

GTINDIA2013-3605

CFD SIMULATION OF OPEN-CELL ALUMINUM METAL FOAMS FOR PRESSURE DROP CHARACTERIZATION

Abhay G K

Amrita School of Engineering
Amritapuri, Kollam, Kerala, India
abhay.g.k@gmail.com

Jayakumar J S

Professor, Amrita School of Engineering
Amritapuri, Kollam, Kerala, India
jsjayan@am.amrita.edu

Giridhar Babu Y

Sr. Scientist, Propulsion Division
CSIR-National Aerospace Laboratories
Bangalore, Karnataka, India
giris@nal.res.in

Felix J

Scientist, Propulsion Division
CSIR-National Aerospace Laboratories
Bangalore, Karnataka, India
felix@nal.res.in

ABSTRACT

Many researchers have done the CFD simulation of open-cell aluminum metal foams with a unit cell with periodic boundary conditions. However, this does not represent a real life situation, as the foam-fluid interactions cannot be properly modeled. In the present study the simulation is done for metal foam with the more number of foam cells to approximate the conditions close to the actual situations. The CFD simulation of open-cell aluminum metal foams was done using ANSYS FLUENT. The results are obtained by solving the Continuity, Momentum and Energy equations and standard $k-\epsilon$ turbulence model is used for simulation. The boundary conditions applied are same as those applied during the experiments conducted at Heat Transfer Lab, National Aerospace Laboratories, Bangalore. In this study the Aluminum Alloy (Al 6101-T6) metal foam of pore density 10 ppi is used for CFD analysis.

INTRODUCTION

In recent years, researchers focus has changed to novel materials for the improved efficiency and better performance in the energy transfer systems. Open-cell metal foams are such kinds of materials which can be considered as a replacement of conventional fins of compact heat exchangers used in aerospace applications as they provide the higher surface area to volume ratio in the order of $10,000 \text{ m}^2/\text{m}^3$ and better heat transfer values in comparison to the conventional fins with a marginally increased pressure drop values. Hence the optimization of these

foams with suitable porosity can be considered for the heat exchanger applications. Also the light weight of these metal foams is an added advantage.

Commercially the metal foams can be prepared from a small subset of metals by casting in different ways whose parent metals are usually Al, Ti, Ni, Mg etc. As a result of the production processes a sponge type metal is formed with polyhedral cells repeated periodically. So the properties of the base metal determine the physical properties of metal foam. Fig.1 shows 10 ppi aluminum metal foam which is used for experiment.

Different models have been developed in past in order to characterize the flow of fluids through the porous medium. In 1856, Darcy proposed Darcy's law [1] [2], which states the pressure drop per unit length for a flow through porous medium is proportional to the product of the fluid velocity and the dynamic viscosity. Later Kruger [3] added the inverse proportionality constant to the Darcy law known as the fluid permeability, which is the measure of the resistance, the fluid undergoes when passing through a porous medium.

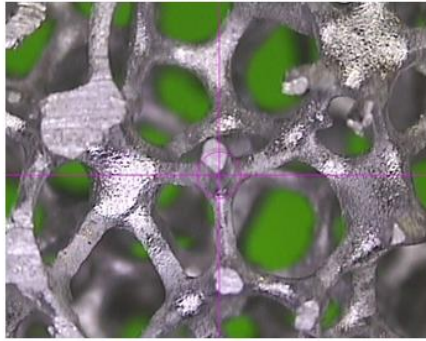
$$-\frac{dp}{dx} = \frac{\mu}{K} v \quad (1)$$

The Darcy's law is applicable for slow moving fluids only and experiments show that as the velocity increases a form drag becomes more established and should be considered for an

appropriate calculation of the permeability describing the porous medium.



(a)



(b)

FIGURE 1. (a) 10 PPI ALUMINUM METAL FOAM (b) DETAILED VIEW

Hence a modification of the Hazen-Dupuit- Darcy equation [3] is done by including a second term to Eq. (1) and modified this to a quadratic equation for the pressure drop relation.

$$-\frac{dp}{dx} = \frac{\mu}{k} v + \frac{f\rho}{\sqrt{K}} v^2 \quad (2)$$

This is the most widely accepted form equation for steady state pressure drop in a homogenous, uniform, isotropic porous medium, fully saturated with a Newtonian incompressible fluid.

