Applied Thermal Engineering 137 (2018) 532–544

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

Simulation intricacies of open-cell metal foam fin subjected to convective flow

Tisha Dixit, Indranil Ghosh*

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

HIGHLIGHTS

- Digitization of 10 PPI and 90.8% aluminium foam through X-ray μ-CT.
- Simulation of convective flow through extracted section of digitized foam.
- Listing of existing uncertainties in mapping computational and experimental results.
- Heat and flow intricacies visually captured in close vicinity of foam matrix.
- Comparison of cubic-cell and real topology based thermal characteristics.

ARTICLE INFO

Keywords:
Aluminium foam
X-ray μ-CT
Computational simulation
Uncertainties

ABSTRACT

An exact 3D digital image of 10 PPI open-cell metal foam matrix used in this study has been generated through X-ray micro-computed tomography. The precise foam geometrical model, in combination with the commercial computational software, facilitates synthesizing typical thermal applications of metal foam subjected to convective fluid flow and creates unique opportunity of probing some of the finer fluid flow aspects usually beyond the scope of most of the experimental investigations. Simulation of fluid flow through 10 PPI open porous metal foam subjected to constant terminal temperature has been carried out and validated a priori with the experimental data. The convective flow simulation, being computationally intensive, has been carried out in a small piece of metal foam to eventually extrapolate the data over the original length. The practical limitations of foam modelling leading to the deviations between experimental data and the predicted results have been explained in details. Subsequently, the temperature and flow fields have been probed to understand the complicated flow physics around the metal foam strut. This study gives a better insight of the state of the art research and development in the field of high porosity open-cell metal foam. The computational predictions have also been compared at length with the analytical outcomes based on the *simple cubic* model of metal foam. This work, in all, provides the end-user an overview of the associated intricacies while considering the computational, experimental and *simplified* models for open-cell metal foams when used as extended heat transfer surfaces or fins.

1. Introduction

One of the promising extended heat transfer surface emerged in the last few years is the high porosity (>90%) open-cell metal foam [1–2]. Characterized by the distinctive framework of inter-connected high conductivity metallic ligaments, these generate a tortuous path for the fluid to flow. The porous matrix provides a high interfacial surface area density in addition to imparting it light weightedness. Owing to these characteristics, high porosity open-cell metal foams have been profoundly used in several heat transfer applications such as heat sinks [3–5], recuperative and regenerative heat exchangers [6–9], release and storage of heat [10–12], volumetric solar receivers [13–14], fuel cells [15], and impregnation of phase change materials [10,16–17], to name a few.

Practical utility of high porosity open-cell metal foam calls for sound knowledge of their thermal transport properties [18]. Experimental techniques are the best means to obtain reliable data [4,14,19–26]. The alternative methods of evaluation include analytical studies or computational simulations [27–39]. Integral part of both these techniques is the geometrical modeling of the complex metal foam structure. Literatures reveal that three alternatives for metal foam models are in existence (a) simplified geometries like mutually

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

https://doi.org/10.1016/j.applthermaleng.2018.04.011
Received 3 January 2018; Received in revised form 1 April 2018; Accepted 3 April 2018
Available online 04 April 2018
1359-4311/

Nomenclature

m	fluid mass flow rate (kg/s)
PPI	pores per inch
$T_{a,in}$	inlet temperature of fluid (K)
$T_{a,out}$	outlet temperature of fluid (K)
T_b	temperature of foam base (K)

orthogonal square ducts [40] and cubic-cells [20,27,29] (b) polyhedral structures such as Kelvin and Weaire-Phelan cells [32,41] (c) digitized *real* foam topology [42]. While the simplified geometries can yield analytical solution, analysis involving other two models demands use of specialized computational software. However, quantification of the loss in precision using the former case for solving practical engineering problems presently remains unidentified.

Amongst the aforementioned models, the three-dimensional (3D) replica of the actual foam is the best means to represent open-cell metal foam structure. Use of the digitized *real* topology of high porosity open-cell metal foam has materialized only in the recent years given that it is bounded by the requirement of high-end computational facilities. The handful of available research articles [43–55] have been enumerated in Table 1. It can be observed that the demand of large system memory requirements have imposed restrictions of the domain size that can be simulated.

Computational simulation of convective flow over actual three-dimensional foam structure reproduced using X-ray μ-CT has been presented in this article. One end of the metallic foam is maintained at a constant temperature mimicking the thermal condition of extended heat transfer surface. Earlier, the authors have carried out experiments and also developed analytical model based on *simplified model* to explain the experimental outcomes [25,38]. An end-user opting for *simplified model* based simulation, may also be interested to know the margin by which his results are offset in comparison to those obtained through computationally intensive numerical scheme using digitized real (μ-CT) foam model. The numerical simulation presented here focuses on the relative merits and demerits of different techniques. Furthermore, the various physical phenomena which cannot be obtained experimentally or modeled easily have been visually captured in this work.

