HEFAT2005
4th International Conference on Heat Transfer, Fluid Mechanics, and Thermodynamics
Cairo, Egypt
Paper number: KA2

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

*Author for correspondence: paras@cheng.auth.gr

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, encountered in commercial plate heat exchangers. A *CFD* code 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. Experimental data on pressure drop and overall temperature differences acquired for counter-current flow of water at both sides of the model heat exchanger are used to validate the *CFD* simulation. The *SST* model (Shear-Stress Transport) is selected as the most appropriate turbulence model. The computational results, presented in terms of friction factors, local temperatures and Nusselt numbers, are compared with the experimental data. 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 limited results of relevant experimental and theoretical work reported in the literature are found to be in fairly good agreement with results from the present study, which is in progress.

INTRODUCTION

Compact heat exchangers with corrugated plates are constantly replacing conventional shell-and-tube ones mainly due to their high thermal effectiveness, close temperature approach as well as ease of inspection and cleaning [1,2]. A compact heat exchanger consists of plates embossed with some form of corrugated surface pattern and assembled being abutting, with their corrugations forming narrow passages. The heat transfer augmentation in this type of equipment is accompanied by a substantial increase in pressure drop. Consequently, due to the relatively high pressure drop, the Reynolds numbers used in this type of equipment must be lower than those for shell-and-tube heat exchangers for equivalent flow rates, so as the resulting pressure drops would be generally acceptable [1]. Moreover, when this equipment is used as a reflux condenser, the limit imposed by the onset of flooding reduces the maximum Reynolds number to a value less than 2000 [3].

The type of flow prevailing inside the narrow passages of the compact heat exchangers is still an open issue in the literature. Focke & Knibbe [4], who performed flow visualization experiments in narrow passages with corrugated walls, deduce that the flow patterns are complex, due to the existence of secondary swirling motions along the furrows of their test section and suggest that the local flow structure controls the heat transfer process. Ciofalo et al. [5], in a comprehensive review article concerning modelling heat transfer in narrow flow passages, state that, for the Reynolds number range 1500-3000, transitional flow is encountered, a kind of flow among the most difficult to simulate by conventional turbulence models. On the other hand, Shah & Wanniarachchi [1] declare that there is evidence of turbulent flow even for the Reynolds number range 100-1500. The above statement is supported by Vlasogiannis et al. [6], whose experiments in a plate heat exchanger verify that the flow is turbulent for Re>650. Lioumbas et al. [7] studied experimentally the flow in narrow passages during counter-current gas-liquid flow and suggest that the flow exhibits the basic features of turbulent flow even for the relatively low gas Reynolds numbers tested (500<Re<1200).

The requirement for detailed and accurate measurement of the design parameters (e.g. temperature, pressure and velocity fields) is very difficult to be achieved, because the flow passages in compact heat exchangers are complex in geometry and of relatively small dimensions. Computational Fluid Dynamics (*CFD*) can be considered an effective tool for estimating the momentum and heat transfer rate in this type of heat exchangers and evaluating their performance. However, the accuracy of the calculations depends on the choice of the most appropriate flow model for *CFD* simulation. The most common two-equation turbulence model, based on the equations for the turbulence energy k and its dissipation ε, is the k-ε model [8]. To calculate the boundary layer, either "wall functions" are used, overriding the calculation of k and ε in the wall adjacent nodes [8], or integration is performed to the surface, using a "low turbulent Reynolds (*low-Re*) k-ε" model [9]. Menter & Esch [9] state that in standard k-ε the wall shear stress and heat flux are overpredicted (especially for the lower range of the Reynolds number encountered in this kind of equipment) due to the overprediction of the turbulent length scale in the flow reattachment region, which occurs on the corrugated surfaces in these geometries. Moreover, the standard k-ε model requires the grid to be coarse near the wall based on the value of y^+=11 [9,10], which is difficult to accomplish in confined geometries. On the other hand, the low-Re k-ε model, which uses "*dumping functions*" near the wall [8,9] to avoid the use of wall functions, is not considered capable to predict the flow parameters in the complex geometry of a corrugated narrow channel [5], requires finer mesh near the wall, is computationally expensive compared to the standard *k-ε* model and is unstable in convergence.

