2013 NDIA GROUND VEHICLE SYSTEMS ENGINEERING AND TECHNOLOGY SYMPOSIUM
MODELING & SIMULATION, TESTING AND VALIDATION (MSTV) MINI-SYMPOSIUM
AUGUST 21-22, 2013 - TROY, MICHIGAN

SIMULATION AND VALIDATION OF METAL-FOAM COOLING DESIGN

Nihad Dukhan, PhD
Department of Mechanical Engineering
University of Detroit Mercy
Detroit, MI

AHMED S. SULEIMAN
Department of Mechanical Engineering
University of Detroit Mercy
Detroit, MI

MUNTADHER A. AL-RAMMAHI
Department of Mechanical Engineering
University of Detroit Mercy
Detroit, MI

ABSTRACT

The open-cell metal foams have an internal structure is a web of connected ligaments. Foams are made from pure or alloys of aluminum, nickel, steel, magnesium, titanium and copper. In addition to being light weight, the foams exhibit excellent stiffness-to-weight ratios. Some foams can be resilient materials in harsh environments and have high impact resistance. The foams have high conductivities and large surface area per unit volume. All of these attributes make metal foam an attractive core for heat exchange. For example, cooling of power electronics and for thermal management of ground vehicles can employ metal-foam designs. Numerical simulation of convection heat transfer due to airflow inside commercial aluminum foam is conducted in a commercial numerical package. For validation, actual air temperatures were locally measured inside heated commercial aluminum foam, and cooled by forced air flow using a specially-developed technique. Good agreement between the modeling and experimental results is obtained.

INTRODUCTION

Metal foams are an emerging class of porous materials for engineering applications in various sectors including the defense industry. The open-cell foams are (solid) sponge-like materials with an internal structure that is similar to a web composed of connected ligaments that define a relatively repeated, and on average uniform, 3-dimensional open-cell structure. Currently the metallurgical and powder metallurgy routes are the most common producing techniques. Foams are also produced by sintering and casting. Other metals of the widely-produced metal foams include pure or alloys of aluminum, nickel, steel, magnesium, titanium and copper.

The ratio of the void space to the total volume of a piece of foam is called the porosity. Metal foams typically have very high porosity (above 90%), which makes them very light materials. With this the foams exhibit excellent stiffness-to-weight ratios. Depending on the mother metal, the foams can be resilient materials in harsh environments, have high impact resistance and crash-worthiness. Among the important characteristics of foams are the relatively high conductivities of the solid phase and the large surface area per unit volume. This is in addition to the vigorous mixing of the flowing fluid due to the internal structure of these materials, which enhances convection between the solid and the fluid. All of these attributes make metal foam a very attractive core for many engineering applications, e.g. heat exchange, filtration, reactors and light-weight components. Applications of metal foams are sought in aerospace and military systems. Cooling of power electronics and compact heat exchange systems based on metal foams are prime candidates for thermal management of ground vehicles.

The internal morphology of open-pore foams is complex and intrinsically random. Exact solutions of the complete transport equations are virtually impossible [1, 2]. Researchers have solved a small number of simplified cases of the governing equations, and relied heavily on numerical modeling. Calmidi and Mahajan [3] numerically studied forced convection of air flow in aluminum foam. The solid and fluid temperatures decayed gradually as the distance from the heated wall increased. Angirasa [4] numerically studied convection heat transfer due to water flow in metal

foam heat dissipaters. He invoked local thermal equilibrium, which is not physically valid due to convection between the solid ligaments of the foam and water resulting from a temperature difference between the two.

The rather complex and intrinsically random architecture of the foam is extremely difficult to capture exactly. Numerous studies have attempted to model the geometry of open-cell metal foam by defining a representative elementary volume (REV) that captures the relevant characteristics of these materials for fluid flow and heat transfer. du Plessis et al. [5] represented the foams by a set of rectangular prisms and Boomsma et al. [6] represented the cell of the foam by a tetrakaidecahedron, and used a 'periodic' cluster that consisted of eight such cells.

