Synopsis

Ketan Yogi

Department of Mechanical Engineering, IIT Bomaby

Study of fluid flow in porous metal foams using openFoam

This research migration project is about numerical simulations of the fluid flow in a channel filled with an open-cell metal foam using OpneFOAM. The objective of the study is to numerically investigated the pressure drop in the channel filled with porous metal foam. The simulation is carried out for three aluminium metal foam samples having pore density of 10, 20 and 40 PPI (pores per inch). The geometry and mesh are defined using blockMesh utility. A grid independence study is performed to remove the dependencies of the solution with the mesh. A steady-state "porousSimpleFoam" solver is used. The geometry of a 2D rectangular channel filled with porous metal foam is shown in Fig. 1 with the dimensions.
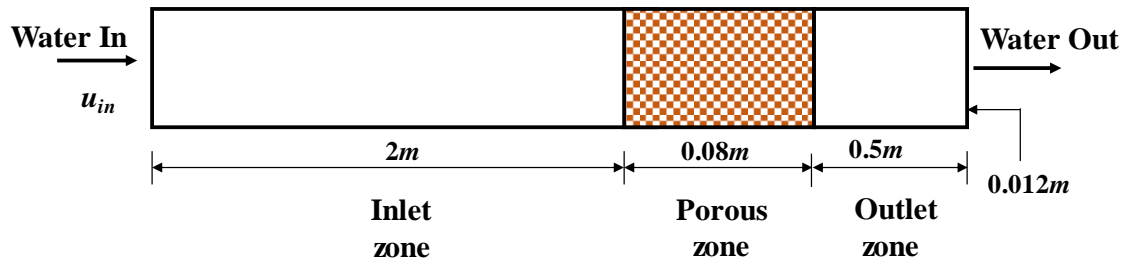

Fig.1. Geometry of 2D rectangular channel filled with porous foam

The fluid (water @ 20℃) with uniform velocity enters at the inlet face passes through porous region and exiting from the outlet face. The simulation is carried out for inlet velocities ranging from 0.01 to 0.40 *m/s*. The numerical results of pressure drop plotted against experimental data reported by Boomsma and Poulikakos [1] to observe the deviation of simulation results.

References

[1] K. Boomsma, D. Poulikakos, "The effects of compression and pore size variations on the liquid flow characteristics in metal foams", Journal of Fluids Engineering, Volume 124 (2002), pp. 124-133

URL: https://asmedigitalcollection.asme.org/fluidsengineering/article/124/1/263/462786/The-Effects-of-Compression-and-Pore-Size.DOI: 10.1115/1.1429637.

1. Introduction

A metallic porous foam is a cellular structure consisting of solid metal with gas-filled pores comprising a large portion of the volume. Metal foams are a new class of material with low densities and a high surface to volume ratio. Metallic foams offer interesting perspectives due to the combination of properties that are related to the metallic character on the one and the porous structure on the other hand. The important property of metal foams is high porosity. In metal foam typically only five to twenty percent of the volume is the base metal, its low density and high porosity making them ultralight. Metallic foams typically retain some physical properties of their base material foam made from non-flammable metal remains non-flammable and can generally be recycled as the base material. Its coefficient of thermal expansion is similar while thermal conductivity is likely reduced.

The metal foam is defined by various parameters such as mean pore diameter, pores per inch (PPI), porosity, and strand thickness. The porosity (ε) is defined as the ratio of the volume occupied by the pores and the total volume of the porous medium. The average pore diameter (d_p) is defined as the average diameter of the sphere that occupies a complete volume of a given pore measured at different points in the metal foam. The average strand thickness (t_s) is defined as the average thickness of the metal strand measured at different points on the open-cell metal foam. The surface area per unit volume of solid metal foam (S_B) is defined as the ratio of the total surface area of the solid phase of metal foam and the total volume occupied by the solid-phase of metal foam.

1.1 Classification of the metal foam

Metal foam is classified in Open-cell metal foam and Closed-cell metal foam based on its pore structure. Metal foams with pores that are completely sealed are defined as closed-cell metal foam. On the other hand, when pores are interconnected then it is defined as open-cell metal foam. The open-celled metal foam is also called a metal sponge. Closed-cell foams retain the fire resistance and recycling potential of other metallic foams but add the property of floatation in water. Open-cell metal foam and close cess metal foam are shown in Fig.2 (a) and (b) respectively.