An alternative to *k-ε* model is the *k-ω* model developed by Wilcox [11], which uses the turbulence frequency *ω* instead of the turbulence dissipation *ε*. The *k-ω* model, albeit is more robust and does not require very fine grid near the wall [8], is sensitive to the free stream values of turbulence frequency *ω* outside the boundary layer. A combination of the two models is the *SST* (Shear-Stress Transport) model, which, by employing specific "*blending functions*", activates the Wilcox model near the wall and the *k-ε* model for the rest of the flow [9] and in this way it benefits from the advantages of both models.

Efforts have already been made towards the effective simulation of a plate heat exchanger [5,12,13]. Due to the modular nature of a compact heat exchanger, a common practice is to think of it as composed of a large number of Representative Element Units, *RES* (i.e. of unit *cells*), and obtain results by using a single cell as the computational domain and imposing periodicity conditions across its boundaries. However, the validity of this assumption is considered an open issue in the literature [5].

In this paper, a *CFD* code is employed to study the flow and thermal characteristics within the complicated passages of a plate heat exchanger. The computational domain used includes a ***complete*** conduit of an existing plate heat exchanger. The *CFD* results are compared with experimental data.

EXPERIMENTAL SET-UP AND PROCEDURE

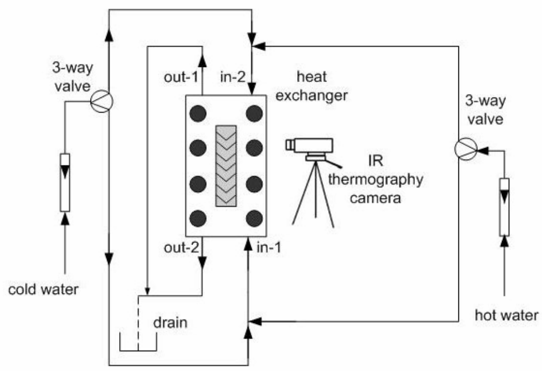
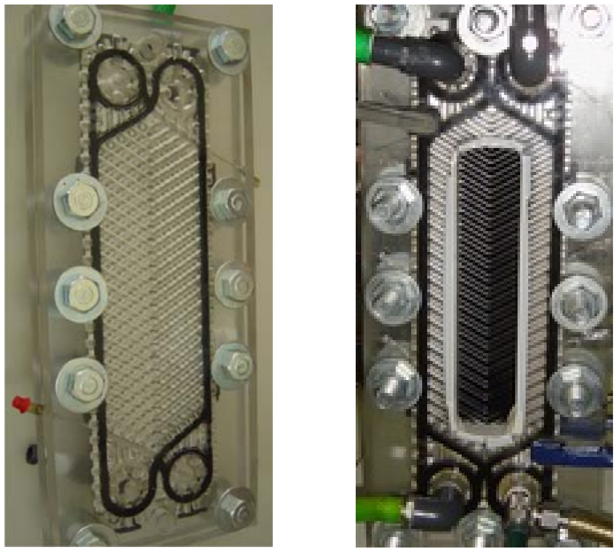


Figure 1. The experimental set-up.

The experimental set-up consists of a model compact heat exchanger (***Fig. 1***) and the necessary piping to maintain the hot and cold fluid flow. The two streams are in counter-current flow. This simple model heat exchanger comprises three corrugated plates that form two isolated fluid paths (conduits) and is similar to the one used by Vlasogiannis et al. [6]. Two of the plates are stainless-steel, whereas, in order to facilitate the visualization of the flow, the third one is replaced by a Plexiglas® plate (***Fig. 2a***), where the corrugation pattern is embossed by a lithographic technique. On the opposite side of the model heat exchanger the usual metal cover is substituted by a Plexiglas® plate, placed on top of the corrugated metal plates (***Fig. 2b***). A couple of 3-way valves are used to alternatively direct the hot/cold fluid flow to the desired conduit.

