Proceedings of CHT-17
ICHMT International Symposium on Advances in Computational Heat Transfer

May 28-June 1, 2017, Napoli, Italy

CHT-17-049

Thermally Developed Flow in Open-Cell Foams: A Pore-Scale Analysis

Marcello Iasiello[*,§], Salvatore Cunsolo[*,**], Nicola Bianco[*], Wilson K. S. Chiu[***] and Vincenzo Naso[*]
[*] Dipartimento di Ingegneria Industriale, Università degli Studi di Napoli Federico II, Napoli, Italy
[**] LaMCos, INSA-Lyon, UMR CNRS 5259, Villeurbanne, France
[***] Department of Mechanical Engineering, University of Connecticut, Storrs, CT, USA
[§] Corresponding author. Email: marcello.iasiello@unina.it

ABSTRACT Convection heat transfer in open cell foams plays a primary role when a significant local temperature difference exists between the solid matrix and the fluid that passes through it. Recently, it has been recognized that the flow thermally develops after a few cells, for both uniform temperature and heat flux conditions at the solid/fluid interface. Convection heat transfer in thermally developed flow at the pore scale in foams, that so far has received limited attention in the literature, is investigated numerically in this paper. The open cell foam geometry is built up with reference to Kelvin's tetrakaidecahedron foam. The velocity, pressure and temperature fields are predicted with a finite element scheme, for different porosities and Reynolds numbers. Results show that convection heat transfer has a periodical behavior through the single developed cell, with local values affected by porosity and Reynolds number. Details are given about the variation of the local convection heat transfer over the solid/fluid interface.