Experimental Thermal and Fluid Science 77 (2016) 28–37

Contents lists available at ScienceDirect

Experimental Thermal and Fluid Science

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

An experimental study on open cell metal foam as extended heat transfer surface

Tisha Dixit, Indranil Ghosh *

Cryogenic Engineering Centre, Indian Institute of Technology, Kharagpur 721 302, India

ARTICLE INFO

Article history:
Received 29 September 2015
Received in revised form 16 February 2016
Accepted 11 April 2016
Available online 11 April 2016

Keywords:
Metal foam
Extended surface
Double wall
Single Wall
Thermo-hydraulic behaviour

ABSTRACT

Experiments have been performed with high porosity, open cell copper foam blocks sandwiched between plates at constant temperature. Convective air passing through the foam carries the heat being conducted through the plate and foam matrix. Foam acts as an extended heat transfer surface adhered to the primary one (plate). Air, entering the foam at uniform temperature, leaves with a spatial temperature variation. Mathematical modelling with the repetitive "simple cubic" structure representing metal foam has been used to explain the exit air temperature variations. Governing foam heat transfer equations and the resulting solution have strong resemblances with those of the conventional fins with appropriate corrections required for the interconnected foam structure. Good agreement between the experimental data and theoretical prediction has been observed. The proposed model is simple yet effective. One can avoid rigorous numerical calculations necessary to analyse heat transfer in metal foam used as extended heat transfer surfaces in many engineering applications such as heat sinks and heat exchangers.

1. Introduction

High porosity open-cell metal foam is a relatively recent development. The characteristic features like high surface area density, low weight and ability to mix fluids have accredited metal foam a potential choice in a variety of thermal engineering applications [1–10]. Some of the applications of this novel material can be found in plate-foam type recuperators [1–3], regenerators [4], heat sinks [6], tubular reactors [8], condensers [5–9] and fuel cells [10,11], to name a few.

The knowledge of design and simulation, for the obvious reason, is an essential requirement for using open porous metal foam in thermal devices and applications. This in turn needs reliable experimental data of its physical [12] and thermo-hydraulic properties [13–21]. Appropriate mathematical modelling representing the actual system is also crucial to keep the experimental endeavours within reasonable bound [22–28]. Recently, Zhao [29] has reviewed different published works in reference to the thermal transport mechanisms in high porosity open-cell metal foams.

Metal foam owing to its highly interconnected and intricate geometry renders heat transfer modelling quite difficult. Keeping the actual metal foam structure intact, computational simulation [28,30–33] has been applied to study the heat transfer in porous media. On the other hand, simplified assumptions have been made in terms of its geometry or compromise has been made with the flow regime. When the velocity of convective flow through the porous body is low enough, assumption of local thermal equilibrium between metal and fluid renders considerable simplicity in foam modelling [34,35]. Based on this assumption Dukhan et al. [34] have proposed 'lumped parameter' engineering model for analysing heat transfer in porous metal. Later, Dukhan et al. [35] have reported heat transfer in metal foam attached to a plate under constant heat flux condition. The validity of this hypothesis, however, is restricted to low Reynolds number applications. Alternatively, assumption of local thermal non-equilibrium between the convective fluid and the foam struts corresponds to more generalised and realistic representation [36–38].

Calmidi and Mahajan [13] have conducted experiments with high porosity aluminium foam. They have correlated the data with numerical model using 'semi-empirical volume-averaged' governing equations. Modifying the heat transfer coefficient for cylinders in crossflow with the experimental data, they have obtained a correlation for the interfacial heat transfer coefficient for metal foam. Kim et al. [14] have performed experimental study to investigate the effect of porous fin on plate foam heat exchanger and compared the performance with that of a unit with louvered fin. They have considered foam as local volume averaged continuous medium. According to these authors, metal foam heat transfer

---

* Corresponding author.
E-mail addresses: tisha8889@iitkgp.ac.in (T. Dixit), indranil@hijli.iitkgp.ernet.in (I. Ghosh).

http://dx.doi.org/10.1016/j.expthermflusci.2016.04.010
0894-1777/

Nomenclature

A_c	strut cross-sectional area
$C_{p,a}$	specific heat
d_f	strut diameter
d_p	pore size
h	interstitial heat transfer coefficient
H	height of foam
k_f	thermal conductivity of foam material
L	length of foam
Nu	Nusselt number
P	strut perimeter
PPI	pores per inch
Pr	Prandtl number
q	heat transfer
Re	Reynolds number
$T_{a,in}$	temperature of inlet air
$T_{a,out}$	temperature of outlet air
T_{w1}, T_{w2}	wall temperatures
v_a	velocity of air
W	width of foam