The temperature at various locations along the hot and the cold side of the heat exchanger is simultaneously measured by nine thermocouples implanted into the conduit and the data are collected by a *PC* through a terminal board with cold junction compensation and an *A/D* card (*PCI 1710HG*). An *IR* thermography camera (*FLIR Agema 570*) is also used to obtain the detailed temperature profile of the fluid inside the conduit. This camera is a device that performs non-intrusive temperature measurements by capturing the infrared radiation emitted by a body and converting it to the corresponding temperature at a point of a surface. Due to the high thermal conductivity of the stainless steel, it can be presumed that the temperature distribution over the external surface of a heat exchanger plate is practically equal to that of the fluid inside the narrow conduit.



(a) (b)

Figure 2. The model compact heat exchanger: a) corrugated Plexiglas® plate and b) cover plate with "window".

Because the Plexiglas® cover plate (***Fig. 2b***) is acting as a thermal insulator, i.e. it blocks the infrared radiation emitted by the steel plate, a "*window"* is created by cutting out a rectangular section of the Plexiglas® cover plate and exposing approximately 50% of the corrugated plate surface. In addition, the steel plate is painted black to reduce light reflection on the metal surface of the corrugated plate. The detailed geometric characteristics of the corrugated plate are given in ***Table 1***.

Experiments were conducted for various hot fluid flow rates corresponding to a Reynolds number range from 1500 to 2300. The Reynolds number used hereafter is defined as:

$$Re_i = \frac{u_i D_h \rho_i}{\mu_i} \tag{1}$$

and by substituting the hydraulic diameter

$$D_h = \frac{2wb_c}{w+b_c} \simeq 2b_c \tag{2}$$

and taking into account that

$$u_i = V_i / A_f \quad (3)$$

and, for the conduit,

$$A_f = wb_c \quad (4)$$

where V_i is the volumetric flow rate of stream i, the Reynolds number is given by:

$$\mathrm{Re}_i = \frac{2\rho_i V_i}{w\mu_i} \quad (5)$$

A constant inlet temperature (~40°C) for the hot fluid was maintained throughout each set of experiments. The temperature of the cold fluid was also constant (~18°C) and for each hot fluid flow rate several cold fluid flow rates were used. For each couple of hot/cold fluid flow rates the pressure drop of hot/cold fluid side and the temperature at the nine thermocouple locations is recorded. A thermography image of the temperature distribution over the hot/cold fluid plate surface is also obtained by alternatively directing the hot/cold fluid flow to the appropriate conduit. Tap water was used throughout this study both as hot and cold fluid. The pressure drop is measured using a differential pressure transducer (*Validyne*) between two taps located at the entrance and the exit of one of the two conduits.

Table 1. Geometric characteristics of a corrugated plate.

Plate length	0.430 m
Plate width	0.123 m
Mean spacing between plates, b_c	0.024 m
Corrugation angle with respect to *z*-axis	60 deg
Port-to-port length	0.352 m
Plate width inside gasket, *w*	0.100 m
Mean flow cross-section per channel, A_f	2.4×10^{-4} m^2
Heat transfer area	3.2×10^{-2} m^2

COMPUTATIONAL PROCEDURE

A commercial *CFD* code, namely *CFX® 5.7*, was employed to simulate the flow and to obtain results concerning the flow pattern and the heat transfer characteristics inside ***one*** of the heat exchanger conduits. The complexity of the conduit combined with the need for fine grid inside the furrows leads to a high increase of computational demands. For the needs of the present study a parallel computing cluster, which consists of six *AMD64* processors and a total of 6GB *RAM*, was employed. The *MPICH* protocol, running under *Gentoo Linux*, was used for the parallel messaging, while the cluster nodes are using Gbit Ethernet network connections for increased speed.

