CFD CODE APPLICATION
TO FLOW THROUGH NARROW CHANNELS
WITH CORRUGATED WALLS

S.V. Paras*, A.G. Kanaris, A.A. Mouza, A.J. Karabelas

Department of Chemical Engineering
and Chemical Process Engineering Research Institute
Aristotle University of Thessaloniki
Univ. Box 455, GR 54124 Thessaloniki, GREECE
* *tel*: +30310996174 *e-mail*:paras@eng.auth.gr

Abstract

Novel compact condensers and in general two-phase heat exchangers made of corrugated plates hold significant advantages over conventional equipment. In an effort to optimise design and operation of this type of equipment, a commercial CFD code (CFX®) is employed to simulate the flow through an element of a model compact heat exchanger and to provide information on the local flow structure. For simplicity, the channel used for the simulation is formed by only **one** corrugated plate, which is comprised of twelve equal sized and uniformly spaced corrugations and two ***side-grooves***, while the other plate is flat. The Reynolds numbers examined are 290, 850, 1150 and 1450. A standard k-ε model was used for the calculations and, in addition to isothermal flow, heat transfer simulations are carried out for the case of hot air (60^oC) in contact with a constant-temperature wall (20^oC).

Results are presented in terms of velocity, temperature, wall shear stress, wall heat flux and local Nusselt number profiles. The results confirm the dominant role of the vertical side-grooves in flow distribution. These calculated mean heat transfer coefficients are found to be in reasonable agreement with the limited published experimental data.

Introduction

Novel compact condensers and, in general, two-phase heat exchangers made of corrugated plates hold significant advantages over conventional equipment. Such exchangers are being rapidly adapted by food and chemical process industries, replacing shell-and-tube exchangers. Plate exchangers offer high thermal effectiveness and close temperature approach, while allowing ease of inspection and cleaning (Kays & London, 1984; Shah & Wanniarachchi, 1991). In order to be able to quantitatively evaluate the performance of a corrugated-plate compact heat exchanger, methods to predict the heat transfer coefficient and pressure drop must be developed. In this direction, a CFD code simulation would be an effective tool to estimate momentum and heat transfer rates in this type of process equipment.

Ciofalo et al. (1998) in a comprehensive review article concerning modelling heat transfer in narrow flow passages stated that, in the Reynolds number range of 1500-3000, *transitional* flow is expected, a kind of flow among the most difficult to simulate by conventional turbulence models. More precisely, the "low-Reynolds number" k-ε model is not considered capable of predicting the flow parameters in the complex geometry of a corrugated narrow channel, whereas the standard k-ε model using "wall functions" overpredicts both wall shear stress and wall heat flux, especially for the lower range of the Reynolds number encountered in this kind of equipment. This might be related to the grid characteristics near the wall. Due to the modular nature of a compact heat exchangers, a common practice is to think of it as composed of a large number of unit *cells* and obtain results by using a single cell as the computational domain and imposing periodicity conditions across its boundaries (e.g. Ciofalo et al., 1998; Mehrabian & Poulter, 2000). However, the validity of this assumption is considered an open issue in the literature (Ciofalo et al., 1998).

Another open issue is the type of flow prevailing in such narrow passages. Contrary to Ciofalo et al. (1998), Shah & Wanniarachchi (1991) stated that, for the Reynolds number range 100-1500, there is evidence that the flow is turbulent. Recently, Vlasogiannis et al. (2002), who experimentally tested a plate heat exchanger under single and two-phase flow conditions, verify that the flow is turbulent for $Re>650$. Lioumbas et al. (2002), who studied experimentally the flow in narrow passages, during counter-current gas-liquid flow, suggest that the flow exhibits the basic features of turbulent flow even for the relatively low gas Reynolds numbers tested ($500< Re <1200$).

Focke & Knibbe (1986) performed flow visualization in narrow passages with corrugated walls using an electrode-activated pH reaction. They concluded that flow patterns in such geometries are very complex and suggested that the local flow structure controls the heat transfer process in the narrow passages. The salient feature of the flow is the existence of secondary swirling motions along the furrows of their test section.

