Developing thermal flow in open-cell foams

M. Iasiello [a, *], S. Cunsolo [a, b], N. Bianco [a], W.K.S. Chiu [c], V. Naso [a]

[a] Dipartimento di Ingegneria Industriale, Università degli Studi di Napoli Federico II, P.le Tecchio 80, 80125, Napoli, Italy
[b] LaMCoS, INSA-Lyon, UMR CNRS 5259, 18 — 20 Rue des Science, F-69621, Villeurbanne, France
[c] Department of Mechanical Engineering, University of Connecticut, 191 Auditorium Road, 06269, Storrs, CT, USA

A R T I C L E   I N F O

A B S T R A C T

Predicting heat transfer is a primary task in the design of open-cell foams. When a Local Thermal Non Equilibrium (LTNE) model is employed, convection heat transfer between the solid and fluid phases is considered, and a volumetric heat transfer coefficient needs to be defined. Some recent studies pointed out that the effects of developing convection heat transfer between the fluid and the solid in a foam are to be taken into account. The developing thermal flow of air through an open-cell foam, with a uniform heat flux solid/fluid boundary condition, is investigated numerically in this paper. The geometry is modeled with reference to Kelvin's tetrakaidecahedron foam model. A correlation among the porosity, the cell diameter and the ratio of heat transfer surface to volume is derived. Three regions are identified along the flow direction: an impingement region, a thermally developing region and a thermally developed region. Dimensional and dimensionless convection heat transfer coefficients have been predicted numerically as a function of the axial coordinate of the foam, for different values of the Reynolds number and the porosity. A correlation is presented among the predicted values of the volumetric Nusselt number, the porosity, and the Reynolds number in the thermally developed region, which is in good agreement with experimental data and numerical predictions by other authors. Finally, the analysis of the convection heat transfer through a single foam cell, at a local pore-scale, is presented.

1. Introduction

Open-cell foams have recently become interesting in applications where heat transfer plays a primary role, because of their tortuosity and high heat transfer surface to volume ratio. The performance of heat exchangers, microelectronic heat sinks, volumetric solar receivers and burners can be improved by using open-cell foams. Their use requires a good knowledge of the thermo-fluid-dynamic characteristics. This is still a challenging task because of the complex geometry and of the very small scales.

Velocity and temperature fields in an open cell foam have been investigated experimentally, analytically and numerically. A far as the latter technique is concerned, governing equations are usually written with the Volume Averaging Technique (VAT) [1]. Besides, the temperature field can be predicted by using the Local Thermal Equilibrium (LTE) model [2], that assumes the solid and the fluid phases in thermal equilibrium, or with the Local Thermal Non-Equilibrium (LTNE) model [3], that doesn't make thermal

equilibrium assumption. When the LTNE model is used, an interfacial volumetric heat transfer coefficient needs to be defined to close the volume-averaged governing equations. The volumetric heat transfer coefficient is defined as the product of the convection heat transfer coefficient times the heat transfer surface to volume ratio. It can be either evaluated experimentally or predicted with a model. Dietrich [4] used an inverse method to obtain the volumetric heat transfer coefficients from measurements on Al_2O_3 and Cordierite open-cell foams. Mancin et al. [5] evaluated interfacial heat transfer coefficients by combining measurements in twenty-one aluminum foam samples with a parameter that considered the foam as an extended surface enhancing the heat transfer surface area [6].

Because of the complex geometry of a foam, the accuracy of predictions is strongly affected by the geometry of the model. Geometry can be constructed with the support of X-ray Computed Tomography (XCT), developed on real foam samples [7,8]. Many models, based on different geometries, have been proposed. One of the first models, proposed by Lu et al. [9], used simple cubic unit cells based on heated slender cylinders. More recently, models for open-cell foams are based on geometries as close as possible

* Corresponding author.
E-mail address: marcello.iasiello@unina.it (M. Iasiello).

