FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
April 16, 2022

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.

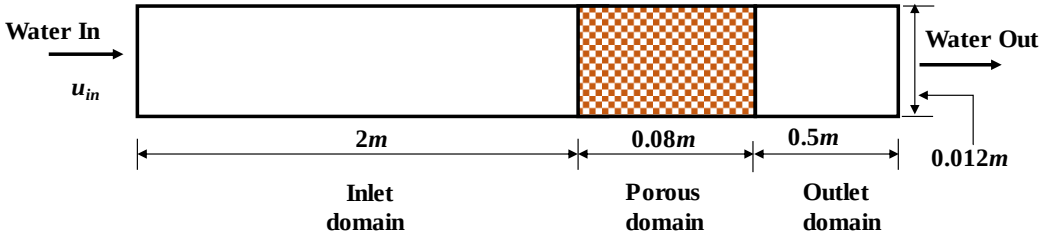

Fig.1. Geometry of 2D rectangular channel filled with porous foam

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.