In a previous study in this Laboratory (Paras et al., 2001), visual observations were made of counter-current gas-liquid flow, in a special Plexiglas test section, simulating a vertical channel of a corrugated plate heat exchanger. On the two plates manufactured by *VICARB-Alfalaval*, corrugations were machined at a 45^o angle, as well as ***side grooves***, i.e. vertical side channels (***Figure 1***). The two plates were superposed so that the opposite corrugations formed a cross-type pattern with the crests of the corrugations nearly in contact. The experiments revealed that the two side-channels of the corrugated plate (***Figure 1***) play a significant role in the liquid flow through the furrows, promoting even distribution. The lateral drainage into the side channels tends to increase with increasing gas flow rate, leading to a progressive elimination of the liquid film. This situation, referred as “maldistribution”, may be favorable for the operation of such a device as a condenser because of the exposure of nearly ‘fresh’ wall to the condensing vapors.

In this paper, an attempt is made to simulate the flow field within the complicated passages of an *entire* element of a plate heat exchanger and not only a single cell. The aim is to obtain information on the flow pattern prevailing inside the furrows and the side channels of the conduit, which affects the local momentum and heat transfer rates of this type of equipment.

Model and solution procedure

The geometry studied in the present simulations is consistent with an existing compact heat exchanger described in detail elsewhere (Paras et al., 2001). However, to keep the computational demands at acceptable levels, a simpler channel is studied. This channel is formed by only **one** of the corrugated plates (***Figure 2***), which is comprised of twelve equal sized and uniformly spaced corrugations and two side-grooves, while the second plate is flat. Details of the plate geometry are presented in ***Table 1***. The simpler case of single-phase flow of air is investigated here. The Reynolds numbers examined are 290, 850, 1150 and 1450, based on the distance between the plates at the entrance (d=10mm), the mean flow velocity and the fluid properties at 60^o C. In addition to isothermal flow, heat transfer simulations are carried out for the same Reynolds numbers, where hot air (60^oC) is cooled in contact with a constant-temperature wall (20^oC). The latter case is realized in condensers and evaporators. Additionally, it is assumed that heat is transferred only through the corrugated plate, while the rest of the walls are considered adiabatic.

Table 1. Plate geometric characteristics.

Plate length	0.200 m
Plate width	0.110 m
Maximum spacing between plates	0.010 m
Number of corrugations	12
Corrugation angle	45^o
Corrugation pitch	0.008 m
Corrugation width	0.014 m
Corrugation top width	0.002 m
Plate length before and after corrugations	0.050 m
Groove width	0.005 m
Heat transfer area	2.7×10^{-2} m^2