Bhattacharya et al. [4] investigated high porosity metal foams both analytically and experimentally for the determination of effective thermal conductivity, permeability and inertial coefficient with porosity between 90-98% with 5, 10, 20 and 40 ppi. Boomsma et al. [5] performed experiment to measure the hydraulic and thermal performance of the open cell aluminum foams when they are used as heat exchangers in a forced convection flow arrangement. Detailed studies of forced convection in high porosity (89 to 97%) aluminum metal foams are done by Calmidi et al. [6]. Fuller et al. [7] studied heat transfer characteristics of an iron based alloy (FeCrAlY) foams and was compared with copper foams. Liu et al. [8] proposed a correlation of friction characteristic based on the measured

pressure drop of air through foam matrixes. They developed an empirical equation to correlate the dimensionless pressure drop with the dimensionless flow velocity for seven types of aluminum foams. During this study, series of experimental tests were conducted to determine the flow friction characteristics of metal foam. An experimental investigation to determine the feasibility of using a high conductivity porous channel as a heat sink for high performance forced cooling in heat exchangers was done by Noh et al. [9]. Mancin et al. [10] studied pressure drop across aluminum metal foam during air flow. They investigated the hydraulic characteristics of air flowing through different rigid open cell aluminum metal foams. They found that pressure gradient increases as the relative density decreases, and the inertia coefficient was found to increase as the porosity increases. They also found that the permeability increases with mean pore diameter.

Many researchers had analyzed the metal foams computationally to determine the characteristics. Through experimentation the unit cell of actual metal foam is found to be similar to a sphere-centered tetrakaidecahedron [11], which is a polyhedron consisting of six quadrilaterals and eight hexagonal faces which is repeated periodically, but it is not mathematically proven to be the optimal packing cell. Weaire and Phelan [11] developed a periodic unit of cells which reduced the surface energy of the packing cells by 0.3%. This is a unit of eight equal volume cells consisting of six 14-sided polyhedron having 12 pentagonal and two hexagonal faces, and two pentagonal Dodecahedra.

Boomsma et al. [12] did the modelling of the flow through metal foam. The periodic boundary condition is applied on the walls parallel to flow direction, in which the flow is driven by specifying a particular volumetric flow rate and the velocity distribution at the inlet is adjusted according to the flow velocity at the outlet. The pressure drop values predicted by their numerical model are 25% less than that of the experimental value. Shankar et al. [13] did direct simulation of transport in open-cell metal foams. In this study they assumed the foam geometry to be spherical. They calculated effective thermal conductivity, friction factor and Nusselt number from the simulation of the periodic unit cell. Also for metal foams with porosities greater than 94% their numerical results are well compared with the experimental measurements. Kopanidis et al. [14] did 3D numerical simulation of flow and conjugate heat transfer at a pore scale model of high porosity open cell metal foam. In this study the calculations are done for different combinations i.e. (a) with and without neighbouring solid wall (b) a constant velocity vs an imposed pressure drop in the flow direction (c) a specified constant temperature at the fluid-solid interface vs a conjugate temperature solution both the fluid and the solid and (d) a pore density of 40 ppi vs 10 ppi.

Mo Bai and Chung [15] did analytical and numerical prediction of heat transfer and pressure drop in open-cell metal foams. To predict the heat transfer capacity of channel filled with foam analytically a diamond shaped unit cell is developed. The CFD simulation of unit cell (tetrakaidekahedron) is done to find out the pressure drop across the metal foam. Due to

repeated cell structure of metal foams periodic boundary conditions are applied across unit cell. Simulation is done with both 2-D and 1-D periodicity. The cell and channel models are created and meshed by GAMBIT, the pre-processing mesh generation software of Fluent. The pressure drops are simulated for the coarse, medium, and fine grid models. Both the heat transfer and pressure drop predictions for the foam filled channel were compared with available experimental data and a good agreement is found out.

MODELLING AND SIMULATION

For proper modeling of fluid flow in open-cell metal foams, the geometry should be realistic. In the present study, a number of unit cells are arranged in a systematic fashion. The foam cells are arranged such that each cell has thermal contact with each other, as the heat has to be conducted along the metal foam. The metal foam of size 9 mm x 10 mm x 10 mm modeled for simulation and is shown in fig 2.

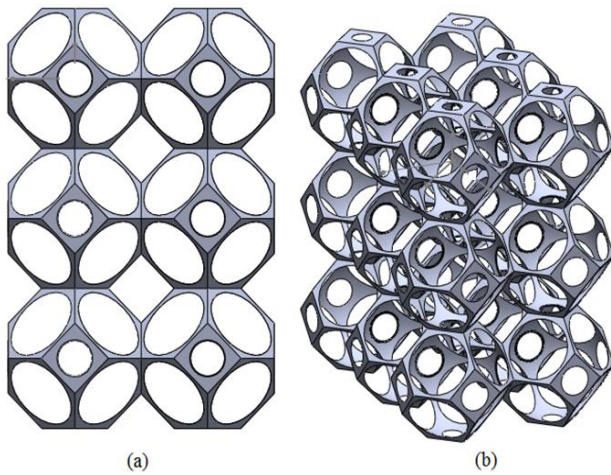


FIGURE 2. (a) FRONT VIEW (b) ISOMETRIC VIEW OF METAL FOAM

Simulation Methodology

In this study the basic methodology of simulation of open-cell metal foam is similar to that used by Mo Bai and Chung [15] in a different loom. Here the simulation is done for metal foam with a number of repeated unit cells unlike that of simulation of unit cell done by Mo Bai and Chung [15].

