Modelling and Numerical Simulation of Fluid Flow and Heat Transfer through Open Celled Metal Foam

K Gokul Kumar¹ and C D Naiju²
¹,²School of Mechanical Engineering, Vellore Institute of Technology, Vellore, India
Email:kgokulkumar@vit.ac.in
Email:cdnaiju@vit.ac.in

Abstract - The use of open-cell metal foam in numerous technologies is increasing rapidly. Heat transfer measurements inside rectangular blocks of commercially-available aluminium foam subjected to constant heat flux at metal foam surface are presented. Cases for different pore velocities, porosities and pore densities of the foam are given. Each foam block is cooled by a confined stream of ambient air and the inlet-outlet boundaries of the computational domain were assigned. Simulations were carried out using the commercial software Fluent with a specific mass flow rate at the periodic boundaries of the domain and heat flux at the structure of the metal foam thus enabling convection. Different meshing schemes and different grid sizes were used for mesh independent to get the results with different flow velocities. Different approaches for heat transfer analysis of metal foam is considered and the most suitable approach is proposed. The temperature and the pressure drop were determined with different velocities. The heat removal rate for one unit cell was extrapolated to calculate it for the whole bed. For flow and heat transfer analysis, two fluids: air and water are used and the various results obtained are compared. Thus the variation of flow and heat transfer properties was studied with varying flow and foam parameters.

Keywords: Simulation, Open Cell Form, Heat Transfer.

1. Introduction

Only in the past twenty five years, transport phenomena in metal foam have received more attention. They are a relative new class of materials with very promising applications in which low density and other thermal, mechanical, electrical and acoustical properties make this material an excellent means of performance improvement. High-porosity metal foam continues to be applied in many new technologies, which warrants more characterization and assessment of its performance and properties. Different metal foams have been used in aerospace systems, geothermal operations and in petroleum reservoirs. Thermal management applications of foams include compact heat exchangers for airborne equipment, air-cooled condenser towers and compact heat sinks for power electronics.

Open cell metal foams are found useful for the construction of light weight structures, energy absorption devices, currently being used by some vehicle manufacturers, and for various fluid flow and thermal applications which is our interest in this work. The open high porosity (often greater than 0.9), high thermal conductivity of the solid ligaments, the large embedded surface, and the ability to generate turbulence and high-level mixing in the cooling fluid, make metal foam heat exchangers compact, efficient and light-weight. Considerable amount of work has been reported on the numerical simulation of open celled form. An analytical study of the forced convection heat transfer characteristics in high porosity open-cell metal-foam filled pipes was reported [1]. The Brinkman-extended Darcy momentum model and two-equation heat transfer model for porous media were employed. Based on the analytical solutions, the velocity and temperature distributions in metal-foam filled pipes were obtained. The use of open-celled metal foams as compact heat exchangers, exploiting convective cooling was studied and presented [2]. Based on heat transfer data, an analytical model is developed with simple cubic unit cells with heated slender cylinders. Analysis is carried out for a form chilled channel with constant wall temperatures. As a function of foam density, distribution of temperature inside channel is found. Studies of flow through an intricately structured porous medium was modelled using a new approach is reported [3]. A numerical simulation was carried out and results for mesh-independent on a flow through the form were compared with experiments and reported. The increase in pressure drop by the container walls was also compared. Using a periodic unit cell, a direct simulation of the transport in open cell metal foam is carried out assuming the pore to be spherical [4]. From a direct simulation, the effective conductivity, pressure drop, and local heat transfer coefficient is compared with existing experimental values.
Generally metal foams are categorized in two groups based on the liquid fraction and cell geometry. The liquid fraction of what is considered dry foam is relatively low, typically less than 1% and with film type geometry. This dry foam has such a small liquid fraction that the individual cells can be considered single surfaces in which soap bubbles represent its geometry very well. The differences between open cell and closed cell metal foams are mainly how the geometry of the cell is formed. In the open cell group the cells are not closed from each other and the flow of other materials through one cell occurs freely to another adjacent cell. This type of foam is generally created during the foam manufacturing by removing the inter-cellular membranes of closed cell foam. In the closed cell arrangement the surface tension and wetness scales increase and the geometry takes a very different shape compared to the open cell arrangements. In the open cell group the cells are not closed from each other and the flow of other materials through one cell occurs freely to another adjacent cell. This type of foam is generally created during the foam manufacturing by removing the inter-cellular membranes of closed cell foam. In the closed cell arrangement the surface tension and wetness scales increase and the geometry takes a very different shape compared to the open cell arrangements. It is characterized by the continuous cell walls which completely close the cells from one another with the formation of individual cell compartments. The cells nearly take a spherical shape in these types of metal foams. It is designated as wet foam when the liquid fraction exceeds 5%. The variation of flow and heat transfer properties was studied with varying flow and foam parameters. Heat transfer measurements inside rectangular blocks of commercially-available aluminum foam subjected to constant heat flux at metal foam surface are presented. Cases for different pore velocities, porosities and pore densities of the foam are given. The heat removal rate for one unit cell was extrapolated to calculate it for the whole bed. For flow and heat transfer analysis, two fluids: air and water are used and the various results obtained are compared. The proposed structure of the unit cell used in this work is a Tetrakaidecahedron model which is the most efficient packing structure proposed for the unit cells.

