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

May 28-June 1, 2017, Napoli, Italy

CHT-17-023

NUMERICAL PREDICTION OF FLUID FLOW AND HEAT TRANSFER IN CROSS-FLOW MICRO HEAT EXCHANGERS

Carlo Nonino[*,§] and Stefano Savino[*]
[*]DPIA, Università degli Studi di Udine, Italy
[§]Correspondence author. Fax: +39 0432 558027, Email: carlo.nonino@uniud.it

ABSTRACT The results of CFD calculations can represent a useful complement to the experimental data on micro heat exchangers since they allow the designer to fetch details of the thermal fields inside these micro devices. On the other hand, obtaining the numerical solution for a complete heat transfer unit is very demanding in terms of required computational resources. This is particularly true for the case of cross-flow micro heat exchangers since, due to the features of the flow patterns, the only symmetry planes that can be considered to simplify the computational domain are those parallel to both fluid streams. Therefore, as an alternative to massive CFD, a simplified FEM procedure was developed to allow the analysis of the fluid flow and heat transfer in cross-flow micro heat exchangers using ordinary workstations. A two-stage procedure is adopted on the assumption that all the thermophysical properties are constant within each microchannel and that all layers of microchannels are identical except for edge effects in the outer layers. First, an in-house FEM code for the solution of the parabolized Navier-Stokes equations is employed to compute the velocity field and the pressure drop in a single microchannel. Then, an appropriate mapping of the velocity field thus determined is used to obtain the velocity components in the fluid parts of a three-dimensional computational domain corresponding to a suitable portion of the micro heat exchanger. On this domain, the elliptic form of the equation energy is solved using another in-house FEM code.

Since in a cross-flow micro heat exchanger the microchannels of one layer are perpendicular to those of the adjacent layers, it is impossible to discretize the computational domain with a structured grid consisting of hexahedral elements that are elongated in the flow direction. Thus, the possibility of independently meshing different parts of the domain using grids that do not match at the common interface (domain decomposition) has been implemented. To this purpose, an original method has been proposed, which requires neither iterations nor the evaluation of integrals on the interface (never easy when grids do not match).

The original procedure has been subsequently extended to allow the modelling of the effects of flow maldistribution, which in cross-flow micro heat exchangers can originate from an inappropriate design of headers and manifolds and/or from the viscosity change associated with the temperature variation in the microchannels. An iterative scheme has been adopted so that the velocity field in each microchannel can be updated to account for a redistribution of the total mass flow rate on the basis of the constraint that the pressure drop must be the same for all the microchannels of each layer, even if microchannel velocities and average viscosities are different. The iterative procedure does not require new solutions of the parabolized Navier-Stokes equations. However, the mapping of the velocity field between grids with different nodal densities hinders the fulfilment of the discrete mass conservation constraint. This is restored by solving a Poisson equation for a velocity correction potential that allows the calculation of appropriate velocity corrections.

Numerical results concern validation tests and sample applications.