Krishnan et al. [7] simulated flow and heat transfer in open-cell metal foam using a single ideal body center cube (BCC) unit cell. They assumed a fully-developed flow and periodicity. Karimian and Straatman [8] used a unit cell based on interconnected sphere-centered cubes to represent the internal structure of foam. For their computational domain, they used two such cells in the flow direction, and assumed fully-developed laminar flow. There was no direct comparison to experimental results.

[9] Bai and Chung (2011) used a sphere-center Kelvin structure to simulate fluid flow in metal foam using a single unit cell. Horneber et al. [10] used a single and multiple Kelvin cells to simulate flow in a metal-foam reactor. Kopanidis et al. [11] carried out numerical simulations for the conjugate flow and heat transfer in open-cell metal foam. They used a cell geometry that was obtained by minimizing surface energy electronically and then by cleaning manually. The computational domain consisted of 10 cells. For validation, comparisons to experimental pressure-drop data from the literature were made.

In this paper, we use an ideal unit cell geometrical model to numerically simulate convection heat transfer due to airflow inside commercial aluminum foam. The non-thermal-equilibrium energy equations are solved directly, and temperature fields are obtained using a commercial numerical package. The details of the modeling process are given in this paper.

Direct comparisons to experimental values of the fluid and solid temperatures inside the foam are lacking in [12-14], and seem to be non-existent in the literature of heat transfer in metal foam. In a recent comprehensive review, Zhao [15] asserted that there is a lack of reliable experimental heat transfer data for open-cell metal foam.

In this study, direct comparisons with experimental data are carried out. Actual air temperatures were locally measured inside commercial aluminum foam cylinder heated at the wall by a constant heat flux, and cooled by forced air flow. The specially-developed experimental technique for such measurements is described.

GEOMETRICAL MODELING AND SIMULATION

The actual structure of commercial open-cell aluminum foam is photographed in Fig. 1. There is an approximately spherical void space (called a cell), surrounded by solid ligaments that connect to form open windows- each called a pore. On average this basic structure is the same and repeats to make up the foam. For simulation purposes, this geometry is idealized using a unit geometrical cell that should resemble the actual foam's cell as much as possible.

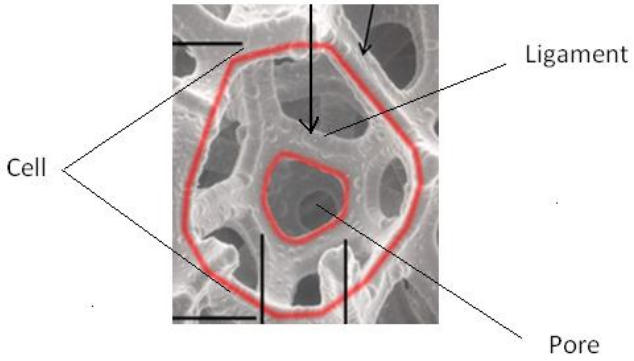

Figure 1: Typical internal structure of open-cell metal foam

Consider 14 equal spheres having a radius R. Let another equal sphere be contained in a cube of side length a1. Let 14 spheres initially surround the central sphere (and intersecting the cube at its faces), and then be subtracted from the cube, as shown in Fig. 2. The distance between the centers of the central sphere and a polar sphere after overlapping can be adjusted to be equal to a1. Let a2 be center-to-center distance between the central sphere and a quadrant sphere, also after some overlapping.

The distance a_1 can be related to actual foam's morphology, i.e., the number of pores per inch (ppi), which is a common industrial designation. Open-cell with10-ppi commercial aluminum foam on average has 5 central cells in one linear inch. Through geometrical reasoning, the porosity of the unit cell is given by the relation

$$\epsilon = \frac{1}{a_1^3}\left[\frac{8}{3}\pi R^3 - 6\frac{1}{3}\pi\left(R-\frac{a_1}{2}\right)^2\left(2R+\frac{a_1}{2}\right) - 16\frac{1}{3}\pi\left(R-\frac{a_1\sqrt{3}}{4}\right)^2\left(2R+\frac{a_1\sqrt{3}}{4}\right)\right] \quad (1)$$