2. Modelling Of Cell Form

2.1 Cell Modelling

The first step in understanding the structure of a cell is obtained by considering how cells of equal volume pack themselves under pressure to most efficiently use space. From a popular experiment of peas Lord Kelvin postulated that the ideal packing cell shape was the so-called “tetrakaidecahedron” which is a figure consisting of six planar quadrilateral faces and eight non planar hexagons of zero net curvature. This shape also satisfies Plateau’s conditions for a network of foam films. Surfaces which bound the cells meet at 1200, and the lines which are formed by their intersections meet at the tetrahedral angle cos-1 (-1/3). The structure of the metal foam used in this work is Tetrakaidecahedron which is modeled using commercial software GABIT 2.3.16. This geometric shape has been shown experimentally to be less representative than a multi cell unit cell in polymer foam; however, it is the most representative single-celled, space-filling, unit cell of the foam. This is similar to the structure proposed by Boomsma and Poulikakos [5] except with spherical nodes, which was deemed more physically accurate than cubes. The parameters used to fully describe the Tetra K unit cell are the relative density, specific surface area, average ligament diameter, ligament length, and a node size parameter. The following relationships are used to calculate modelling parameters of the model.

An elongated tetrakaidecahedron also packs to fill the space. It contains eight hexagonal faces, two horizontal square faces and four vertical diamond faces. The horizontal square faces have sides of length b and the diamond faces have sides of length L. The hexagonal faces have four sides with length L and two sides with length b. The inclination angle defines the orientation of the hexagonal faces with respect to the rise direction as well as the obtuse angle of the vertical diamond faces. When using a tetrak structure with all its sides equal then b=L=1; we obtain ≈45. Thus each hexagonal plane is inclined at an angle of 450 with horizontal thus forming a closed geometry.

Figure 1 shows the structure modeled in Gambit 2.3.16 as per the dimensions of ligament and the spherical nodes calculated. The TetraK model ligament cross section is circular and the diameter is constant over its length. The geometry of the structure was determined by solving the above equations using the values. The structure is similar to the structure proposed by Boomsma and Poulikakos except with spherical nodes, which was deemed more physically accurate than cubes.
2.2 Parameters of Metal Foam Model

Diameter of ligament, \( d_f = 0.20 \) units.
Diameter of spherical nodes = 0.45 units.
Average Pore Diameter = 1.52 units.
Side of the unit cell: \( 4.242 \times 10^{-3} \) m
Surface area of the Aluminum ligament: \( 4.550367 \times 10^{-5} \) m\(^2\)
Volume of Al ligament: \( 2.678266 \times 10^{-9} \) m\(^3\)
Volume of fluid: \( 7.363946 \times 10^{-8} \) m\(^3\)
Total volume of unit cell: \( 7.631772 \times 10^{-8} \) m\(^3\)
Porosity of the unit cell: 96.49 %

2.3 Computational Domain

A computational domain is to be defined for applying the boundary conditions and for limiting the volume for simulation. A box is made as a computational domain enclosing the unit cell such that it is symmetric at all surfaces and is cut from the foam bed. Figure 2 illustrates the bounding box enclosing tetrakaidecahedron as the unit cell. The bounding box, the Representative Elementary Volume (REV) which defines the symmetry is constructed and the foam is trimmed at the intersection with the bounding box so that the cell ligaments are fused with the bounding box. The dimensions of the box taken in the present work are \( 4.242 \times 4.242 \times 4.242 \) units. The unit cell is a squared Tetrakaidecahedron structure such that, Height = Width = 2.82 Units.