1.2 Motivation for the work

Metal foams have ideal combinations of physical, mechanical and thermal properties, such as high stiffness, flow permeability and thermal conductivity. They have a huge importance in heat transfer

due to their huge surface area per unit volume and their ability to provide vigorous mixing of fluids, open-celled foams are excellent heat transfer cores that can be implemented in various thermal systems. However, the enhancement with the metal foam comes with the penalty of increase in the pumping power. The presence of the metal foam offers additional pressure drop to the flow resulting in the increase in the pumping power.
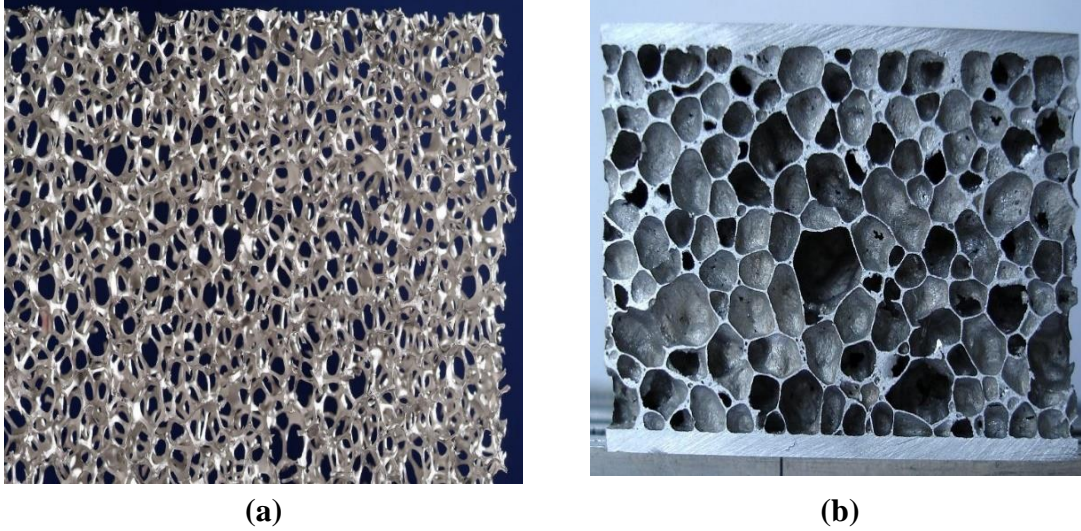

(a)   **(b)**

Fig. 1.2 (a) Open-cell metal foam[2] (b) Closed-cell metal foam[3]

In the last 150 years, different models have been developed to characterize the fluid flow in a porous matrix on the basis of macroscopically measurable flow quantities. Darcy was the first who studied the pressure drop in the porous media and established the well-known Darcy's law which states that the pressure-drop per unit length for a flow through a porous medium is proportional to the product of the fluid velocity and the dynamic viscosity and inversely proportional to the permeability [4].

$$\frac{\Delta P}{L} = \frac{\mu}{K}u \tag{1}$$

Where, u is the Dacian velocity. However, Darcy's law is applicable only for relatively low velocity flows, where the permeability-based Reynolds number is small ($Re_K = \rho\sqrt{K}u/\mu < O(1)$). Diedericks and Du Plessis [5] reported that with the increase in the flow velocity, the form drag becomes more dominant and must be considered for an accurate description of the pressure-

drop Later, Dupit [6] modified equation 1 by accounting the additional foam drag and gave a quadratic relation of the pressure drop.