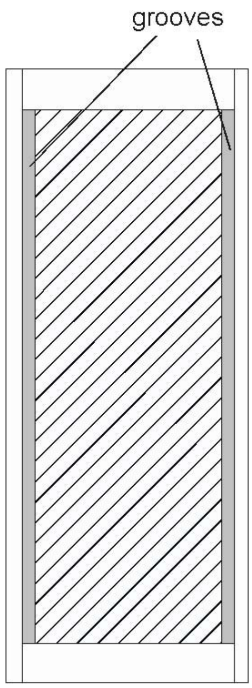


Figure 1. Schematic of the corrugated plate

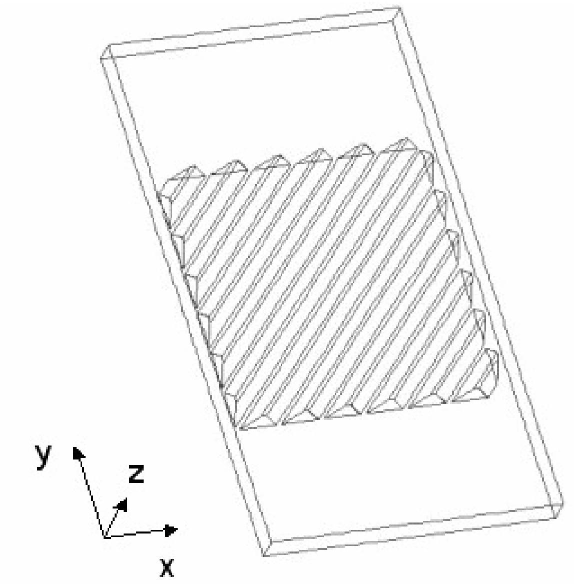


Figure 2. CFD model channel

A commercial CFD code, namely the CFX® 4.4 code developed by AEA Technology, was employed to explore its potential for computing detailed characteristics of this kind of flow. In general, the models employed in CFD codes give reasonably good results for single-phase flow systems. The first step in obtaining a solution is the division of the physical domain into a solution mesh, in which the set of equations is discretised. The grid size used is chosen by performing a grid dependence study, since the accuracy of the solution depends on the number and the size of the cells (Versteeg & Malalasekera, 1995). The standard k-ε model with "wall-functions" was used in the calculations for reasons explained in the Introduction. The mean velocity of the gas phase was applied as boundary condition at the channel entrance (i.e. Dirichlet BC on the inlet velocity) and no-slip conditions on the channel walls. The flow is considered steady and weakly compressible (i.e. the gas density is only temperature dependant) for the heat transfer problem. A constant temperature boundary condition was applied only on the corrugated wall, whereas the rest of the walls are considered adiabatic.

Calculations were performed on a *SGI* O_2 R10000 workstation with a 195MHz processor and 448Mb RAM. The CFX®4.4 code uses a finite volume method on a non-orthogonal body-fitted multi-block grid. In the present calculations, the *SIMPLEC* algorithm is used for pressure-velocity coupling and the *QUICK* scheme for discretisation of the momentum equations (Versteeg & Malalasekera, 1995; CFX User Manual, 2001).

Results

The results of the present study confirm the dominant role of the vertical side-grooves in flow distribution and suggest that gas flow is mainly directed to the left side channel of this model plate (***Figure 3a***). Part of the flow ("reflected" initially on the left side wall) follows the furrows and reaches the opposite side channel. It appears that if two corrugated plates with angles $+45^0$ and -45^0 were superposed (as in real condensers) part of the gas phase would also be directed to the right channel, creating a symmetrical overall flow distribution. Experiments performed in this Laboratory (Paras et al., 2001) suggest that the above flow pattern promotes the drainage of the liquid phase through the side grooves in counter-current two-phase flow.

This type of flow behavior is also described by Focke & Knibbe (1985), who made visual observations of the flow between two superposed corrugated plates without side channels (grooves). They confirm that the fluid, after entering a furrow, mostly follows it until it reaches the sidewall, where it is reflected and enters the anti-symmetrical furrow of the plate above (***Figure 3b***), a behavior similar to the one predicted by the CFD simulation. More specifically, the velocity inside the left-side channel progressively decreases (***Figure 4a***), while that in the right-side channel increases (***Figure 4b***). It seems that most of the flow passes through the furrows, where enhanced heat transfer characteristics are expected, but this remains to be verified by more computations.

Figure 5 shows details of the flow inside a furrow where swirling flow is identified. This secondary flow is capable of bringing new fluid from the main stream close to the walls, augmenting heat transfer rates. Focke & Knibbe (1986), who conducted visualization experiments in similar geometries, describe also this kind of swirling flow.

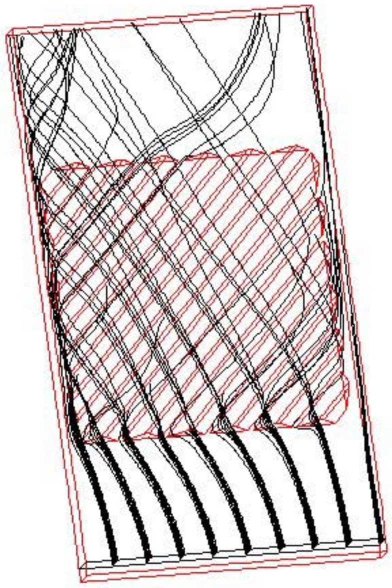


Figure 3a. Typical flow pattern inside the channel predicted by CFX, (Re=1450).