Greek symbols

η	fin efficiency of half strut
ε	foam porosity
θ	temperature differential between foam filament and convective fluid
ρ_a	density of air

performance is almost similar to that of louvered fin; however, the latter depicts a slightly better performance in terms of pressure drop. Hwang et al. [15] have experimentally determined the heat transfer and flow friction characteristics using transient single-blow technique. Empirical correlations of pore Nusselt number for a specific length of foam have been proposed by the authors. They have found volumetric heat transfer coefficient to increase with decreasing porosity [15]. Experimental measurement of pressure drop and development of empirical correlation have been reported by Dukhan et al. [17]. Dukhan and Chen [18] have performed thermal measurements in metal foam subjected to constant heat flux. Data reduction has been achieved with a two-dimensional, simplified model assuming local thermal equilibrium between the foam and convective fluid. Salas and Waas [19] have experimentally determined the effect of foam thickness on convective heat transfer. The authors have used model based on finite element method for analysing their experimental data. Tzeng and Jeng [20] have performed experimental and numerical studies in uncompressed/compressed metal foams. They have concluded that the uncompressed sample has a larger Nusselt number than the compressed one. Kurtbas and Celik [21] have performed experiments to investigate the mixed convective flow through metal foam of different pore densities sandwiched between bounding walls subjected to constant heat flux. They have used both Reynolds number and Richardson number to present average and local Nusselt number.

It is noticeable from the literature review that upcoming technologies revolving around metal foams are practically possible. However, the design and simulation of metal foam products such as foam heat sinks and heat exchangers is yet to be streamlined. Assumption of local thermal equilibrium between the metal foam matrix and the convective fluid [34] may not be acceptable for high velocity convective fluids. Experimental studies of metal foam subjected to constant heat flux condition are common [18,21,35]. On the contrary, porous material in many engineering applications, such as in plate foam heat exchangers, is used as an extended heat transfer surface attached to a plate or surface at constant temperature. Heat transfer analysis of such unit rectangular block can be essentially used to handle rating and sizing problem [39], but the complicated geometry of metal foam renders the analysis difficult. In order to avoid rigorous analytical and numerical computations for engineering applications, often it might be possible to apply simplifying assumptions without violating the basic physical laws. Repetitive simple cubic structure of open porous foam [36,37,40–43] has been found effective in understanding the heat transfer mechanism. Nevertheless, in the simple cubic model, researchers have often neglected the interconnectivity of the foam structures transforming it into flow over bundle of non-interconnected, straight tubes. In the simple cubic foam model proposed by Ghosh [27,37] heat transfer through individual struts has been considered explicitly in a purely 3D mode. This model can give a better insight of the actual heat transfer in porous metal besides making one aware of the risks involved in its use [43]. In this article, the proposed microscopic thermal model [37] has been used to determine the heat transfer rate from metal foams when one or both ends are maintained at constant temperature. This theoretical study has been followed by experimental testing of copper foams, having varying pore densities, with air as the convective fluid medium. The measured thermo-hydraulic characteristics have been employed to validate the developed mathematical model.

2. Theory

A graphic view of the open porous metal foam pieces attached to a solid plate has been shown in Fig. 1.

In order to construct a unit of this type, one may have to join several pieces of metal foam with dimensions ($W.H.l$) on a plate of dimensions ($W.L$). Thickness of the solid plate is much less than the height of metal foam ($t \ll H$). The porous elements can be thought as secondary (or extended) heat transfer surface area attached to the primary solid surface (plate). In a steady state condition, conduction of heat occurs (in the x-direction) from the hot surface to the metal foam, while the convective fluid flow (in the z-direction) takes away this heat. As a result, there is a continuous change in the temperature of convective fluid as it passes through the porous media. This convective heat transfer also develops temperature gradient in the porous solid as a function of foam length (x). While the temperature of the fluid entering foam matrix at $z = 0$ is known, the objective is to find an expression of the fluid exit temperature (at $z = L$) in a differential element Δx at a distance x, measured from the hot plate, with the help of the theory developed in Ghosh [37].

Avoiding the computational complications of analysing the real structure of metal foams, a repetitive cubic structure model has been used by many researchers [36,37,40–43]. In this model, shape of the unit cell is reformed keeping other characteristic features like pore density, porosity or surface area density unaltered. The unit foam cell, with this assumption, gets transformed into a cube of slender tubes of diameter d_f and length d_p, rendering substantial simplification in foam modelling. Assuming that the uniformly distributed, equal-sized cubic cells are attached to a plate at constant temperature (T_{w1}) and convective fluid is flowing over the struts, it