Since a_1 is linked to the ppi of actual foam, Eq. (1) has only two unknowns, the porosity ε and cell radius R, and thus it can be solved by trial and error. It should be noted

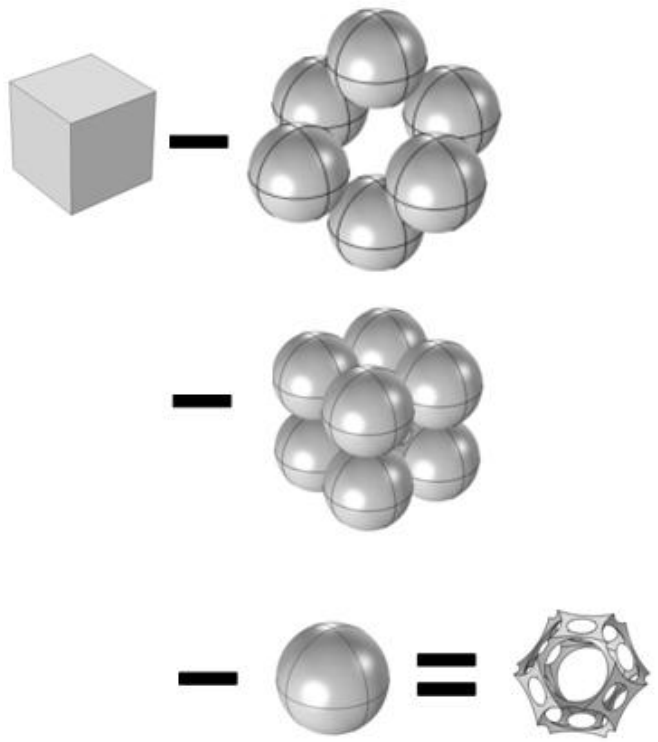

Figure 2: Steps of generating the idealized unit cell: Last step is subtraction of central sphere

that the steps of Fig. 2 for obtaining the idealized cell. geometry are performed in a Computer-Aided-Design (CAD) package. The idealized cell geometry has been known to have minimum surface area per unit volume [7]. In addition, the geometry produced a cell diameter and a pore diameter within 3.3% and 15.0% of averaged measured values for commercial aluminum foam, respectively [16].

To represent the metal foam structure, a matrix of many unit cells having 94% porosity and 20 PPI, was placed inside a rectangular channel. This matrix was subtracted from a rectangle that contained it, in order to get the fluid (air) domain as shown in Fig. 3. Air was used as the working fluid (with density of 1.2 kg/m^3 and a viscosity of 1.814x10^{-5} Pa.s). For simulating heat transfer through the foam, the commercial software COMSOL was used. The Navier-Stokes equations were directly solved for the velocity field along with the two equation model for the heat transfer. The solution was given at the pore level.

The computational domain was meshed using tetrahedral elements (3D). A finer mesh was applied to areas on the porous cell surface to account for the large gradients that occur in these areas. Mesh independence was established to ensure that the solution was independent of the mesh size. To confirm mesh independence, the same computational domain was spatially discretized using three meshes of varying refinement.