2. X-ray μ-CT structure

Open-cell aluminium foam (*Al 10*) with 90.8% porosity and 10PPIpore density has been selected for this study. The dimensions of this sample are 75 mm × 75 mm × 6.35 mm. In order to digitally duplicate the actual foam structure an in-house X-ray μ-CT facility has been employed. The actual foam sample and its digital image have been shown in Fig. 1. However, this μ-CT three-dimensional model is not compatible with most of simulation software. The three-dimensional structure needs to be converted to a CAD importable file to be employed in any simulation software. This has been done with the help of CAD design software “SolidWorks” and open source software “MeshLab”. However, the *Al 10* digitized topology consists of nearly 1.33 million surfaces. This is beyond the handling capacity of the laboratory system memory (Intel core i7-6700 processor @ 3.40 GHz, 31.9 GB usable RAM, 64-bit operating system). Consequently, after a number of trials, elemental section of 5.8 mm × 4.6 mm × 15.8 mm extracted from the original 3D foam geometry (Fig. 1b) has been found to generate successful simulations.

3. Modeling environment

The schematic of the physical model, describing convective fluid flow through metal foam as extended heat surface, is shown in Fig. 2. The fin base of the metal foam is maintained at constant temperature (T_b) and the air flows through the porous matrix picking up the heat conducted via the porous fin. Simulation of convective flow over metal foam, in the steady state condition, has been carried out using the *Conjugate Heat Transfer* module of the *COMSOL Multiphysics* software. This module couples the fluid flow along with heat transfer interface. Snapshot of the modeling environment has been given in Fig. 3.

Once the foam element has been imported, a block is created to form the fluid domain. Fluid domain at the exit has been kept about ten times the dimension of foam in flow direction $x = 0$ so that pressure and velocity fluctuations can subside [2,50]. Fluid domain is taken as air while the solid foam geometry is assigned aluminium-6061. Both the materials are part of the in-built material library available with *COMSOL Multiphysics* which takes into account the temperature dependence of material and fluid properties. It may be emphasized from the graphics window in Fig. 3 that the fluid flow occurs in + x direction entering the domain at, while the temperature gradient within foam geometry has been imposed along the $-z$ direction.

In order to successfully implement the physical model depicted in Fig. 2, the appropriate boundary conditions, pertinent to steady state simulation are mandatory. Since the *Conjugate Heat Transfer* module of the *COMSOL Multiphysics* is a combination of heat transfer and fluid flow modules, separate boundary conditions have been imposed. While the conditions related to fluid flow have been specified in the *Laminar Flow* module, the thermal conditions are given in the *Heat Transfer in Solids* module as summarized in Table 2 along with the corresponding mathematical formulas. Following are the governing equations for the *Laminar Flow* module,

$$\rho_f(u\cdot\nabla)u = \nabla\cdot[-p\,\mathrm{I} + \mu(\nabla u + (\nabla u)^{\mathrm{T}})] + F$$
$$\rho_f\nabla\cdot(u) = 0 \tag{1}$$

where, ρ_f is the fluid density, μ is the fluid viscosity, p is the fluid pressure and F is the volume force. Within the *Heat Transfer in Solids* module, the governing equations for the solid and fluid domains are given by Eqs. (2) and (3) respectively,

$$\rho_s C_{p,s} u\cdot\nabla T = \nabla\cdot(k_s\nabla T) + Q \tag{2}$$

$$\rho_f C_{p,f} u\cdot\nabla T = \nabla\cdot(k_f\nabla T) + Q + Q_{vd} + Q_p \tag{3}$$

where, ρ_s is the solid material density, $C_{p,s}$ is the solid material heat capacity at constant pressure, k_s is the solid material thermal conductivity, Q is the heat source, $C_{p,f}$ is the fluid heat capacity at constant pressure, k_f is the fluid thermal conductivity, Q_{vd} is the viscous dissipation and Q_p is the point heat source.

Complexity in the foam geometry poses added difficulty while meshing the computational domains. When a block is created around the foam geometry, *COMSOL* by default subtracts the solid volume to obtain the fluid domain. The mesh elements are formed at the solid-fluid interface. However, meshing can be challenging, if any surface at this interface becomes very narrow. Due to compatibility in resolution between μ-CT post-processing software (*VGStudio MAX 2.2*), *SolidWorks* and *MeshLab,* few surfaces of hairline thickness could be observed. Such surfaces at the solid-fluid interface could not be meshed even with extremely fine elements. This is precisely the problem encountered while meshing. Consequently, in order to repair these surfaces, the foam geometry had to be shuffled between *MeshLab* and *SolidWorks*. Finally, successful meshing of the computational domains could be done with tetrahedral elements. The number of mesh elements, obtained by adequately satisfying the grid independence test conducted additionally, for *Al 10* PPI foam is 1360506. Once meshing has been done, the *COMSOL* in-built direct stationary solver (for steady state simulation) proceeds to solve the Navier-Stokes, continuity and energy equations in the solid and fluid domains.