Geometry Modelling

The production process of metal foam causes the metal foam geometry a repeated cell structure. The cell structure of metal foam is analogous to a polyhedron consisting of six quadrilateral and eight hexagonal faces named tetrakaidekahedron or Kelvin cell as it is first introduced by Lord Kelvin. This structure has very minimum surface area to volume ratio so it can be the best representation of unit cell. The tetrakaidekahedron is modelled in Solidworks. First a truncated octahedron is modelled and a sphere is subtracted

from the truncated octahedron to obtain a sphere-centred tetrakaidekahedron as shown in fig.3.

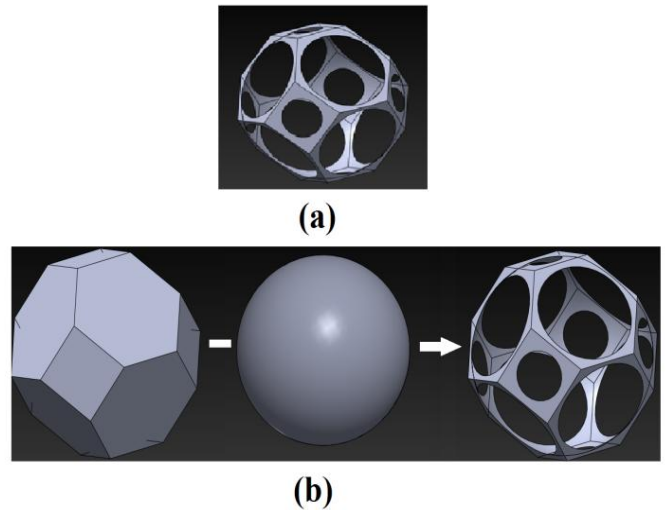


FIGURE 3. (a) TETRAKAIDEKAHEDRON (b) GEOMETRY CREATION OF TETRAKAIDEKAHEDRON

Validation

The validation is done with unit cell model as it is simulated with the same boundary conditions as applied in Mo Bai and Chung [15]. In their work periodic boundary conditions are applied in 2 directions except in the flow direction. The computational domain is reproduced and is shown in fig.4. The numerical simulation is done with air as working fluid and the inlet is set as velocity inlet and the outlet is set as pressure outlet.

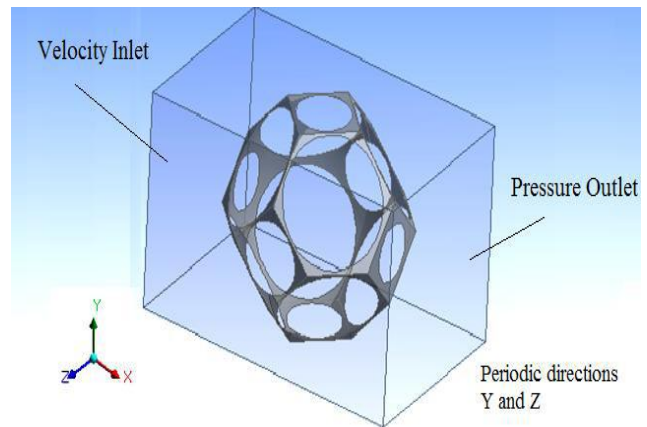


FIGURE 4. COMPUTATIONAL DOMAIN

They did the study for the metal foam of pore density 10ppi with 97.4% porosity. The model is meshed to 1,187,729 cells.

This methodology is validated by keeping the geometry and mesh density same as that used by Mo Bai and Chung [15]. The velocity and pressure contours at various locations when inlet velocity is 4 m/s is shown in fig.5 and fig.6. These contours can be compared with fig.12 and fig.13 of Mo Bai and Chung [15]. The plots shows excellent match with the plots from Mo Bai and Chung [15]. Hence the methodology is validated.

Proposed Model and Grid Generation

In this study the simulation is done for metal foam, i.e. clusters of unit cell are arranged in a systematic fashion such that the boundary conditions resemble the actual situations. The metal foam modeled is about 9 mm x 10 mm x 10 mm. The isometric and front view of the metal foam is shown in fig.2. The foam structure along with the flow area is discretized using a tetrahedral volume mesh.

Also the grid independence of solutions was generated and the calculations are performed on a delicate model with 3,057,286 tetrahedral cells. The meshed foam structure is shown in fig.7. The simulation of the created mesh is done in FLUENT with SIMPLE algorithm for obtaining the velocity fields. A second-order upwind scheme was used for the calculations and it got terminated when the scaled residuals had dropped below 10^{-5} .

Boundary Conditions

Boundary conditions must be applied in such a way that it best resembles the actual scenario. Boundary condition options will vary depending on which the boundary has been selected. The boundary conditions applied are same as that applied for the experiment conducted at National Aerospace Laboratories, Bangalore. The boundary conditions applied during the simulation is listed in Table 1.