$$\frac{\Delta P}{L} = \frac{\mu}{K}u + \rho C u^2 \tag{2}$$

Equation 2 is known as Forchheirmer Equation where K is the permeability of the porous medium and C is a foam drag coefficient. Hence, the permeability and inertial co-efficient of the porous media are the key parameters to estimate the pressure drop in the porous material. Many have reported the measurement of the pressure drop within the porous media like packed bed of sphear. Boomsma and Poulikakos [1] experimentally measure the pressure drop across the channel filled with an open cell metal foam. They have studied the compression and pore size variation on the flow characteristics. The have used three aluminium open cell metal foam (80mm × 38mm × 12 mm) having same porosity of around 92 % and different pore densities. Figure 3 shows the picture of 10 PPI aluminium metal foam with its microscopic pore structure view. The characteristics of the aluminium metal foam is tabulated in Table1.
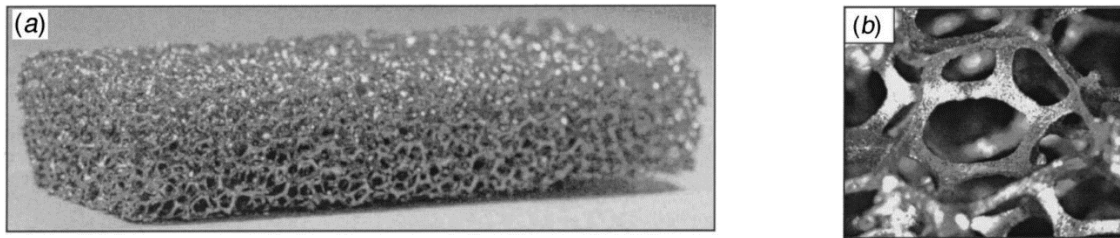

Fig. 3 (a) Aluminum foam block (92% porous, 10 PPI, 6.9 mm pore diameter) (b) magnified view of a uncompressed Al foam [1]

Table 1. Characteristics of the metal foam samples used in Boomsma and Poulikakos [1]

Foam	Pore diameter (mm)	Specific Surface Area (m^2/m^3)	Measured Porosity (%)	Permeability ($10^{-10}m^2$)	Inertial coefficient (m^{-1})
10 PPI	6.9	820	92.1	3529	120
20 PPI	3.6	1700	92.0	1089	239
40 PPI	2.3	2700	92.8	712	362

The pressure drop across the metal foam samples is measured using the experimental setup show in Fig.4. The metal foam sample is placed in the metal foam test housing. The water (at 20℃) from

the water recirculator enters at the metal foam test housing. The flow of the water is regulated using a regulator valve located at the upstream of the metal foam test housing. The flow rate of the water is measured with the rotameter. Water pass through the metal foam sample is again recirculated in the close loop. The pressure at the inlet and outlet of the foam sample is measured using pressure transducers connected to data acquisition system. The positon of the pressure transducers are shown in Fig.4(d) by black dots.
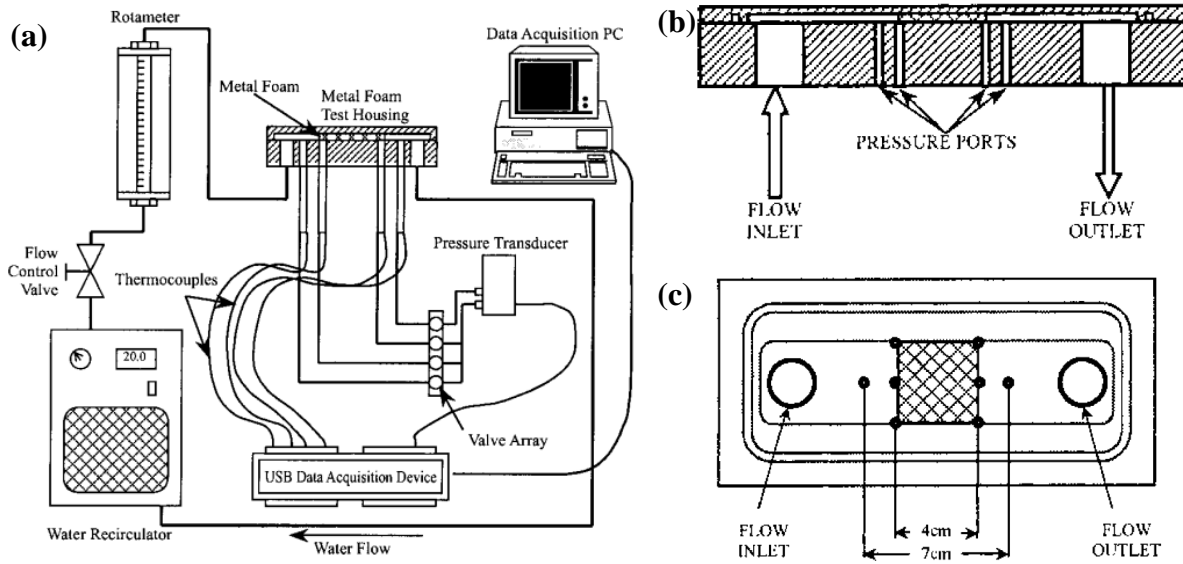

