}{\partial y}\right) = h_v\left(T_s - T_f\right) \quad (5)$$

$$\frac{\partial}{\partial x}\left(\left(k_{fe}+k_d\right)\frac{\partial T_f}{\partial x}\right) + \frac{\partial}{\partial y}\left(\left(k_{fe}+k_d\right)\frac{\partial T_f}{\partial y}\right) = \rho_f c_{pf} u_P \frac{\partial T_f}{\partial x} - h_v\left(T_s - T_f\right) \quad (6)$$

Lee et al. [27] determined the analytical solution for the above mentioned two-equation model by considering diffusion terms only in the transverse direction (y-direction of applied heat flux) without the consideration of thermal dispersion (i.e. $k_d = 0$). For conductivity ratio [$\kappa = (k_{fe}/k_{se})$], $\kappa >> 1$, they concluded that fluid-phase diffusion is dominant and for $\kappa << 1$ solid-phase diffusion is dominant. Although, the above mentioned study was conducted for low porosity porous media, but still a comparative analysis in case of forced convection of air (k_f = 0.0265 W/m K) through high porosity open cell metal foams verifies this theory. As Calmidi et al. [20] demonstrated for $\kappa << 1$, the case of forced convection of air through aluminum foams, the bulk of the heat is transferred directly to the solid-phase from the heating plates and from solid-phase to the fluid-phase through convection. Hence, the direct dispersion of heat from the heating plate to the fluid-phase is negligible. They also justified the LTNE assumption to be realistic for high porosity open cell foams. But in case of forced convection of water through aluminum foams, Hunt et al. [3] and Calmidi et al. [20] both concluded for a similar case of $\kappa << 1$ that direct dispersion effects are significantly high and account for most of the heat transfer. Nevertheless, as already mentioned that Calmidi et al. [20] determined h_v in case of LTNE, but by neglecting thermal dispersion i.e. by considering k_d as zero. But by using a similar theory of considering k_d as zero for a similar combination of high porosity aluminum foams and air, Garrity et al. [17] were not able to