TABLE 1. BOUNDARY CONDITION TYPES

Boundary	Boundary condition Type
Inlet	Velocity-inlet
Outlet	Pressure-outlet
Metal foam	Wall
Flow area	Symmetry

For experiment aluminum alloy (Al 6101-T6) metal foams of pore density 5, 10, 20 and 40 ppi with almost constant porosity of about 90% - 92% is used. The aluminum metal foams are investigated experimentally with an air mass flow rate ranging from 0.002 to 0.011 kg/s. The CFD analysis was done for metal foam with 10 ppi as it shows the higher performance in terms of heat transfer with lower pressure drop in comparison to the other considered foams during experiments.

During simulation in FLUENT the velocity inlet boundary condition is set at the inlet and pressure outlet is set at the outlet. The rest of flow area is set as symmetry. The flow direction is always in x-direction as shown in fig.8. The flow is

turbulent inside the metal foam as it is porous, so standard k- ϵ model is used. 5% of turbulent intensity and 6.15 mm hydraulic diameter is given for both the inlet and outlet section.

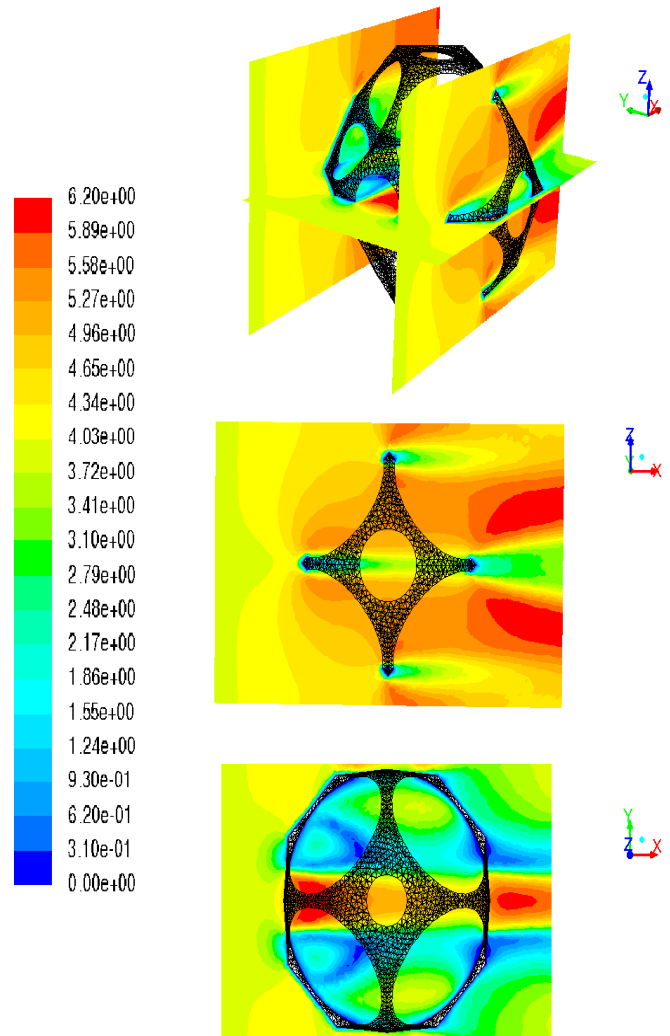


FIGURE 5. VELOCITY CONTOURS

During simulation the material properties are taken at the average temperature and pressure. The simulation is done with COUPLED algorithm, which uses pressure-based coupled solver, as it obtains a robust and efficient single phase implementation in single phase flows. The coupled algorithm solves the momentum and pressure-based continuity equations together. A second-order upwind scheme was used for the calculations. The convergence criteria for dependent variables like continuity, momentum, energy etc are specified as 10^{-7} . The simulation is done in parallel mode of a system having quad core Intel® Core™ i5 processors. The calculations are done over 10000 iterations to get the converged results in a 16GB RAM system for 8 hours.