Fig. 4 (a) Experimental setup (b) side view and (c) top view of metal foam test housing [1]

The variation of the pressure drop along the metal length for different velocities for 10, 20 and 40 PPI metal foam is shown in Fig. 5. It is observed that the pressure drop within the metal foam having same porosity varies with the variation of the pore density. The metal foam having 40 PPI pore density shows maximum pressure drop followed by 20 and 10 PPI metal foam.

The permeability and inertial coefficient foam drag obtained by curve fitting of the pressure drop verses velocity data is reported in table 1. The permeability and inertial coefficient can be used to get the pressure drop within the metal foam numerically. Motivating from the work of the Boomsma and Poulikakos[1], the present work focus on the study of the fluid flow in the porous metal foam using openFoam tool.
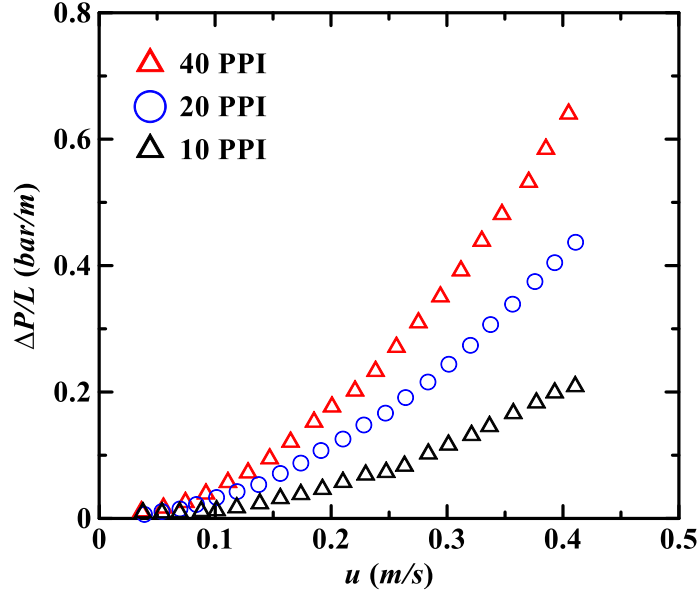
**Fig. 5 Pressure-drop versus fluid flow velocity for the three metal foams
from Boomsma and Poulikakos [1]**

1.3 Objectives of the present work

Motivating from the work of Boomsma and Poulikakos[1], the present work focus on the study of the fluid flow in the porous metal foam using openFoam tool. The objectives of the present work are

- Numerically investigated the pressure drop in the channel filled with porous metal foam in openFoam.
- Grid independence study refers to approaching an optimal grid size which minimises the computational time and also simultaneously removes the dependencies of the solution with the mesh.
- Numerical results are plotted against experimental data to observe the deviation of simulation results

2. Governing equations

Porous media is modeled by attenuating the time derivative and by adding a sink term to the Navier-Stokes equations.

$$\frac{\partial}{\partial x}(\gamma \rho u_i) + u_j \frac{\partial}{\partial x_j}(\rho u_i) = -\frac{\partial p}{\partial x_i} + \mu \frac{\partial \tau_{ij}}{\partial x_j} + S_i \tag{3}$$

The value of γ should be between 0 and 1, where the latter is a complete porosity. The source term S_i is composed of two parts, a viscous loss term and an inertial loss term, creating a pressure drop that is proportional to the velocity and velocity squared, respectively. The source term is represented as equation 4 also known as Darcy-Forchheimer equation.

$$S_i = -\left(\mu D_{ij} + \frac{1}{2}\rho|u_{kk}|F_{ij}\right)u_i \tag{4}$$

In the case of simple homogeneous porous media it becomes

$$S_i = -\left(\mu D + \frac{1}{2}\rho|u_{jj}|F\right)u_i \tag{5}$$

where D_{ij} and F_{ij} are represented as the scalars D and F.