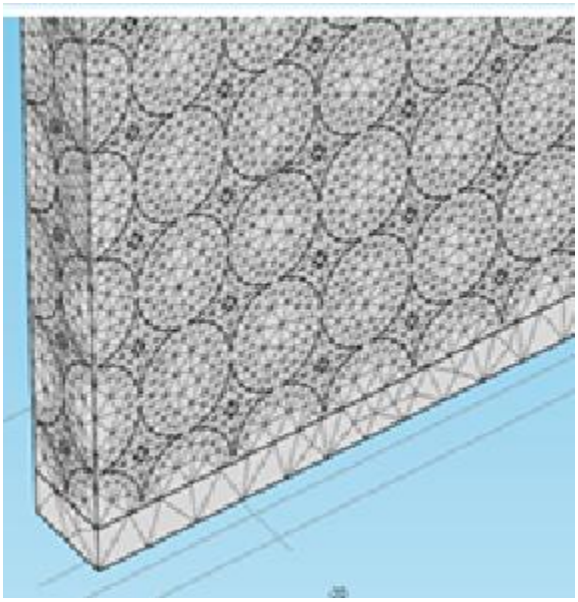

Figure 3: The computational domain with fine mesh

The three different unstructured-tetrahedral-element meshes were generated using the commercial software COMSOL 4.3. The coarse mesh consisted of 1,640,205 elements, the medium mesh consisted of 2,868,913 and the fine mesh consisted of 4,662,368. As such, the coarsest mesh was refined up to 2.8 times to perform the mesh independence study. All meshes were applied to the same computational model in Fig. 3, in which the inlet air velocity was 0.2 m/sec, the porosity was 94% and the heat wall bottom wall flux was 8495 W/ m^2.

Figure 4 shows the fluid temperatures obtained from the simulation results perpendicular to the flow direction (x direction), and for a distance of 6.35 mm from the inlet for the three different meshes. As can be seen in this figure, there is no significant difference among the three meshes: coarse, medium, and fine.

Average differences in fluid velocities, for the same entrance velocity at 6.35 mm from the inlet, were 4.7% between the medium and the coarse meshes, and 0.17% between the medium and the fine meshes. Thus the medium mesh was sufficient for producing results practically independent of the mesh, and was, therefore, chosen to perform all the simulations.

For the simulation, the flow was assumed to be laminar, steady and incompressible. The simulation was run until convergence, which took 1 hour, 24 minutes and 53 seconds. The computer ran on a 64-bit operating system, with a processor operating at 3.40 GHz and 31.7 GB of usable RAM. Other pertinent parameters are listed in Table 1.

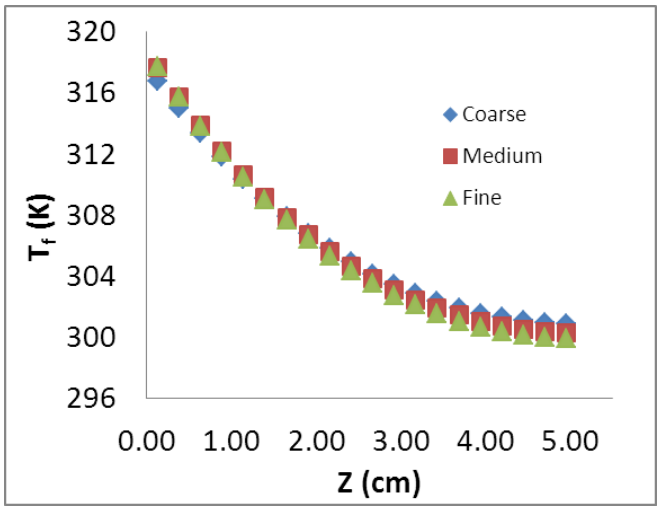

Figure 4: Fluid temperatures at x = 6.35 mm for the three meshes

Table 1: Foam simulation parameters

Domain Size (mm)	50.8X2.54X50.8
Porosity %	94
Heat Flux (W/m^2)	8495
Inlet Velocity (m/s)	0.2
Outlet Pressure (bar)	1.0
Inlet Temperature (K)	295.15

RESULTS

The temperature field is presented in the Fig. 5. The fluid flows along the positive x direction. Figure 5 is color-coded with temperature magnitudes, with red representing high temperatures zones and blue representing low temperature zones. Other colors indicate intermediate temperatures.

The temperature field exhibits very physically realistic trends: the temperature is highest close to the heated wall and decays as the distance from the heated wall increases. Close to the entrance of the porous channel, the temperature of the air is the same as the inlet temperature 295 K. As the distance in the flow direction increases, the air picks up more heat from the heated wall and its temperature increases. Obviously, the heat transfer is not fully developed, and seems to be evolving towards a fully-developed condition. These realistic trends lend some confidence to the simulation approach.

