HEFAT 2005
4th International Conference on Heat Transfer, Fluid Mechanics, and Thermodynamics
Cairo, Egypt

Numerical and experimental investigation of flow and heat transfer in narrow channels with corrugated walls

AG Kanaris, AA Mouza, SV Paras*

Laboratory of Chemical Process and Plant Design
Department of Chemical Engineering
Aristotle University of Thessaloniki
Univ. Box 455, GR 54124 Thessaloniki, GREECE

Abstract

The purpose of this work is the numerical and experimental investigation of the flow field and of the heat transfer augmentation in conduits with corrugated walls, similar to those encountered in commercial plate heat exchangers. Although the application of this type of equipment in industry is continuously increasing, the necessary knowledge for modeling and optimization of their performance is still limited. In this study, a commercial CFD code (*CFX® 5.7*) is used to simulate the performance of a model heat exchanger comprised of stainless steel plates, following a herringbone design, assembled for single-pass counter-current flow. The ultimate goal of the CFD simulation is the study of the effect of the various design parameters (e.g. plate geometry, type of corrugations) and of fluid physical properties on the overall performance of such type of equipment.

The CFD simulation is validated using experimental data on pressure drop and overall temperature differences acquired for counter-current flow of water at both sides of the model heat exchanger for various hot fluid flow rates (corresponding to a Reynolds number range from 1500 to 2300). The temperature at various locations along the hot and the cold side of the heat exchanger is simultaneously measured by nine thermocouples. An *IR* thermography camera is used in order to obtain a temperature profile of the plate wall, and, in conjunction with the thermocouples, to acquire detailed information on the fluid temperature profile.

Since the type of flow prevailing in this type of conduits is considered to be turbulent, even for the Reynolds number range employed in this study, the most appropriate turbulence model must be chosen by taking into account both the complexity of the geometry and the restrictions induced by each computational model available in the CFD code. In this way, the *SST* (Shear-Stress Transport) model is selected, which is a combination of two well-known turbulence models, i.e. k-ε and k-ω. The computational domain used comprises a complete conduit of an existing commercial plate heat exchanger, where the hot fluid flows, whereas the cold fluid temperature distribution (obtained experimentally) is set as boundary condition on one of the conduit walls. The large size of the resulting computational grid requires the use of a parallel computing cluster of six 64-bit processors.

The results of this study, presented in terms of friction factor, wall heat flux and heat transfer coefficients, are compared with the collected experimental data and suggest that the CFD code is capable of predicting the flow and heat transfer characteristics in this kind of complex geometries. It must be also noted that the limited results of relevant experimental and theoretical works reported in the literature are found to be in fairly good agreement with the results of this study, which is currently in progress.

e-mails:

*paras@cheng.auth.gr**
mouza@cheng.auth.gr
kanaris@cheng.auth.gr

**author for correspondence*