The source term can also be modeled as a power law of the velocity magnitude, i.e.

$$S_i = -\rho C_0|u_i|^{(C_1-1)/2} \tag{6}$$

where C_0 and C_1 are user defined empirical coefficients.

3. Simulation Procedure

3.1 Geometry

Figure 6 shows the 2D rectangular channel filled with porous metal foam which replicate the metal foam test housing discussed in figure 4 (b) and (c) is taken as geometry for the simulation. The width and height of the channel kept same as that of metal foam. Hence, the hydraulic diameter of the channel is 0.018 m.
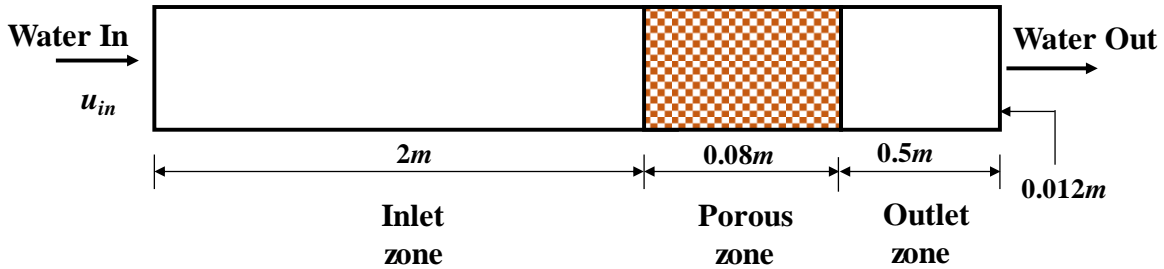

Fig.6. Geometry of 2D rectangular channel filled with porous foam

The geometry consists of three rectangular zones: inlet zone, porous zone and outlet zone. Here, the flow is assumed fully developed and in order to remove the entrance effects in the simulation, entrance length can be estimated around 2 m using equations 7 for laminar range of the flow.

$$L_e = 0.06 \times Re \times d_h \tag{7}$$

The length of the porous zone is 0.08 *m*. The length of outlet zone is 0.5 *m* and its length does not affect the solution to a greater extent.

3.2 Mesh

The block mesh utility is adopted to make the geometry and meshing. A structured grid has been used. Once the geometry is made, a proper grid independence study is performed in meshing of the geometry. Grid independence study refers to approaching an optimal grid size which minimises the computational time and also simultaneously removes the dependencies of the solution with the mesh. The grid independence test is performed by varying the number of elements in both x (axial direction) and y (radial direction) direction in the mesh. First, keeping the number of the element of the radial direction fixed, the number of the elements along the axial direction changed until the pressure at the inlet of the porous domain gets unchanged. Figure 7 shows the variation of the pressure at the inlet of the porous domain with the change in the number of cells in the axial direction. It is observed that beyond 27000 cells (corresponding cell size is 1.33 *mm*), the pressure at the entry of the porous domain is not changing. Hence, in the axial direction the 1.33 *mm* cell size is chosen.
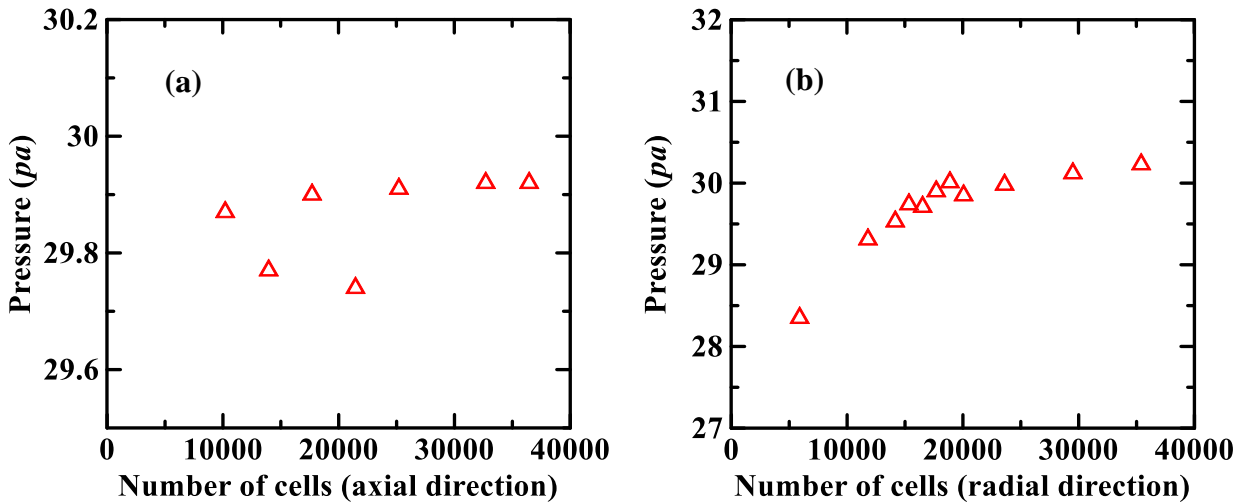