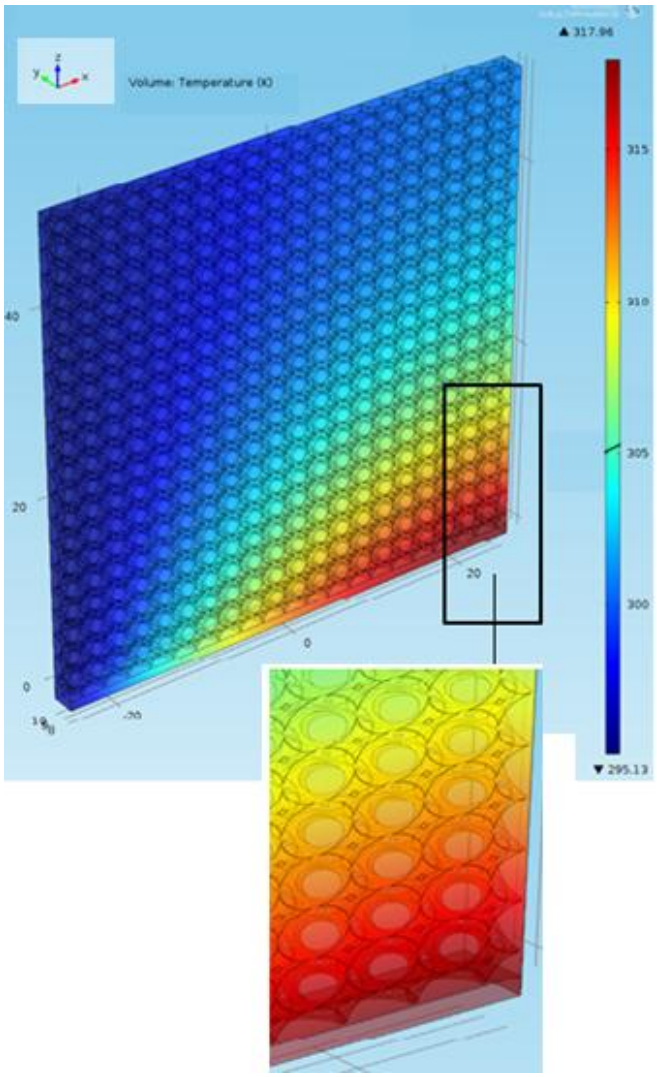

Figure 5: Air temperature field inside the foam

EXPERIMENTAL VALIDATION

For validation of the simulation approach, an experiment was conducted using a metal foam model. The experimental model was an aluminum tube (15.24-cm long and 25.56-cm in diameter) filled and brazed to an aluminum foam core. The commercial foam had a porosity of 91% and 20 ppi (pores per linear inch) [16].

Several thermocouples were used to directly measure the fluid temperatures inside the foam. To insure that these thermocouples measured the fluid temperature only, each thermocouple was inserted and shielded in a specially designed small perforated aluminum tube. The bead of each thermocouple did not extend out of the small tube, and remained shielded since each thermocouple as fixed in place using epoxy, as seen in Fig. 6(a). As such, the bead would not touch the solid ligaments of the foam or the wall of the small tube.

At a distance of 6.35 cm form the foam entrance, set of ten holes were drilled through the wall of the foam reaching 2.03, 3.30, 4.57, 5.84, 7.11, 8.38 9.56, 10.92, 12.19 and 13.46 cm, measured from the outer surface of the tube. The holes were arranged around the cross section with an angle of 36^{o} between each two adjacent holes. The holes were organized as to minimize interference with the air flow through the foam and maintain the internal structure of the foam as much as possible. The small perforated tubes with their thermocouples were inserted in these holes, Fig. 6 (b).