The grid used for the simulation is an unstructured mesh consisting of tetrahedral and prism elements. To create a computational grid that would efficiently describe the complexity of the narrow passages, tetrahedral elements were created using the Delaunay method [10]. A layer of prismatic elements was imposed in the vicinity of the walls to facilitate the boundary layer calculations. This procedure, also known as "*inflation*", is necessary for confined geometries [10]. The final grid, which was chosen by a grid dependence study, comprises 2.6 million elements.

The hot fluid mass flow rate is given as boundary condition at the conduit inlet, while a constant pressure boundary is imposed at the fluid outlet. The flow rates examined correspond to the Reynolds number range employed in the experiments. The outer conduit side wall is considered adiabatic, while a temperature boundary (either a constant wall temperature or a temperature profile representing the cold fluid temperature distribution) is applied at the internal wall (i.e. the one that separates the two streams). Simulations were run for steady state conditions both for a conduit with corrugated walls and for a conduit with smooth walls.

RESULTS

Initially the temperatures extracted from the thermography images are compared with those acquired by the thermocouples and are found to be in good agreement (±5% deviation). This is illustrated in ***Fig. 3***, where for a typical experiment the temperatures measured by the thermocouples (T_{TC}) are compared with the corresponding ones (T_{TG}) obtained by the *IR* thermography. In ***Fig. 4*** a typical temperature distribution over the whole area of a corrugated plate obtained by *CFD* simulation is illustrated. The figure also shows the hot/cold fluid inlet/outlet areas. It is evident that the hot fluid temperature changes gradually over the plate. As boundary condition for the *CFD* calculations the temperature distribution of the cold fluid, which was acquired experimentally from the cold fluid thermograph (***Fig. 5***), was imposed. Although the temperature profile is not symmetric across the direction of the flow (***Fig. 5a***) a linear distribution can be considered a good first approximation (***Fig. 5b***).

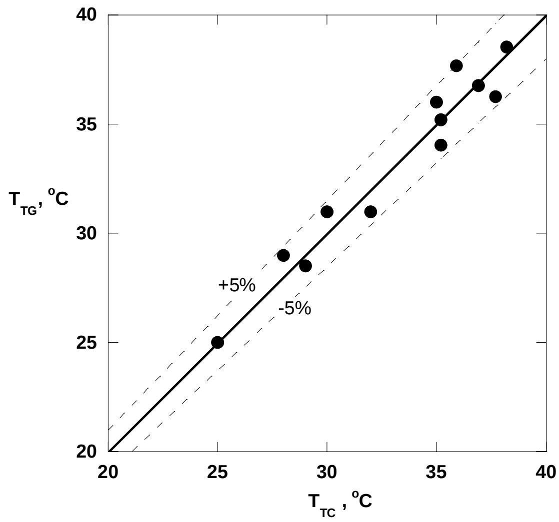


Figure 3. Typical temperature values measured by thermocouples (T_{TC}) vs. those by IR thermography (T_{TG}) (lines represent ±5% deviation).

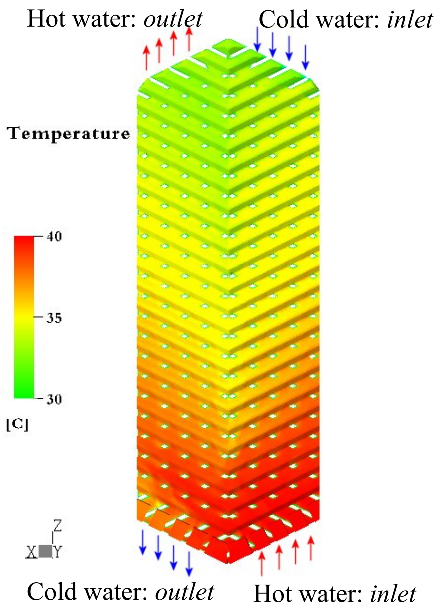


Figure 4. Typical *CFD* temperature profile of the hot side (Re_c=1000 & Re_h=1470).