Figure 3b. Visualization of flow between two superimposed corrugated plates (Re=125) (Focke & Knibbe, 1986)

(a) **(b)**

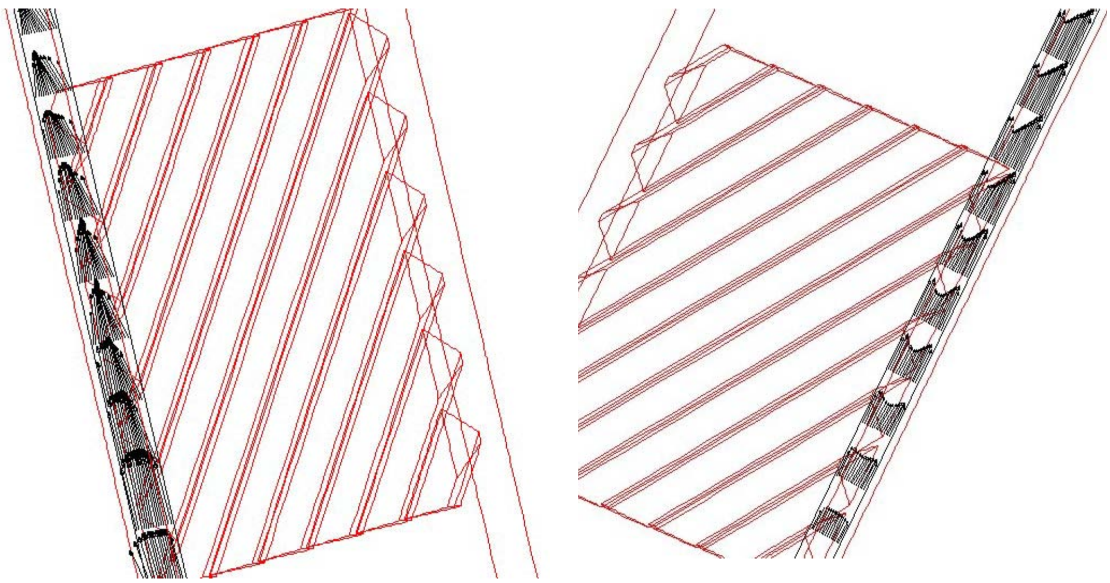


Figure 4.Velocity vectors inside grooves: (a) left-side groove; (b) right-side groove