2.4 Meshing the Structure

The tetrak structure should be meshed as 2D triangular mesh on the surface and 3D tetrahedral mesh in the volume. The basic periodic cellular unit of the foam structure can be used as a building block to represent a much larger foam network. This is accomplished by meshing the visible void region of the structure as a fluid volume with a periodic grid at the boundaries. While meshing the geometry a bounding box or a computational domain is created and the volume enclosed by it is meshed using tetrahedral elements. The mesh generated on the surface of the structure is made to connect to the mesh generated in the void volume. Connectivity of the
mesh is necessary for the flow simulation to be continuous both in the void and over the surface of the structure. The mesh generated is of size such that it contains 214194 tetrahedral elements. Figure 3 (a) depicts the meshing of the solid structure of cell and the Figure 3 (b) shows the meshing of the fluid region and solid region. With the faces of the whole structure being connected, the mesh of the fluid is generated taking the mesh on solid region as source mesh. This ensures connectivity between the nodes of fluid and solid region. A mesh check was performed throughout the volume and it was found that a minimum percentage of nodes i.e. 0.25% were found to have a skewness of 0.7 to 0.8, and none of the elements had any higher skewness. Thus the mesh generated is of good quality.

![Figure. 3 (a) Solid Volume, (b) Fluid Volume](image)

3. Fluid Flow Simulations Through Metal Foam

The porous structures have long been thought to behave as packed granular or spherical beds. But because of the structural difference between open-cell metal foams and such granular beds, it is important to characterize the particular fluid behaviour of metal foams with a different approach. This approach is different in the sense that metal foam is a rather periodic structure where the geometry and the flow pattern repeat itself. In the flow characterization of these type of material the Reynolds number is a usual non-dimensional parameter used to indicate the flow regime used in the experimentation. Although various definitions of the Reynolds number have been proposed in the past literature, the most accepted definition used at the macroscopic level is the permeability based Reynolds number, another non-dimensional term to describe the ratio of macroscopic form to viscous forces on porous media.

3.1 Boundary Conditions

Boundary conditions are given at the structure of the metal foam as a heat flux value of 1000 W/m² on the walls of the metal foam, thus ensuring the convection of heat taking place. It is assumed that no conduction takes place through the solid wall as there is no temperature gradient. The boundary Conditions are given as follows: Inlet Face: Periodic, Outlet Face: Periodic, Other Faces: Symmetry, Structure of the Cell: Wall /Solid, Void Volume: Fluid.

As Periodic boundary conditions are assigned at the inlet and outlet faces of the computational domain to model the large foam network in the stream wise direction, hence it is obligatory to link these two faces in Gambit, this requires to give each loop on the face and a vertex to link. The faces are linked in the reverse orientation so that node to node correspondence takes place. Linking the faces ensures accurate and exact mapping of mesh nodes of one face on to another. Periodic boundary conditions are translational in nature i.e. the flow repeats itself translationally in one direction i.e. stream wise direction. A volumetric mass flow rate is also specified at the periodic boundaries as the input thus different values of velocities are used hence experimenting with different values of fluid. Symmetry boundary conditions are used when the physical geometry of interest, and the expected pattern of the flow/thermal solution, has mirror symmetry. They can also be used to model zero-shear slip walls in viscous flows.

Symmetry boundary conditions are given to define planes about which the cell can be duplicated. Symmetry boundary conditions take zero shear stress values and zero surface fluxes as q=0. As stated above, these conditions determine a zero flux across the symmetry plane, which is required by the definition of symmetry. Since the shear stress is zero at a symmetry boundary, it can also be interpreted as a "slip" wall when used in viscous flow calculations. There is no diffusion flux across a symmetry plane: the normal gradients of all flow variables are thus zero at the symmetry plane. The symmetry boundary condition can therefore be summarized as follows: Zero normal velocity at a symmetry plane zero normal gradients of all variables at a symmetry plane Structure of the metal foam is given as solid wall and the void volume as the fluid. By giving a constant heat flux value of 1000 W/m² it is assumed that there is no heat conduction in the walls of the metal foam. Only the fluid is taking heat from it by forced convection.