In ***Fig. 6*** a typical thermography image is compared to the corresponding *CFD* simulation results that give the temperature distribution over the *IR* measuring "window". In general the thermography images are consistent with the *CFD* simulation (***Fig. 6a***) considering the inherent limitations of the measuring technique and the small differences of the color palettes used. Moreover, ***Fig. 6b***, where the *CFD* simulation results along the centre line of the plate are compared with the experimental ones, shows that the experimental data can be very well approximated by the *CFD* simulation. Consequently, the *CFD* can be regarded a reliable tool for predicting the heat exchanger performance.

The *CFD* simulation results are presented in terms of velocity, temperature, local Nusselt number and wall shear stress distributions. However, due to space limitations, only typical results are presented in this paper.

Swirling flow is identified inside the furrows of the corrugations. It is suggested [14] that this kind of secondary flow is the result of interaction between the flow inside the narrow channel and the highly accelerated flow over the crest. Focke & Knibbe [4] describe also this kind of swirling flow. Blomerius & Mitra [15] have also observed longitudinal vortices in narrow channels, while Won et al. [16] consider this secondary flow responsible for the increase in turbulence shear stress and turbulence production that result to high heat transfer rates.

The overall Nusselt number (Nu), defined as:

$$Nu = \frac{hD_h}{k} \tag{6}$$

where h the average heat transfer coefficient, D_h the hydraulic diameter and k the thermal conductivity of the fluid, was calculated from experimental data for various Reynolds numbers and found to be in very good agreement with the simulation results (5% deviation). Limited experimental data of heat transfer and pressure drop for the corrugated conduit geometry are available in the open literature. Vlasogiannis et al. [6] studied a similar heat exchanger and for Re>650 proposed the following correlation:

$$Nu = 0.51\,\mathrm{Re}^{0.58}\,\mathrm{Pr}^{0.33} \tag{7}$$

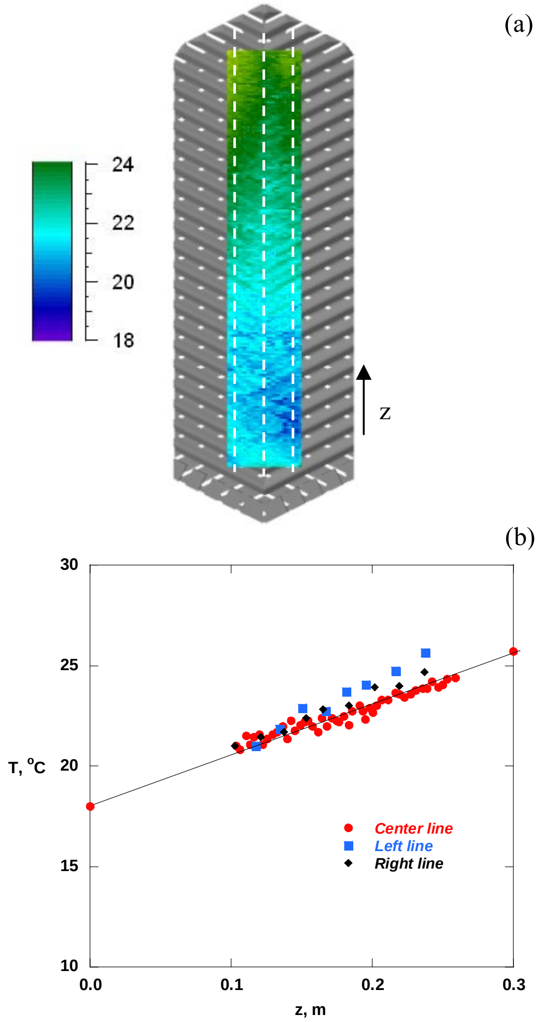


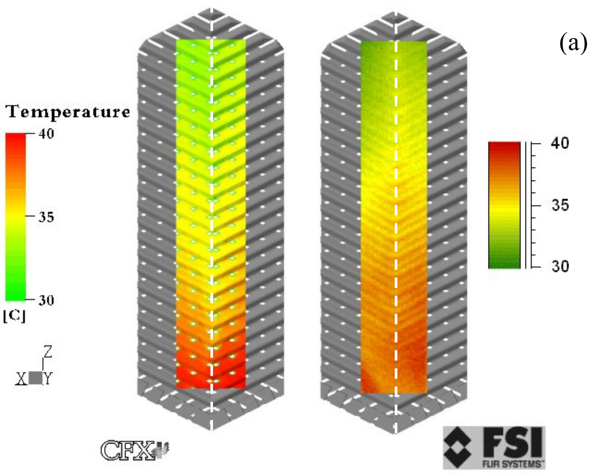
Figure 5. a) Typical temperature profile acquired using *IR* Thermography for the cold stream side and b) temperature distribution across the plate (Re_c=1000 & Re_h=1470).

In ***Fig. 7*** both experimental and *CFD* results of the present study are presented and found to be in good agreement with the above correlation (±10%). The heat transfer augmentation in terms of the ratio of the Nu for the corrugated conduit over the Nu_{sm} for the smooth-wall conduit is presented in ***Fig. 8*** as a function of Reynolds number. As expected, this ratio increases with Reynolds number.

Fig. 9 presents experimental data and *CFD* predictions of the friction factor in the corrugated conduit. Heavner et al. [17] suggested an empirical correlation for the friction factor f of the form:

$$f = m\,\mathrm{Re}^{-n} \qquad (8)$$

Both the experimental data and the simulation results of this study follow this power law with a value of n= 0.135 which is in agreement with the value proposed by Heavner et al [17], for a similar geometry. However, there is a difference (<10%) in the value of m which can be attributed to small differences in the shape of the corrugations. More specifically, the roundness of the corrugations of the experimental setup differs slightly from that of the simulation. Sparrow & Hossfeld [18] report that, even for small changes in the corrugation roundness, the value of friction factor f can be reduced up to the 60% of the f for the non-rounded case.



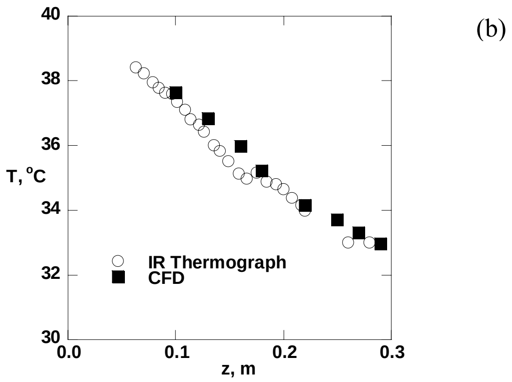


Figure 6. Comparison of temperature profiles obtained by *CFD* simulation and *IR* thermography: a) over the whole plate- "window" and b) along the centerline (Re_c=1000, Re_h=1470).

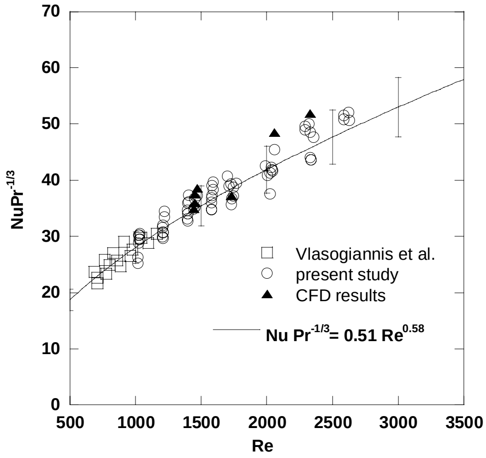


Figure 7. Dependence of $NuPr^{-1/3}$ on Re: CFD predictions and experimental data (±10% error bars).