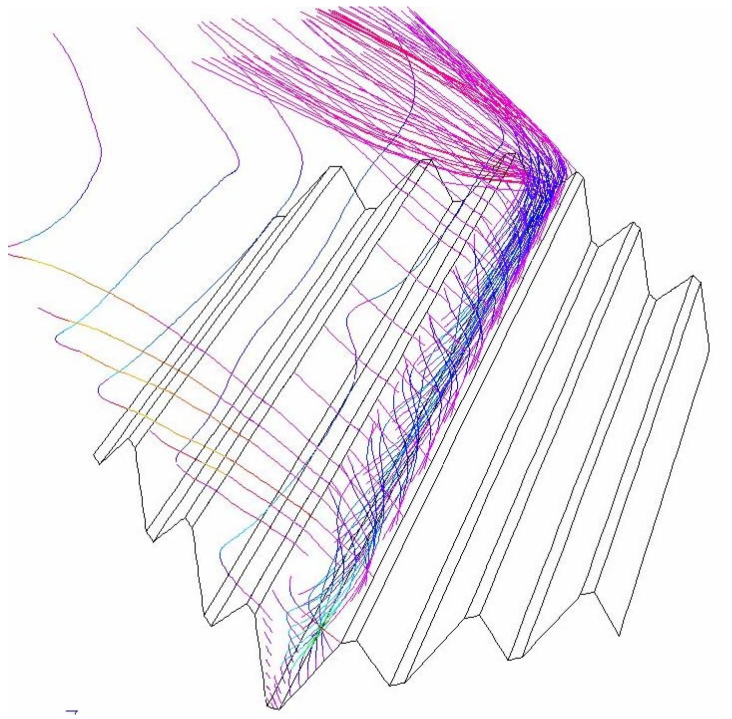
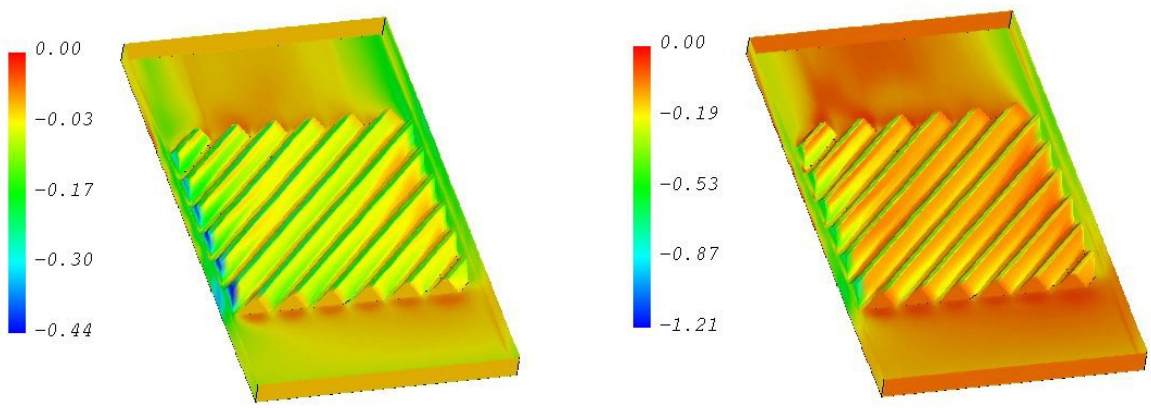


Figure 5. Swirling flow inside a furrow; Re=850.

The values of the y-component of shear stress (***Figure 6***) increase with the Reynolds number –as expected- and are higher at the crests of the corrugations than in the valleys of the furrows. It may be argued that, during gas-liquid counter-current flow in such geometries, this shear stress distribution tends to prevent the liquid layer from falling over the crest of the corrugations and to keep it inside the furrows. The visual observations of Paras et al. (2001) seem to confirm the above behaviour.



(a) **(b)**

Figure 6. Wall y-shear stress distribution on the corrugated plate: (a) Re=850; (b) Re=1450 [*in SI units*]

Wall heat flux through the corrugated plate was calculated by the CFD code (***Figure 7***). In addition, the local Nusselt number was calculated (by a user-Fortran subroutine) by using the expression:

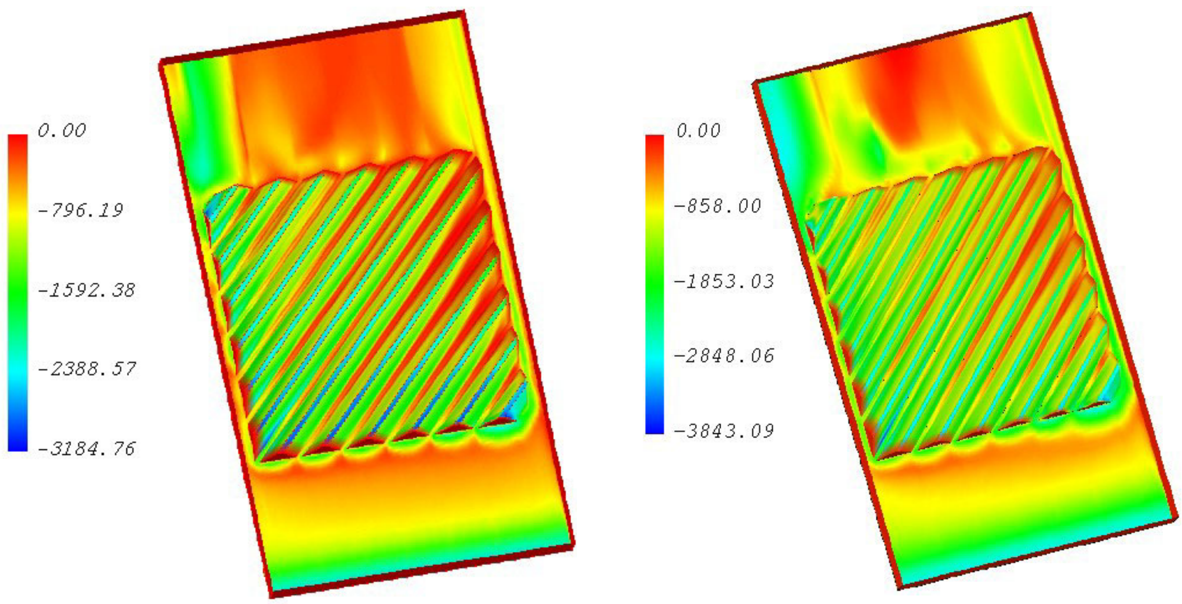
$$Nu_x = \frac{\dot{q}d}{(T_b - T_w)k}$$

where $\dot{q}$ is the local wall heat flux, d the distance between the plates at entrance, T_w the wall temperature, T_b the local fluid temperature and k the thermal conductivity of the fluid.

In addition to the local Nusselt number, mean Nusselt numbers were calculated as follows:

- a mean *Nu* calculated by numerical integration of the local *Nu* over the *corrugated* area only, and
- an overall average *Nu* using the total wall heat flux through the *whole* plate and the fluid temperatures at the channel entrance/exit.

These results are presented in ***Table 2***.