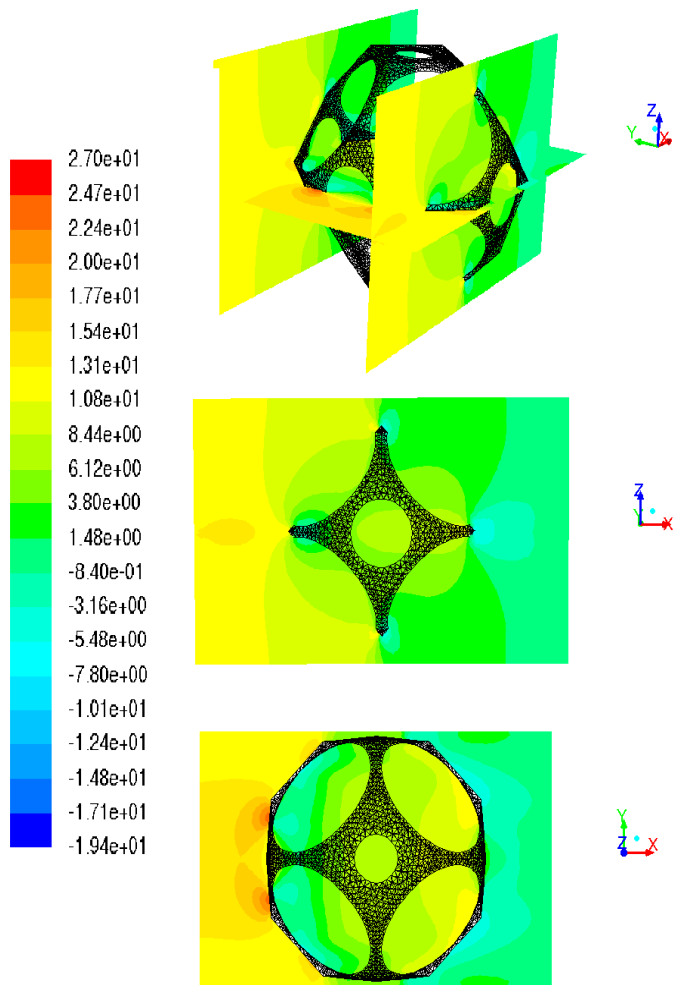


FIGURE 6. PRESSURE CONTOURS

RESULTS AND DISCUSSIONS

The simulations are done for velocities ranging from 3.125 m/s to 8.650 m/s and the pressure drop is found out. The velocity and pressure contours are plotted across the metal foam when the inlet velocity is 4.46 m/s and is shown in fig.9 and fig.10. Fig.9 shows the velocity magnitude contours obtained of several chosen planes for the case with an inlet velocity of 6.703 m/s. There are three planes selected for plotting, fig.9 (a) shows the velocity contours just before the inlet to foam and soon after from the exit. Fig.9 (b) is the horizontal view, showing the contour at the centre plane and Fig.9 (c) shows the contour at centre plane vertically. From the velocity contours it is clear that the velocities between ligaments are relatively high and wakes can be found near ligaments. Fig.10 shows the static pressure contours of the same three planes for an inlet velocity of 6.703 m/s. From the pressure contours it is evident that when the flow bypasses the

ligaments, high pressure is created. Also it is clear that pressure drop happens when flow occurs across metal foams.

During experiment the pressure drop across the foam duct is calculated by taking the difference in average static pressure across the foam duct. The same procedure is repeated during the simulation of metal foam with the same inlet velocity. The difference between the area weighted average static pressure between the inlet and outlet is taken for calculating the pressure drop. The results are then compared with the results obtained from the experiment. The comparison is tabulated in Table 2.

TABLE 2. COMPARISON WITH EXPERIMENTAL DATA

Inlet velocity (m/s)	Pressure Drop (PSI)		Percentage Difference
	Experimental	Numerical	
3.135	0.01	0.008798	12.01
4.734	0.0222	0.019499	12.16
6.703	0.0415	0.03611	12.98
7.451	0.0523	0.044122	15.63
8.650	0.0632	0.055012	12.95