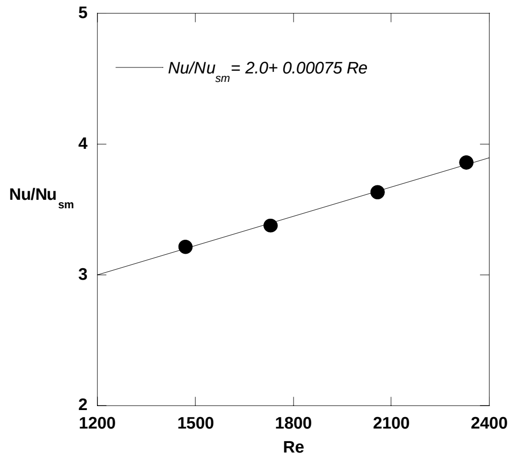


Figure 8. Heat transfer augmentation due to corrugations as a function of Re: CFD predictions.

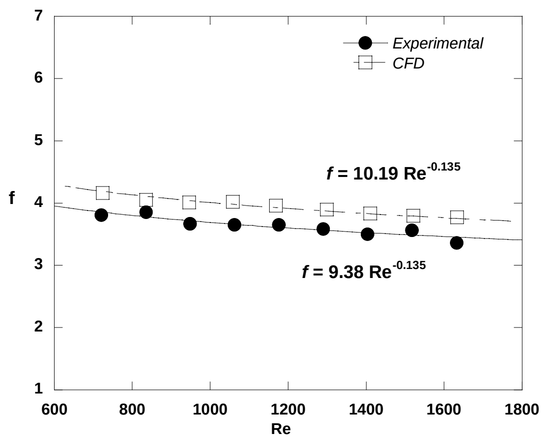


Figure 9. Dependence of friction factor f on Re: Experimental data and CFD predictions.

CONCLUDING REMARKS

The present work examines the capability of a *CFD* code to predict the flow and heat transfer characteristics in a narrow channel with corrugated walls. It is found that the *CFD* code simulation is an effective tool to predict flow characteristics and to estimate heat transfer rates as well as pressure losses in this type of process equipment. In this way various geometrical configurations can be studied and their effect on the overall performance of a heat exchange device can be estimated. Since an optimum design must involve a balance between friction losses and heat transfer rates, the *CFD* simulation helps the designer to decide how to trade off between these two factors.

In order to validate the results of the present work, additional experimental work is needed and indeed is in progress in this laboratory, concerning velocity profiles inside the conduit, pressure drop measurements and flow visualisation.

Acknowledgement: Financial support by *GSRT* (*PENED*) is greatly acknowledged. Dr A.A. Mouza would also like to thank the Chemical Engineering Dept. of AUTh for a Grant to acquire the license for the *CFD* software.

NOMENCLATURE

A_f	Mean flow cross-section per channel (m^2)
b_c	Mean spacing between plates (m)
D_h	Hydraulic diameter (m)
f	Friction factor
h	Heat transfer coefficient (W/m^2K)
k	Thermal conductivity (W/mK)
Nu	Nusselt number
Pr	Prandtl number
Re	Reynolds number
T	Temperature (K)
u_i	Flow velocity of stream i at the entrance (m/s)
V_i	Volumetric flow rate of stream i (m^3/s)
w	Plate width inside gasket (m)
y^+	Dimensionless distance from the wall

Greek letters

μ	Viscocity (kg/ms)
ρ	Density (kg/m^3)

Subscripts

c	Cold stream
h	Hot stream
sm	Smooth
TC	Thermocouple
TG	Thermography

REFERENCES

[1] Shah, R.K., Wanniarachchi, A.S., 1991 Plate heat exchanger design theory, In: *Buchlin, J.-M. Ed., Industrial Heat Exchangers*, von Karman Institute Lecture Series 1991-04.