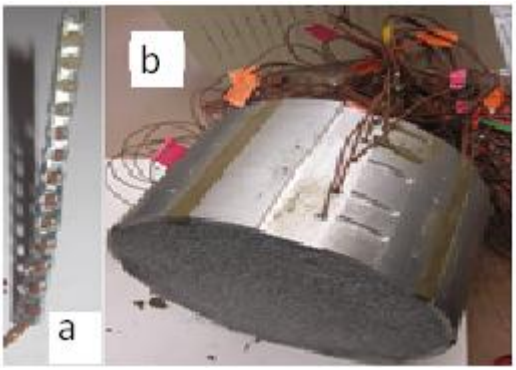

Figure 6: Experimental heat transfer: a) Thermocouple assymbly, b) Themocouple assymblies inserted into holes

Other small holes were drilled in the wall of the cylinder for measuring the wall temperature at various locations in the flow direction. Thermfoil heaters covered the outside surface area of the tube. Each heater could provide up to 645 Watts. The assembly was insulated.

Experiments were performed in an open-loop wind tunnel, Fig. 7. A suction unit which could produce air flow rates up to 17 m^3/min inducted room air through the experimental foam specimen. A gas flow meter that could measure speeds up to 35 m/s was used to measure air flow rate

The flow rate was adjusted, such that the desired speed was realized inside the foam. The surface heaters were powered, and the power input was adjusted using a variac, so that the desired heat flux was achieved. The air temperatures inside the foam, as well as the wall temperature, were monitored until steady-state conditions were achieved, which took about 40 minutes. The steady-state air and wall temperatures were recorded, as was the ambient air temperature, using a data acquisition system. The uncertainty in the non-dimensional temperature of the fluid was determined to be $\pm 14.3\%$ [17].

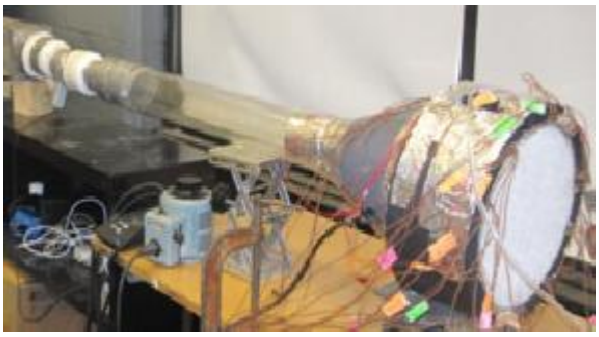

Figure 7: Photograph of the experimental set-up

Figure 8 is a comparison of the numerically-obtained fluid temperature to the measured values. The temperature is presented in its non-dimensional form: $\theta_f = (T_f - T_w)/q''H/k_s$. The effective thermal conductivity of the foam k_s was obtained from [18]. The average error between the experimental and numerical air temperature is about 14 % with a maximum and minimum of 31 % and 5 %. The difference is due to some experimental errors in the measured values, and the idealization of the foam structure in the simulations.

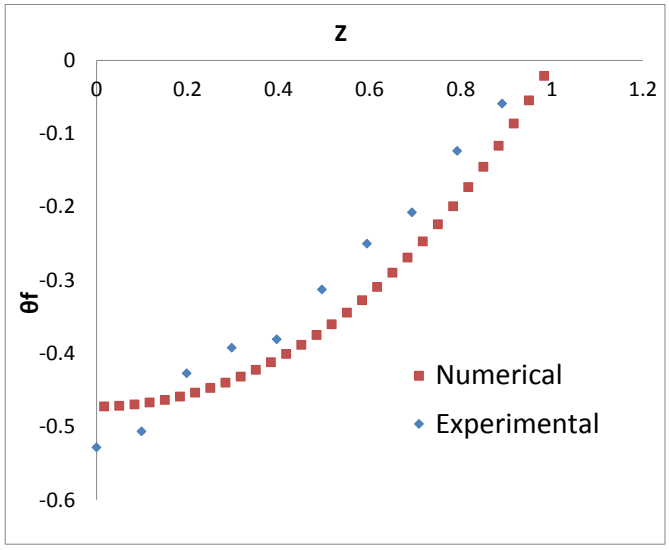

Figure 8: Comparison of the numerical and experimental fluid temperatures

CONCLUSION

Convection heat transfer in open-cell metal foam was successfully simulated using a unit geometrical cell representing the internal structure of 20-pores-per-inch, 94% porous foam. The unit was imported into a finite-element CFD packaged and the flow and convective heat transfer of air was analyzed numerically by solving the pertinent governing equations in the fluid domain. The temperature profiles obtained from the numerical solution were seen to make qualitative physical sense. An experiment was conducted on commercial aluminum foam for validation of the simulation technique. On average, good agreement between the modeling and experimental results is obtained. The results are encouraging and lend confidence to the modeling approach. Applications of the current modeling approach can be used as an estimate for design of air-cooled open-cell metal systems.