Fig.7. Variation of Total pressure with change in the total number of cells in (a) axial and (b) radial direction

Similarly, keeping the fixed cell size of 1.33 mm is axial direction, the number of cells in the redial direction changed till the pressure at the entry of the porous domain get unchanged. It is observed that beyond 18928 cells (corresponding cell size is 0.61*mm*), the pressure at the entry of the porous

domain is not changing. Hence, in the radial direction the $0.61mm$ cell size is chosen. The final mesh generated using the blokMesh have utility the mesh is generated having 82840 cells and 335522 faces.

3.3 Boundary conditions

There are four boundaries that required boundary conditions: An inlet, outlet, upper Wall and lower Wall. The other two boundaries are front and back; that set as empty boundaries. At inlet, uniform specified velocity boundary condition is given. At the outlet, a zerogredient pressure boundary condition is given as the flow is considered as exiting to ambient. All other boundaries are considered as no slip/wall condition.

3.4 Solver

A steady-state for incompressible, turbulent flow-based porousSimpleFoam solver is used to run governing equations in the discretized domain. The porousSimpleFoam solver uses SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm to evaluate NS equations. The solver follows a segregated solution strategy. This means that the equations for each variable characterizing the system (the velocity u, the pressure p, and the variables characterizing turbulence) are solved sequentially. The convergence criteria is set to be 10^5 for all the simulations carried out in the present study.

4. Results and Discussions

The pressure drop across a rectangular channel filled with a porous metal foam is numerically investigated using openFOAM. Aluminium metal foam having pore density of 10, 20 and 40 PPI (pores per inch) is simulated using porousSimpleFoam solver. The simulation is carried out for inlet velocities ranging from 0.01 to 0.40 m/s.

4.1 Velocity and pressure contour

Figure 8 shows the velocity contour for 40 PPI aluminium metal foam of 0.04 m/s inlet velocity. Here, the metal foam is located in between the streamwise distance of 2 to 2.08 m. The velocity contour plot shown in Fig.8a suggest that the flow is fully developed at the entry of the foam. Within the foam the velocity drops and a uniform distribution of velocity is observed. The velocity increases after the porous region to match the constant mass flow criteria. The pressure contour plot shown in Fig. 8b shows that within the porous zone (2 to 2.08m stream wise location), a drop