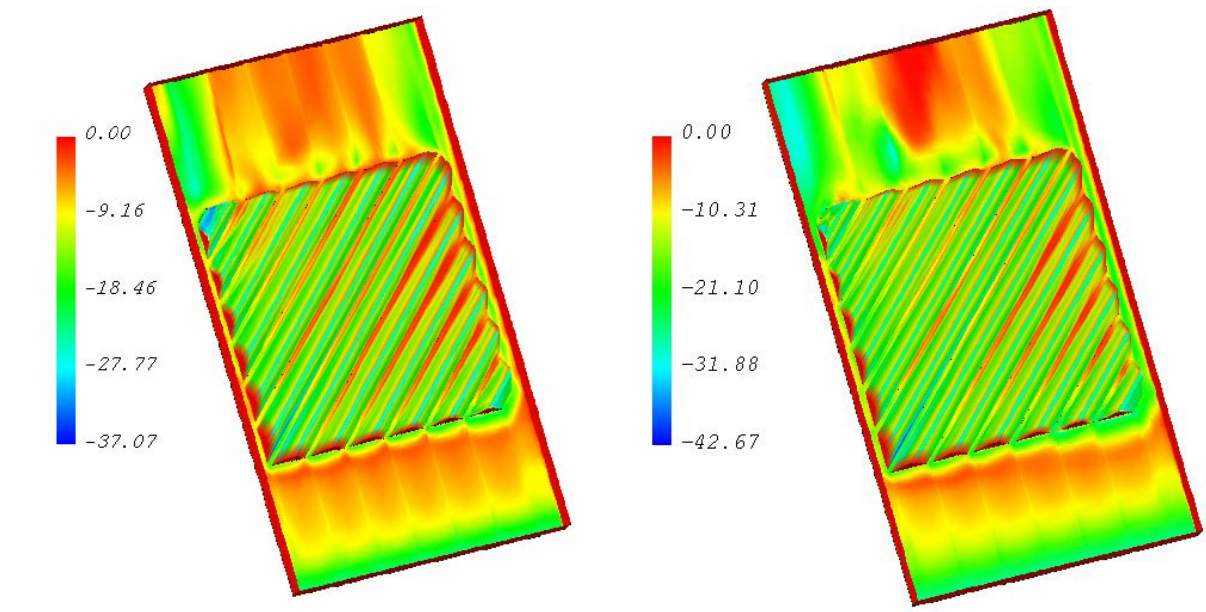


(a) **(b)**

Figure 7. Wall heat flux distribution on the corrugated plate: (a) Re=850; (b) Re=1450 [*in SI units*]

Figure 8 shows typical local Nusselt number distribution over the corrugated wall for two Reynolds numbers (Re=850 and 1450). It is noticeable that on the top of the corrugations local Nusselt numbers attain their maximum value. This confirms the strong effect of the corrugations, not only on the flow distribution, but also on the heat transfer results.

To the best of author's knowledge, laboratory measurements of heat transfer and pressure drop are very limited in the open literature for the corrugated plate geometry. However, these seem to be proprietary data (Ciofalo et al., 1998). In order to validate the simulation results, the data of Vlasogiannis et al. (2002) were used. Their data concern measurements of the heat transfer coefficients both for single (Re<1200) and two-phase flow in a plate heat exchanger with corrugated walls and a corrugation inclination angle of 60^o. Martin (1996) proposed a theoretical approach to predict heat transfer coefficients of chevron-type plate heat exchangers, with support from experimental data. It should be noted that heat exchangers used in Vlasogiannis' and Martin's experimental configurations lack the side grooves of the model plates employed in the present simulation.



(a) **(b)**

Figure 8. Local *Nu* distribution on the corrugatred plate: (a) Re=850; (b) Re=1450 ; *negative Nusselt numbers are due to heat flux sign.*

Table 2 shows that data by Vlasogiannis et al. are in good agreement with those derived from Martin's model. It should be also noted that Focke et al. (1985) measured heat transfer coefficients in a corrugated plate heat exchanger; by placing a partition of celluloid sheet be-tween the two plates, they report that the overall heat transfer rate is reduced to 65% of the value for the plates without the partition. ***Figure 9*** shows that the mean Nusselt number values calculated by the CFD code for the corrugations (i.e. excluding the smooth part of the wall) are practically equal to the 65% the values measured by Vlasogiannis et al. This holds true for all

Reynolds numbers except the smallest one (Re=290). In the latter case the Nusselt number is greatly overpredicted by the CFD code. This is not unexpected, since the *k-ε* model is not capable of correctly predicting the heat transfer characteristics for such a low Reynolds number (Ciofalo et al., 1998).

The comparison of the Nusselt number values of the corrugated wall to the overall *Nu* (***Table 2***) reveals that the presence of the smooth part of the plate does not significantly influence the heat transfer coefficient. Consequently the existence of the side-channels, whose area is a small percentage of the total plate area, although it inhibits flooding, has practically no effect on the thermal behavior of the plate in single phase flow.

Table 2. Mean of heat transfer rates measured and computed.

Re	**Nu**				
	CFD		*data for comparison*		
	corrugations	*overall*	[1]	[2]	[3]
290	13.7	11.2	5.05	-	-
850	17.3	16.7	22.8	23.1	14.8
1150	18.6	17.6	26.9	27.0	17.5
1450	20.6	20.1	30.6	33.2	19.9

[1] data by Vlasogiannis et al (2002)
[2] predictions by Martin (1996)
[3] estimation for single plate; 65% of [1]