REFERENCES

[1] K. Vafai, C. L. Tien, Boundary and inertia effects on convective mass Transfer in porous media, Int. J. Heat Mass Tran., 25 (8) (1982) 1183-1190.

[2] M. L. Hunt, C. L. Tien, Effect of thermal dispersion on forced convection in fibrous media, Int. J. Heat Mass Tran., 31 (2) (1988) 301-309.

[3] V. V. Calmidi, R. L. Mahajan, Forced convection in high porosity metal foams, J. Heat Transfer, 122 (2000) 557-565.

[4] D. Angirasa, Forced convective heat transfer in metallic fibrous materials, J. Heat Transfer, 124 (2002) 739-745.

[5] du Plessis, J.P., Montillet, A., Comiti, J., Legrand, J.: Pressure drop prediction for flow through high porosity metallic foams. Chem. Eng. Sci. 49, 3545-3553 (1994)

[6] Boomsma, K., Poulikakos, D., Ventikos, Y.: Simulation of flow through open cell metal foams using an idealized periodic cell structure. Int. J. Heat Fluid Flow 24, 825-834 (2003)

[7] Krishnan, S., Murthy, J.Y., Garimella, S.V.: Direct simulation of transport in open-cell metal foam, Journal of Heat Transfer. 128(8), 793-799 (2006)

[8] Karimian, S.A.M., Straatman, A.G.: CFD study of the hydraulic and thermal behavior of spherical-void-phase porous materials. Int. J. Heat Fluid Flow 29(1), 292-305 (2008)

[9] Bai, M., Chung, J.N.N.: Analytical and numerical prediction of heat transfer and pressure drop in open-cell metal foams. Int. J. Therm. Sci. 50(6), 869-880 (2011)

[10] Horneber, T., Rauh, C., Delgado, A.: Fluid dynamic characterization of porous solids in catalytic fixed-bed reactors. Microporous Mesoporous Mat. 154, 170–174 (2012).

[11] Kopanidis, A., Theodorakakos, A., Gavaises, E., Bouris, D.: 3D numerical simulation of flow and conjugate heat transfer through a pore scale model of high porosity open cell metal foam. Int. J. Heat Mass Trans. 53(11-12), 2539–2550 (2010)

[12] H. J. Xu, Z. G, Qu, W. Q. Tao, Analytical solution of forced convection heat transfer in tubes partially filled with metallic foam using the two-equation model, Int. J. Heat Mass Tran., 54 (2011), 3846-38.

[13] Z. G. Qu, H. J. Xu, W. Q. Tao, Fully developed forced convective heat transfer in an annulus partially filled with metallic foams: An analytical solution, Int. J. of Heat Mass Tran., 55 (2012), 7508-7519.

[14] H. J. Xu, Z. G. Qu, W. Q. Tao, Thermal transport analysis in parallel-plate channel filled with open-celled metallic foams, Int. J. of Heat Mass Tran., 38 (2011) 868-873.

[15] C. Y. Zhao, Review on thermal transport in high porosity cellular metal foams with open cells, Int. J. Heat Mass Tran., 55 (2012), 3618-3632

[16] ERG Materials and Aerospace: http://www.ergaerospace.com, accessed March, 2013

[17] R. Figliola, D. Beasley, Theory and Design for Mechanical Measurements, John Wiley and Sons, New York, NY, 2000, pp. 149-163.

[18] A. Bhattacharya, V.V. Calmidi, R. L. Mahajan, Thermophysical properties of high porosity metal foams, Int. J. Heat Mass Tran., 45 (2002) 1017-1031