in the pressure is observed. However, the pressure is spanwise direction (vertical Y direction) constant the each streamwise location.
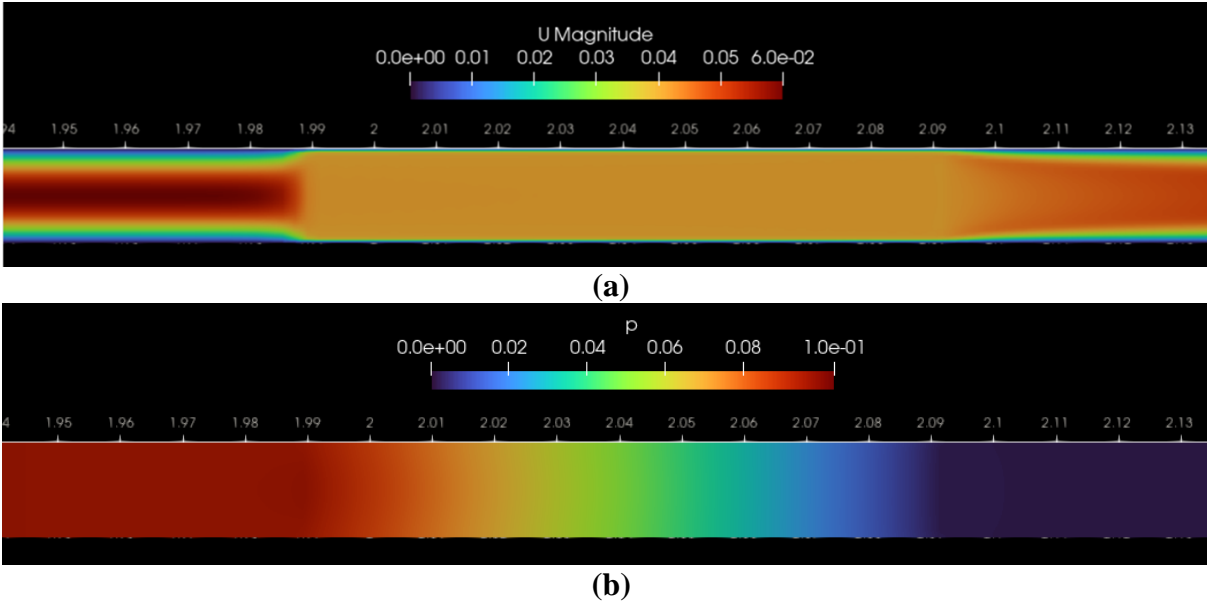

(a)

(b)

Fig.8 (a) velocity contour plot (b) pressure contour plot

4.2 Variation of the pressure and velocity in the streamwise direction

Figure 9a shows the centreline velocity variation in the axial direction for 10 PPI aluminium metal foam for different inlet velocities. The axial velocity increases from the inlet to some streamwise distance till 0.25 to 0.5 depending on the inlet velocity.

Beyond streamwise distance of 0.5, the axial velocity remains unchanged in the inlet zone of the test section suggesting the flow is fully developed. The velocity suddenly drop just entering to the porous zone. This is might be because of the porous foam offers additional resistance of the flow. Within the porous zone the velocity remains unchanged. This might be because of the foam make flow uniform within the porous zone. Beyond the porous zone the velocity increases till outlet of the domain. Figure 9b shows the variation of the pressure in the streamwise direction for different inlet velocities. A sharp drop in the pressure is observed within the porous zone due to presence of the porous foam. The drop in the pressure is decreases with the decrease in the inlet velocity.
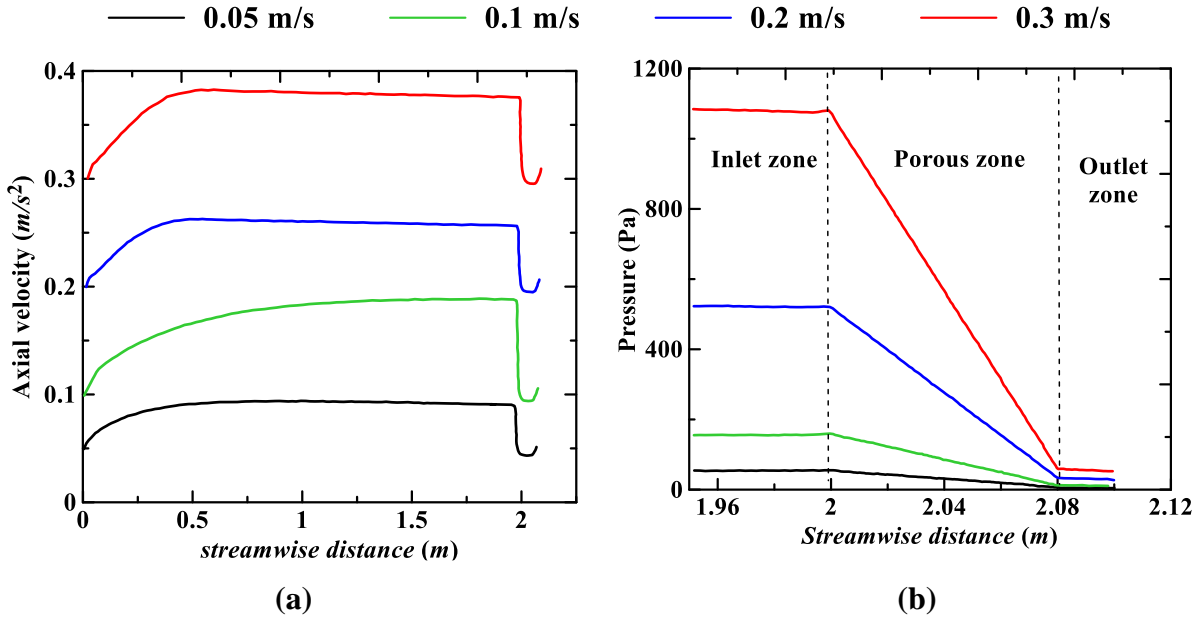
Fig.9. Variation of (a) axial velocity and (b) pressure with streamwise distance for 10 PPI aluminium metal foam