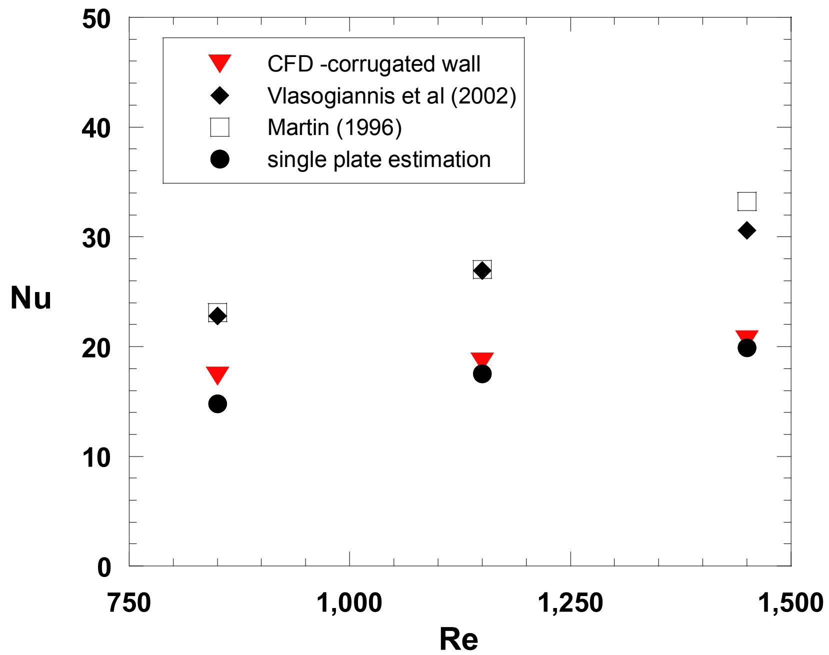


Figure 9. Comparison of predictions with experimental data

Concluding remarks

Compact heat exchangers (*CHX*) with corrugated plates offer a great number of advantages over conventional heat exchangers. Unfortunately, unlike the conventional shell-and-tube heat exchangers, for plate heat exchangers there is a lack of a generalized thermal and hydraulic design method. Variations in design of the geometry of a *CHX*, concerning the basic features and parameters like the aspect ratio of the corrugations or their angle, make it almost impossible to generate an adequate heat transfer database covering all possible configurations. The use of a CFD code is promising in this respect, as it allows computation for various geometries, in order to evaluate their effects and to study them closely. Of course, experimental work is still necessary to help the researchers validate their results.

The simulation of the present work shows that corrugations improve both flow distribution and heat transfer. Calculated Nusselt numbers for the simplified model employed here are practically equal to those of commercial heat exchangers, although the former includes side-channels. The latter tend to improve the operability of the heat exchanger by shifting the flooding limit to higher gas velocities. Nevertheless, further computations are required to examine the effects of these side-channels on the heat transfer coefficients, and optimize their design.

References

1. *CFX Release 4.4 User Guide,* 2001, AEA Technology, CFX International, Harwell, Didcot, UK.
2. 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.
3. Focke, W.W., Knibbe, P.G. 1986, Flow visualization in paral1el-plate ducts with corrugated walls. *J. Fluid Mech*. **165**, 73-77.
4. Focke, W.W., Zachariades, J., Olivier, I. 1985, The effect of the corrrugation inclination angle on the thermohydraulic performance of plate heat exchangers. *Int. J. Heat Mass Transf*. **28**, 1469-1497.
5. Kays, W.M. & London, A.L. 1998, Compact heat exchangers 3rd Ed. Krieger Publ. Co., Florida.
6. 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, *Chemical Engineering Research & Design* (accepted for publication).
7. Martin, H. 1996, A theoretical approach to predict the performance of chevron-type plate heat exchangers. *Chemical Engineering and Processing* **35**, 301-310.
8. Mehrabian, M.A., Poulter, R. 2000, Hydrodynamics and thermal characteristics of corrugated channels: computational approach, *Applied Mathematical Modeling* **24,** 343-364.
9. 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, September 24-28.
10. 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.
11. Versteeg, H.K., Malalasekera, W. 1995, An introduction to Computational Fluid Dynamics, Longman London.
12. 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