4. Simulations Of Metal Forms
While solving for the flow, turbulence and energy equations algorithm is used and the discretization technique used is second order upwind for better accuracy. Convergence can be controlled by using aggressive Algebraic multi grid solver and by using the under relaxation factors of 0.7 for momentum, 1 for energy. For iteration initial values of velocities, energy turbulence kinetic energy and dissipation rate are specified. Heat transfer, convergence and pressure gradient are monitored through the iteration. Following are the approaches and parametric variations carried out for simulations: Simulations are done for different flow velocities varying linearly from 1 to 5 m/s for a constant heat flux. Pressure gradient, heat transfer coefficient and temperature of fluid and solid, Nusselt’s Number and Reynolds’s Number are determined for every velocity and being plotted to show the graphical variation. Mesh independent results are shown using a coarse mesh of size 0.1 at the solid and 0.15 at the fluid. Heat removal rate for a sample foam bed is determined from one unit cell. Curve for the static pressure is drawn which is opposite in nature to the velocity curve thus a higher velocity results in lower pressure. Under relaxation factors are given. These factors are responsible in determining the convergence of the solution.

Algebraic Multi Grid (AMG) solver is used in this work which uses flexible cycle for all equations except energy for which it uses F cycle, the under relaxation factors can be reduced for an F cycle to 1 for convergence. Second order upwind provides a better accuracy although it takes more time. When the flow is aligned with the grid (e.g., laminar flow in a rectangular duct modeled with a quadrilateral or hexahedral grid) the first-order upwind discretization may be acceptable. When the flow is not aligned with the grid (i.e., when it crosses the grid lines obliquely), however, first-order convective discretization increases the numerical discretization error (numerical diffusion). While the first-order discretization generally yields better convergence than the second-order scheme, it generally will yield less accurate results, especially on tri/tet grids. Let the fluid's approach velocity be U. The pore velocity, u, in the pores of open-cell metal foam is also defined. This formulation has been used using fluent to calculate the net area averaged velocity at the inlet-outlet pair for a flow velocity of 1 to 5 m/s.

5. Results And Discussions

The variation of velocity across the unit cell is shown in figure 4 (a) and (b) gives the variation of static pressure across the velocity. By the nature of the two graphs we infer that the nature of pressure variation is opposite to that of velocity variation. A sudden dip in the value of the velocity is because of the momentarily rest position of the fluid particle in the when it collides with the wall surface. This results into an instant accumulation of particles there and hence increasing the pressure.

![Figure 4](image_url)

Figure. 4 (a) Velocity Magnitude Variation, (b) Pressure Static Variation

A sudden decrease in velocity around the mid plane is the result of more intricate geometry that reduces the pressure around this area. The Velocity contour in figure 5 (a) depicts the periodic boundary condition assigned at the inlet-outlet pair. The contour above suggest the exact mapping of velocity values on two periodic modules for modeling the large foam network in the stream wise direction. The Zero velocity of the fluid at the walls or the surface of the metal foam suggests “No slip Condition” at the surface hence due to wall shear stress a boundary layer is formed from pressure contour in figure 5 (b), we obtain that pressure is very low or negative at the face of the ligaments not facing the flow; this is due to the wake formation as the flow moves around the surface of structure.
Figure 5 (a) Velocity Contour, (b) Pressure Contour [Over inlet-outlet pair and Wall]

As the flow velocity is varied from 1 to 5 m/s, variations were observed in pressure variation profiles across the unit cell as plotted in figure 6 (a). With an increase in velocity of the flow, pressure drop across the cell increase. The porosity of the metal foam modeled is higher, hence lesser number of peaks is observed in the pressure profiles across the cell. The variation of negative pressure gradient and the graph is plotted in figure 6 (b). The graph shows a decrease in pressure gradient with the velocity. This shows the pore diameter, and not the porosity, increases the specific surface area \( (Ao) \) through the geometrical relationship involved in the scaling of the structure. This increase in the specific surface area has a larger effect on the permeability and overall flow resistance of foam than an increase in the solid fraction of the foam, since viscous drag is the dominating factor of pressure drop through a porous medium which is governed by the solid-fluid interfacial area.

