11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20–25, 2014, Barcelona, Spain

# CHARACTERIZATION OF FLOW REGIMES AND HEAT TRANSFER INSIDE KELVIN-CELL TYPE FOAMS BY MEANS OF OPENFOAM

**A. Della Torre [1], G. Montenegro[1], F. Brusiani [2] and G.M. Bianchi[2]**

[1] Politecnico di Milano - Department of Energy, Via Lambruschini 4, 20156 Milan, Italy
[2] University of Bologna, Faculty of Mechanical Engineering,Viale Risorgimento 2, Bologna, Italy

**Key words:** *Open cell foam, kelvin cell, pressure drop, heat transfer.*

In the last years open-cell foams have been applied in an increasing variety of engineering fields, ranging from the manufacture of structural components to biomedical applications. With regards to the energy field the structure of open-cell foams, which is characterized by a high specific surface area, is advantageous in a wide range of applications, e.g. in the design of compact heat exchangers or as catalytic substrates in after-treatment devices. The characterization of the foam properties in terms of pressure drop, heat exchange and mass transfer is therefore an issue of extreme concern when the design and the optimization of such devices is addressed.

In this work CFD is applied to investigate the phenomena occurring at the micro-

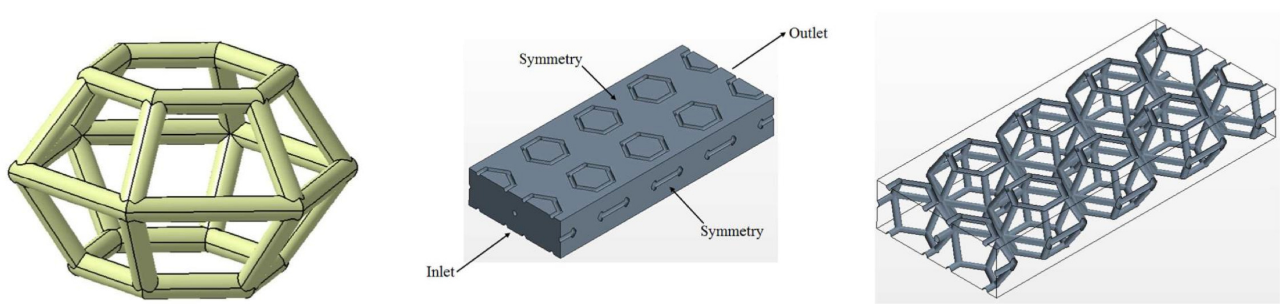


**Figure 1**: Kelvin cell cluster used for the simulations.

scale level. Kelvin cell structures are adopted to reproduce the actual foam microscopic geometry (see Figure 1). The flow inside the foam is investigated under different flow regimes, in order to study their influence on the pressure drop and on the heat transfer coefficient.

Simulations are performed to model the flow through the microstructure, in order to point out how the microscopic geometry influence the pressure drop under different flow conditions. Moreover, a conjugate heat transfer model is applied for investigating the convection between the fluid and the solid structure and the conduction in the solid. Different turbulence models are compared in order to assess considerations about the flow regime inside the foam. The goals of this first step are 1) the enhancement of the understanding of the phenomena occurring at the micro-scale and 2) the definition and extraction at the micro-scale of coefficients describing the fluid-solid interaction in term of permeability and convection properties. The results of the simulations pointed out that the main cause of pressure jump across the microstructure is due to inertial effect rather than viscous ones (Figure **??**). To further validate this aspect, a new boundary condition based of the Maxwell's equations was developed to mimic slip flow inside the microstructure.

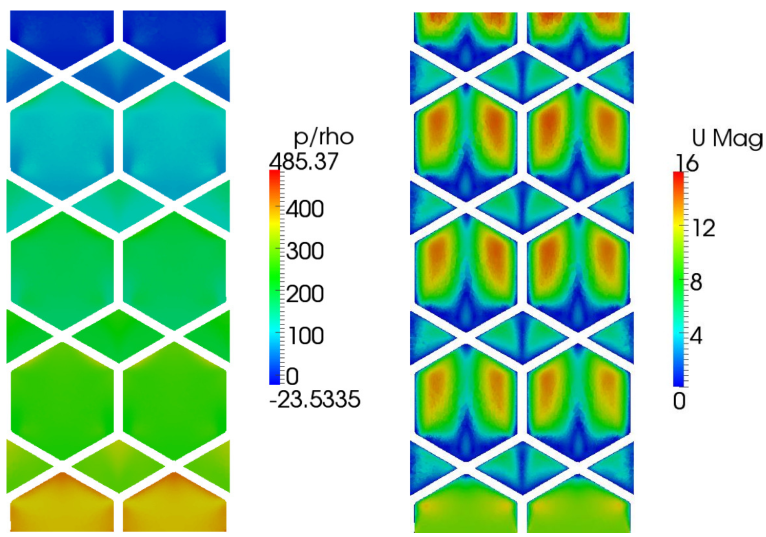


**Figure 2**: Pressure and velocity fields inside the microstructure.