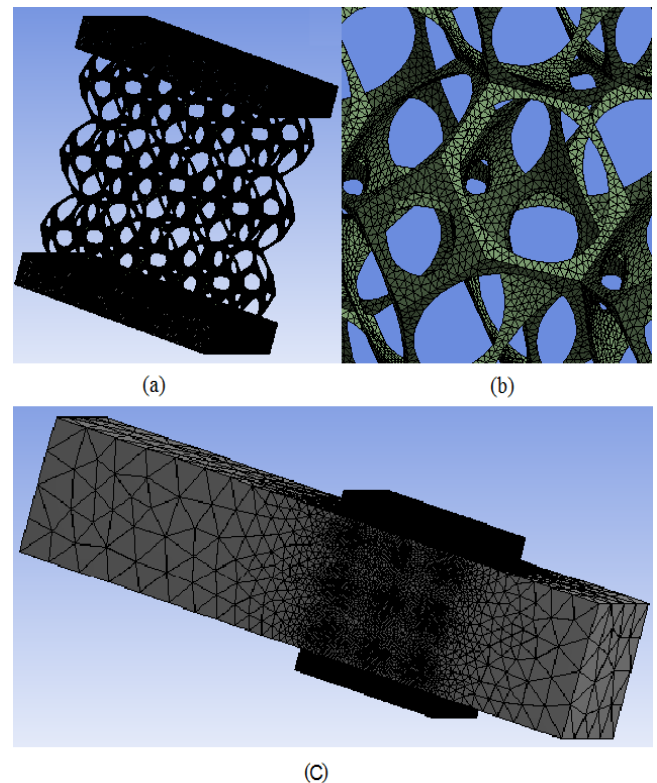


FIGURE 7. MESHED METAL FOAM

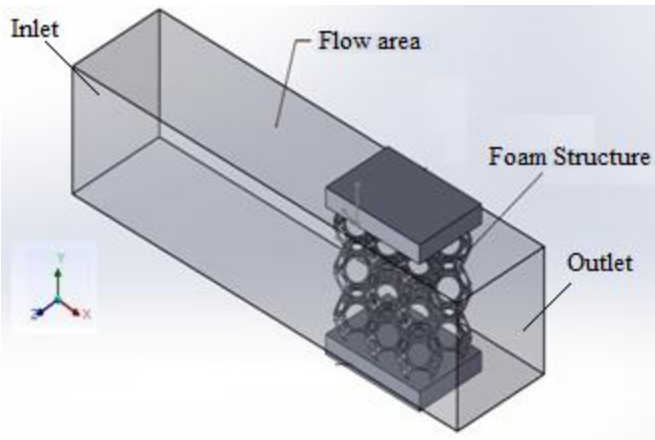


FIGURE 8. COMPUTATIONAL DOMAIN

From table it is clear that the pressure drop increases as the velocity increases. This is due to the increase in the shear stress which occurs due to the increase in the velocity.