4.2 Variation of the pressure and velocity in the streamwise direction

The pressure drop across metal foam is the pressure difference between the inlet and outlet of the porous zone as represented in equation 8.

$$\Delta P = P_{@2.8m} - P_{@2.0m} \tag{8}$$

The pressure drop for 10, 20 and 40 PPI metal foam calculated using equation 8 is compared with the experimental data of Boomsma and Poulikakos [1] as shown in Fig.10. The pressure drop obtained from numerical analysis is matched reasonably well with the experimental data for all metal foam. Hence, the present numerical analysis of the channel filled with open cell metal foam using openFoam validates.
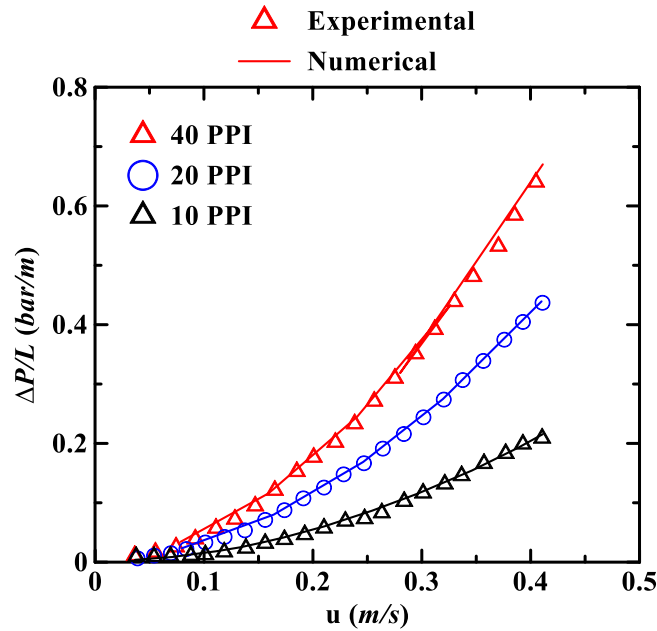
Fig.10 Pressure drop versus fluid flow velocity for the three uncompressed metal foams.

5. Conclusion

This research migration project is about numerical simulation of a 2D rectangular channel filled with open cell metal foam using openFoam. Simulation is performed for aluminium foam having 10, 20 and 40 PPI pore density. The pressure drop within the metal foam obtained from numerical study matches well with the experimental work of Boomsma and Poulikakos [1]. Hence, the presence numerical work validates with the available literature.

6. References

[1] K. Boomsma, D. Poulikakos, "The effects of compression and pore size variations on the liquid flow characteristics in metal foams", Journal of Fluids Engineering, Volume 124 (2002), pp. 124-133

URL: https://asmedigitalcollection.asme.org/fluidsengineering/article/124/1/263/462786/The-Effects-of-Compression-and-Pore-Size.DOI: 10.1115/1.1429637.

(2) http://www.wikiwand.com/en/Aluminium_foam_sandwich

[3] https://www.ifam.fraunhofer.de/content/dam/ifam/en/documents/dd/Infobl%C3%A4tter/open_cell_metal_foams_fraunhofer_ifam_dresden.pdf

[4] Darcy, H., 1856, Les Fontaines Publiques de la ville de Dijon., Dalmont, Paris.

[5] Diedericks, G. P. J., and Du Plessis, J. P., 1997, ''Modeling of flow through homogeneous foams,'' Mathematical Engineering in Industry, 6, pp. 133–154.

[6] Dupuit, J., 1863, Etudes Theoriques et Pratiques sur le Mouvement des Eaux., Dunod, Paris