Nomenclature

a, b, c, e	regression coefficients
d	diameter (m)
h	interfacial convection heat transfer coefficient (W/m^2 K)
l	strut average length (m)
L	entrance length (m)
$\boldsymbol{n}$	normal vector
Nu	Nusselt number
p	pressure (Pa)
PPI	Pores Per Inch (0.0254/m)
$\boldsymbol{q}$	heat flux (W/m^2)
Re	Reynolds number
S	heat transfer surface (m^2)
t	strut average thickness (m)
T	temperature (K)
$\boldsymbol{u}$	velocity vector (m/s)
V	volume (m^3)
x, y, z	rectangular coordinates (m)

Greek letters

α	thermal diffusivity (m^2/s)
ε	porosity
λ	thermal conductivity (W/m K)
μ	dynamic viscosity (kg/m s)
ν	kinematic viscosity (m^2/s)
ρ	density (kg/m^3)

Superscripts

*	Normalized

Subscripts

0	ambient
c	cell
f	fluid
p	pore
s	solid
T	thermally developed
v	volumetric

Others

| |	modulus
< >	average

similar to those of real foams, such as the Plateau structure [10], the Kelvin structure [11] and the Weaire-Phelan structure [12]. Kopanidis et al. [13] investigated heat transfer characteristics of an open-cell foam by using the Weaire-Phelan structure. Kelvin structure was used to calculate interfacial volumetric heat transfer coefficients in Refs. [14,15]. A comparison between predictions based on Kelvin structure and on tomography reconstructed structures was carried out by Iasiello et al. [16].

The effects of the fluid impingement on the foam and of the developing temperature field occurring in the entrance region of foam are very seldom accounted for and most of the studies on convection heat transfer in open-cell foams are based on length averaged values. As a matter of fact, it has to be noticed that, whichever the porous medium, the thermal development of the flow has been scarcely investigated and however with reference to continuous models [17–19]. Wu et al. [15] reported that it is more appropriate to define a local heat transfer coefficient, instead of an averaged one, to account for the variation in the heat transfer coefficient in the flow direction. By imposing a constant surface temperature at the solid/fluid interface, they predicted heat transfer coefficients for open-cell foams based on the Kelvin structure, providing a correlation among process parameters for thermally developed flow. Suleiman and Dukhan [20] simulated the flow of air through a very large computational domain based on arrays of Kelvin structures, heated from below with a uniform wall heat flux. They found that the flow became thermally developed downstream of a certain number of cells. An experimental analysis of thermal developing effects under uniform wall heat flux, in water flowing through aluminum foam samples, was carried out by Dukhan et al. [21]. They concluded that the thermal entrance region cannot be neglected.

The developing thermal flow of air through an open-cell foam, with a uniform heat flux solid/fluid boundary condition, is investigated numerically in this paper. The open-cell foam is generated, according with the Kelvin structure, by means of a computational domain that takes into account the developing thermal flow. Three regions are identified along the flow direction: an impingement region, a thermally developing region, and a thermally developed

region. Simulations are carried out under a uniform solid/fluid boundary heat flux condition, in order to evaluate convection heat transfer coefficients along the flow direction, for different values of the fluid velocity and of the foam porosity. A correlation is presented among the predicted values of the volumetric Nusselt number, the porosity and the Reynolds number in the thermally developed region. The convection heat transfer through a single foam cell, at a local pore-scale, is analyzed.

2. Geometry generation

Which structure best represents a foam is a long-standing problem. Since it has often been proposed as a minimal surface problem, the target is to find which is the space-filling arrangement of equal volume cells that minimizes the surface area per unit volume, complying with Plateau's laws [10]. Kelvin [11] proposed a tetrakaidecahedron unit cell geometry, with slightly curved faces. More than a century later, Weaire and Phelan [12] proposed a unit cell with a 0.3% surface area per unit volume less than Kelvin's geometry. It is made up by eight equal volume cells: six cells with twelve pentagonal and two hexagonal faces, and two pentagonal dodecahedric cells. Since the Kelvin structure is fairly simpler than the Weaire-Phelan structure, it is used in the present study. Differences in heat transfer coefficients and pressure drop between the two models were investigated by Cunsolo et al. [22], who reported negligible differences.

The procedure used to generate the geometry is described in the following. The Kelvin unit cell foam is initially generated with the free-to-use software Surface Evolver [23], which is a program for the modeling of surfaces shaped by various forces and constraints. The surface energy of the unit cell foam is iteratively minimized using a constrained gradient-descent method and the porosity of the foam is imposed in this phase of the generation procedure. A generated Kelvin unit cell with a 0.87 porosity of the foam is sketched in Fig. 1. The foam is then exported into COMSOL Multiphysics, where the unit cell can be scaled or duplicated. Since the aim of the present study is to investigate developing effects, each cell is duplicated 20 times along the flow direction, and a bounding