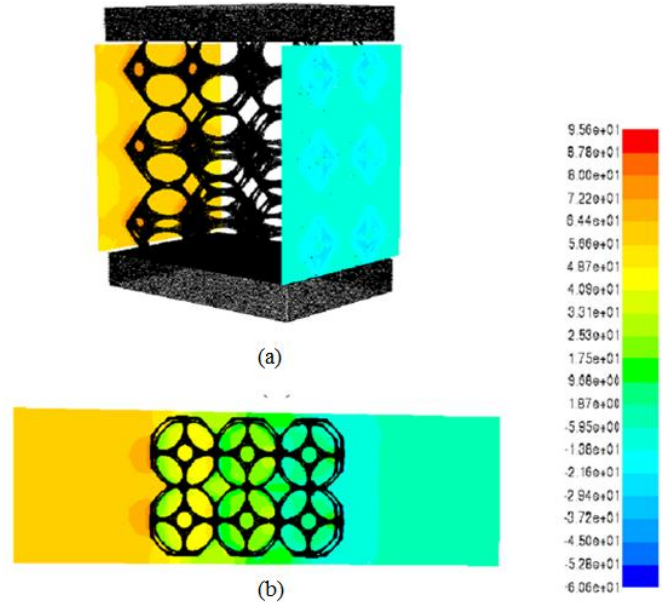


FIGURE 10. PRESSURE CONTOURS

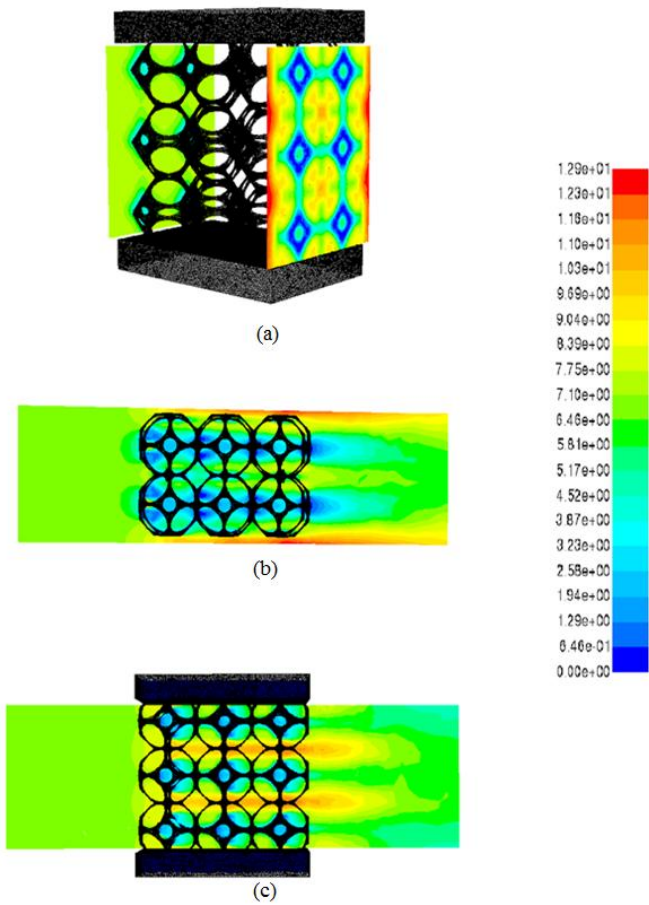


FIGURE 9. VELOCITY CONTOURS AT DIFFERENT LOCATIONS

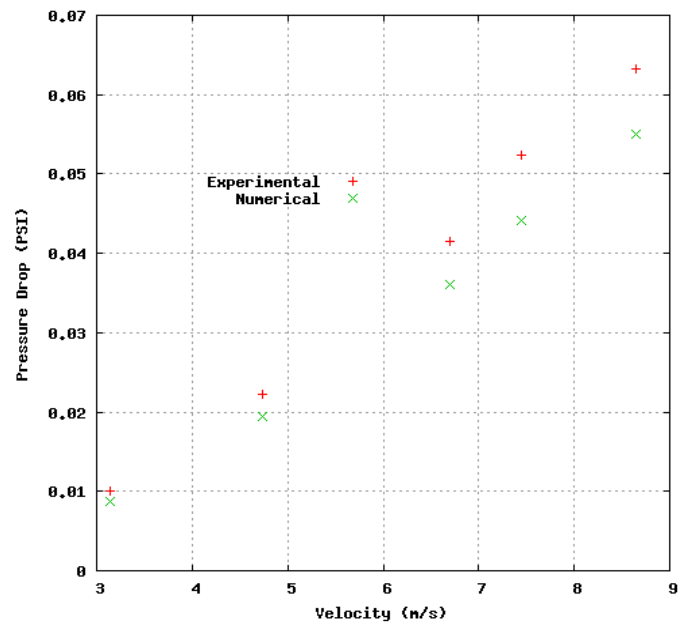


FIGURE 11. PRESSURE DROP VS VELOCITY

Fig. 11 shows the graph between pressure drop and velocity. In this graph pressure drop is plotted against different velocities and it is then compared with the experimental values. There is a difference in the pressure drop values predicted by the simulations and those from experiment. Under similar flow conditions the pressure drop values obtained from experiment are about 12% more than the values. The difference may be due to lack of wall effects applied during simulations, which is present during experiment. Also the entrance effects may also be a chance for the deviation from experimental values.

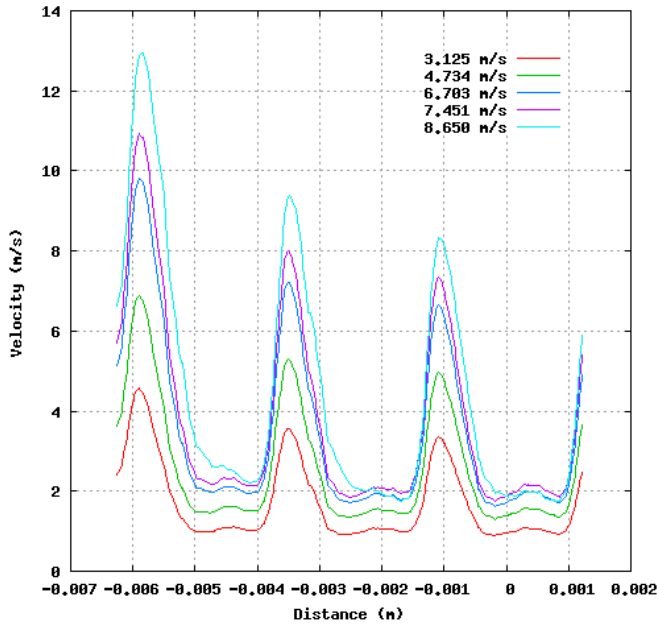


FIGURE 12. VELOCITY DISTRIBUTION

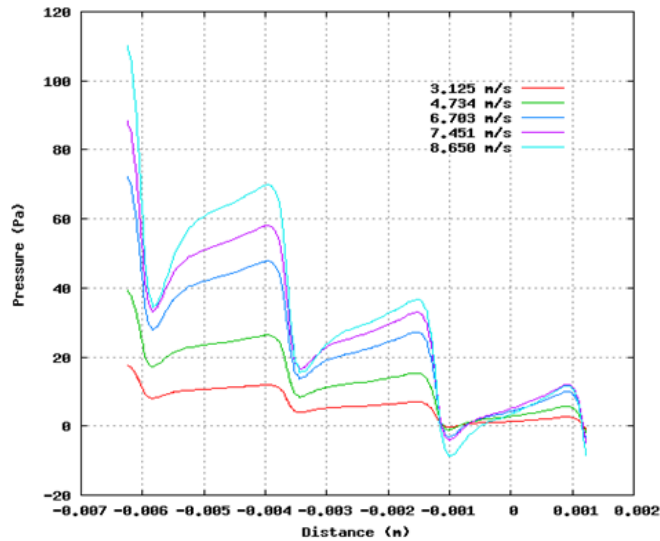


FIGURE 13. PRESSURE DISTRIBUTION

Fig.12 shows the velocity distribution along the flow direction. The velocity increases when the flow reaches near the ligaments due to the formation of wakes and it decreases after crossing the ligaments as the wakes are disappear in the flow. This can be clearly visualized from the plot shown in fig.12. The velocity increases three times in the figure because metal foam is modeled with three unit cells. The pressure distribution is shown in fig.13. The pressure increases to some extent after crossing the ligaments and the trend repeats for three times due to the presence of three unit cells.