[2] Kays, W.M. & London, A.L., 1998 Compact heat exchangers, 3rd Ed. Krieger Publ. Co., Florida.

[3] Paras, S.V., Drosos, E.I.P., Karabelas, A.J, Chopard, F., 2001 Counter-Current Gas/Liquid Flow Through Channels with Corrugated Walls–Visual Observations of Liquid Distribution and Flooding, World Conference on Experimental Heat Transfer, Fluid Mechanics & Thermodynamics, Thessaloniki.

[4] Focke, W.W., Knibbe, P.G., 1986 Flow visualization in parallel-plate ducts with corrugated walls, J. Fluid Mech., 165, 73-77.

[5] Ciofalo, M. Collins, M.W., Stasiek, J.A., 1998 Flow and heat transfer predictions in flow passages of air preheaters: assessment of alternative modeling approaches, In: Computer simulations in compact heat exchangers, Eds. B. Sunden, M.Faghri, Computational Mechanics Publ. U.K.

[6] Vlasogiannis, P., Karagiannis, G., Argyropoulos, P., Bontozoglou, V., 2002 Air–water two-phase flow and heat transfer in a plate heat exchanger, Int. J. Multiphase Flow, 28, 5, 757-772.

[7] Lioumbas, I.S., Mouza, A.A., Paras, S.V., 2002 Local velocities inside the gas phase in counter current two-phase flow in a narrow vertical channel, Chem. Eng. Research & Design, 80, 6, 667-673.

[8] Davidson, L., 2001 An Introduction to Turbulence Models, Department of Thermo and Fluid Dynamics, Chalmers University of Technology, Götenberg, Sweden.

[9] Menter, F., Esch, T., 2001 Elements of Industrial Heat Transfer Predictions, 16th Brazilian Congress of Mechanical Engineering COBEM, Uberlandia, Brazil.

[10] CFX® Release 5.7 Manual, 2004 Ansys, Inc. International.

[11] Wilcox, D., 1988 Reassessment of the scale-determining equation, AIAA Journal, 26, 11, 1299-1310.

[12] Mehrabian, M.A., Poulter, R., 2000 Hydrodynamics and thermal characteristics of corrugated channels: computational approach, Applied Mathematical Modeling, 24, 343-364.

[13] Kanaris, A.G., Mouza, A.A., Paras, S.V., 2005 Flow and Heat Transfer in Narrow Channels with Corrugated Walls: A CFD Code Application, Chem. Eng. Research & Design, 83 (A5), 460-468.

[14] Heggs, P.J., Sandham, P., Hallam, R.A., Walton, C., 1997 Local transfer coefficients in corrugated plate heat exchangers channels, Chem. Eng. Research & Design, 75 (A7), 641-645.

[15] Blomerius, H., Mitra, N.K., 2000 Numerical investigation of convective heat transfer and pressure drop in wavy ducts, Numerical Heat Transfer, Part A, 37, 37-54.

[16] Won, S.Y., Mahmood, G.I., Ligrani, P.M., 2003 Flow structure and local Nusselt number variations in a channel with angled crossed-rib turbulators, Int. J. Heat Mass Transfer, 46, 3153-3166.

[17] Heavner, R.L., Kumar, H., Wanniarachchi, A.S., 1993 Performance of an industrial plate heat exchanger: effect of chevron angle, AIChe Symposium Ser. No 295, Vol. 89, Heat Transfer, Am. Inst. Chem. Eng. Atlanta, GA, 262-267.

[18] Sparrow, E.M., Hossfeld, L.M., 1984 Effect of rounding of protruding edges on heat transfer and pressure drop in a duct, Int. J. Heat Mass Transfer, 27, 10, 1715-1723.