Figure 6 (a) Variation of Pressure, (b) Variation of Inlet Velocity

A unit cell of pore diameter of 2.3 mm has a pore density of 40 ppi and the unit cell in the work is of 1.52 mm, hence we conclude its pore density to be 20 ppi with a porosity of 96%. Since viscous drag is the dominating factor of pressure drop through a porous medium, which is governed by the solid-fluid interfacial area we conclude that since the tetrakaidecahedron geometry provides a higher specific surface area thus producing more viscous drag hence giving larger pressure gradient values. From Pressure contour, we obtain that pressure is very low or negative at the face of the ligaments not facing the flow; this is due to the wake formation as the flow moves around the surface of structure.

A rise in pressure is attributed to the momentarily rest position of the fluid particle at the surface of wall, leading to increase in stagnation pressure. The pressure gradient is composed of two parts (i) The gradient of the periodic component and (ii) The gradient of linearly varying component, this linearly varying component is associated with the viscous drag. With an increase in velocity the shear stress due to viscous effects increase by Newton’s Law of viscosity resulting in increasing pressure gradient. Because the value of pressure gradient is not known a priori, it must be iterated on until the mass flow rate that have been defined is achieved in the computational model. This correction of gradient term occurs in the pressure correction step of the SIMPLE, SIMPLEC, or PISO algorithm where its value is updated based on the difference between the desired mass flow rate and the actual one. In the present work, these iterations have been restricted to 2 and is monitored by providing an under relaxation factor. Area weighted average of pressure at the periodic boundaries gives periodic pressure.

6. Conclusion

In the present work the simulations are done for open celled metal foam with periodic boundary conditions on the inlet-outlet pair. Applying periodic boundary conditions helps us in simulating a unit cell and extrapolating the results for the whole of the bed, thus the periodic conditions in this work helped us to model the large foam network in the stream wise direction. The flow simulations obtained showed the occurrence of flow reversal formation of wake regions around the cell ligament thus affecting the linearly varying component of the periodic pressure, by viscous
drag. Hence it is concluded about the pressure drop that the intricate geometry of Tetrakaidcahedron helped in increasing the net viscous drag thus varying the pressure and the velocity across the cell. This variation creates the necessary turbulence in the cell thus producing fluctuating velocities. Flow simulations led us to conclude that with an increase in velocity the pressure gradient is decreased as given by graph. Forced convection through metallic foams has been proved to substantially enhance heat transfer rates. Given this characteristic of the material, metal foams are well suited for the use in high performance compact heat exchangers and heat sinks for the use in electronic equipment and other industrial applications. Convective heat transfer in a fibrous medium involves the formation of complex flow and temperature fields around individual fibers. Flow separation may occur around the fibers at higher velocities than that of creep flow. Even though this leads to higher pressure drop values, the associated mixing substantially enhances the heat transfer rate. In addition when the fluid saturated medium has a larger effective thermal conductivity than the fluid alone, the heat dissipation is enhanced this in turn helps to keep the surface temperature within acceptable limits.

Acknowledgments

The authors appreciate the support of VIT management. The authors would also like to express their sincere gratitude to all the faculties and lab technicians involved in this research work for rendering their valuable guidance and help.

References:

[1] Lu, W., Zhao, C. Y., Tassou, S. A., “Thermal analysis on metal-foam filled heat exchangers Part I: Metal-foam filled pipes” International Journal of Heat and Mass Transfer, 2005, 49, p.2751-2761.

[2] Lu, T. J., Stone, H. A., Ashby, M. F., “Heat transfer in open-cell metal foams” Acta Materialia, 1998, 46, p. 3619-3635.

[3] Boomsma, K., Poulikakos, D., Ventikos, Y., “Simulations of flow through open cell metal foams using an idealized periodic cell structure” International Journal of Heat and Fluid Flow, 2003, 24, p. 825–834.

[4] Krishnan, S., Jayathi, Y. M., Suresh, V. G., “Direct simulation of transport in open-cell metal foam” Journal of Heat Transfer, 2006, 128, p.793-799.

[5] Boomsma, K., Poulikakos, D., “On the effective thermal conductivity of a three dimensionally structured fluid-saturated metal foam”, International Journal of Heat and Mass Transfer, 2001, 44, p.827-836.