CONCLUSIONS

The CFD simulation was done in ANSYS workbench v.14 using fluent as solver. The modeling of unit cell was done in Solidworks 2010 and Design modeler in workbench. The simulation was done with different flow velocities and pressure drop across the metal foams is calculated. During simulation the boundary conditions applied are same as that applied for the experiment conducted at National Aerospace Laboratories, Bangalore. Initially the two turbulence models i.e. standard k- ϵ and realizable k- ϵ turbulence models are tried to get the solution, among which the standard k- ϵ turbulence model gives the better solution, which is nearer to the experimental values. Hence, the standard k- ϵ turbulence model is used for all the models for the CFD simulation.

From the results it is concluded that the pressure drop increases with the increase in velocity. This result is validated with the results obtained from the experiment conducted at National Aerospace Laboratories, Bangalore. During the experimental work the pressure drop obtained for flow velocity 6.703m/s is 0.0415 PSI and during numerical validation the pressure drop obtained is 0.0361 PSI.

NOMENCLATURE

dp	Pressure drop
dx	Unit length
f	Foam coefficient
k	Turbulent kinetic energy
K	Permeability
ppi	Pores per inch
v	Fluid velocity
μ	Dynamic viscosity
ρ	Density
ϵ	Turbulent dissipation rate

ACKNOWLEDGMENTS

The authors wish to thank Head, Propulsion Division and Director, NAL, Bangalore for permitting this work to be done with the data from NAL. Authors would like to thank all the members and to all others who have directly or indirectly helped us in carrying out this work.

REFERENCES

1. Brown G O, Henry Darcy and the making of a law, *Water Resources Research*, 38 (2002) 1106.
2. Craig T Simmons, Henry Darcy: Immortalized by his scientific legacy, *Hydrogeology Journal* 16 (2008) 1023–1038.
3. Lage J L, Antohe B V, 2000, Darcy's Experiments and the Deviation to Nonlinear Flow Regime. *Journal of Fluids Engineering*, 122, pp. 619-625.
4. A. Bhattacharya, V.V Calmidi, R.L Mahajan, Thermophysical properties of high porosity metal foams, *International Journal of Heat and Mass Transfer* 45 (2002) 1017-1031.
5. K. Boomsma, D. Poulikakos, F. Zwick, Metal foams as compact high performance heat exchangers, *International Journal of Mechanics of Materials* 35 (2003) 1161- 1176.
6. Calmidi V.V and Mahajan R.L, Forced convection in High Porosity Metal foams, *Journal of Heat Transfer*, 122 (2000) 557-565.
7. A.J.Fuller, T. Kim, H.P. Hodson, and T.J.Lu, Measurement and interpretation of the heat transfer coefficients of metal foams, *Journal of Mechanical Engineering Science* 219 (2003) 183-191.
8. J.F. Liu, W.T. Wu, W.C. Chiu, W.H. Hsieh, Measurement and correlation of friction characteristic of flow through foam matrixes, *Experimental Thermal and Fluid Science* 30 (2006) 329–336.
9. Joo-Suk Noh, Kye Bock Lee, Chung Gu Lee, Pressure loss and forced convective heat transfer in an annulus filled with aluminum foam, *International Communications in Heat and Mass Transfer* 33 (2006) 434–444.
10. Simone Mancin, Claudio Zilio, Alberto Cavallini, Luisa Rossetto, Pressure drop during air flow in aluminum foams, *International Journal of Heat and Mass Transfer* 53 (2010) 3121-3130.
11. Vignes-Adler M, Weaire D, 2008, New foams: Fresh challenges and opportunities, *Current Opinion in Colloid and Interface Science*, 13, pp.141-149.
12. K. Boomsma, D. Poulikakos, Y Ventikos, Simulations of flow through open cell metal foams using an idealized periodic cell structure, *International Journal of Heat and Fluid Flow* 24 (2003) 825-834.
13. Shankar Krishnan, Jayathi Y.Murthy, Suresh V Garimella, Direct simulation of transport in Open-cell Metal foam, *Journal of Heat Transfer*, 128(2006) 793-799.
14. A. Kopanidis, A. Theodorakakos, E. Gavaises, D. Bouris, 3D numerical simulation of flow and conjugate heat transfer through a pore scale model of high porosity open cell metal foam, *International Journal of Heat and Mass Transfer* 53 (2010) 2539–2550.
15. Mo Bai, J.N. Chung, Analytical and Numerical prediction of heat transfer and pressure drop in open-cell metal foams, *International Journal of Thermal Sciences* 50 (2011) 869-880.
16. Vinod J Lonkar, J Felix, Y Giridhar Babu, S Ramamurthy, K V Sreenivas Rao, Characterization of Metal Foams for Thermal Performance, *21st National and 10th ISHMT-ASME Heat and Mass Transfer Conference* (2011).
17. M.F.Ashby, A. Evans, N.A. Fleck, L.J. Gibson, J.W. Hutchinson, H.N.G. Wadly, *Metal Foam- A Design Guide*, 2000.