1st International Conference "From Scientific Computing to Computational Engineering"
1st IC-SCCE
Athens, 8-10 September, 2004

DESIGNING NOVEL COMPACT HEAT EXCHANGERS FOR IMPROVED EFFICIENCY USING A CFD CODE

Athanasios G. Kanaris, Katerina A. Mouza, Spiros V. Paras*

Laboratory of Chemical Process and Plant Design
Department of Chemical Engineering
Aristotle University of Thessaloniki
Univ. Box 455, GR 54124 Thessaloniki, GREECE
* *tel*: +302310996174, *e-mail*:paras@cheng.auth.gr, *www*: http://philon.cheng.auth.gr

Keywords: compact heat exchanger, narrow channel, corrugation, CFD, Nusselt number, pressure drop

Abstract. *In an effort to study and optimize the design of a plate heat exchanger comprising of corrugated walls with herringbone design, a CFD code is employed. Due to the difficulties induced by the geometry and flow complexity, an approach through a simplified model was followed as a first step. This simple model, comprised of only one corrugated plate and a flat plate, was constructed and simulated. The Reynolds numbers examined are 400, 900, 1000, 1150, 1250 and 1400. The SST turbulence model was preferred over other flow models for the simulation. The case where hot water (60°C) is in contact with a constant-temperature wall (20°C) was also simulated and the heat transfer rate was calculated. The results for the simplified model, presented in terms of velocity, shear stress and heat transfer coefficients, strongly encourage the simulation of one channel of the typical plate heat exchanger, i.e. the one that comprises of two corrugated plates with herringbone design having their crests nearly in contact. Preliminary results of this latter work, currently in progress, comply with visual observations.*

1 INTRODUCTION

In recent years, compact heat exchangers with corrugated plates are being rapidly adopted by food and chemical process industries, replacing conventional shell-and-tube exchangers. Compact heat exchangers consist of plates embossed with some form of corrugated surface pattern, usually the chevron (herringbone) geometry[1]. The plates are assembled being abutting, with their corrugations forming narrow passages. This type of equipment offers high thermal effectiveness and close temperature approach, while allowing ease of inspection and cleaning [1],[2]. In order to be able to evaluate its performance, methods to predict the heat transfer coefficient and pressure drop must be developed. In this direction, CFD is considered an efficient tool for momentum and heat transfer rate estimation in this type of heat exchangers.

The type of flow in such narrow passages, which is associated with the choice of the most appropriate flow model for CFD simulation, is still an open issue in the literature. Due to the relatively high pressure drop, compared to shell-and-tube heat exchangers for equivalent flow rates, the Reynolds numbers used in this type of equipment must be lower 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]. Ciofalo et al.[4], in a comprehensive review article concerning modelling heat transfer in narrow flow passages, state that, for the Reynolds number range of 1,500-3,000, transitional flow is expected, a kind of flow among the most difficult to simulate by conventional turbulence models. On the other hand, Shah & Wanniarachchi[1] declare that, for the Reynolds number range 100-1500, there is evidence that the flow is already turbulent, a statement that is also supported by Vlasogiannis et al.[5], whose experiments in a plate heat exchanger verify that the flow is turbulent for Re>650. Lioumbas et al.[6], 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[7] performed flow visualization experiments in narrow passages with corrugated walls. They concluded that the flow patterns in such geometries 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 in such narrow passages.

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 is a characteristic phenomenon occurring on the corrugated surfaces in these geometries. Moreover, the standard k-ε

model requires a course grid near the wall, based on the value of y^+=11 [9],[10], which is difficult to accomplish in confined geometries. The low-Re *k-ε* model, which uses "dumping functions" near the wall[8],[9], is not considered capable of predicting the flow parameters in the complex geometry of a corrugated narrow channel[4], requires finer mesh near the wall, is computationally expensive compared to the standard *k-ε* model and it is unstable in convergence.

An alternative to *k-ε* model, is the *k-ω* model, developed by Wilcox[11]. This model, which uses the turbulence frequency *ω* instead of the turbulence diffusivity *ε*, appears to be more robust, even for complex applications, and does not require very fine grid near the wall[8]. However, it seems to be sensitive to the free stream values of turbulence frequency *ω* outside the boundary layer. A combination of the two models, *k-ε* and *k-ω*, 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 thus it benefits from the advantages of both models.

Some efforts have been made towards the effective simulation of a plate heat exchanger. 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 unit *cells* (Representative Element Units, *RES*) and obtain results by using a single cell as the computational domain and imposing periodicity conditions across its boundaries[4],[12]. However, the validity of this assumption is considered another open issue in the literature[4].

2 STUDY OF A SIMPLIFIED GEOMETRY

In an effort to simulate the flow configuration, a ***simple*** channel was designed and constructed in order to conduct experiments and obtain information on the flow pattern prevailing inside the furrows of the conduit. The flow configuration, apart from affecting the local momentum and heat transfer rates of a plate heat exchanger, suggests the appropriate flow model for the CFD simulation. A module of a plate heat exchanger is a single pass of the exchanger, consisting of only two plates. The simple channel examined is a single pass made of Plexiglas (***Figure 1***). It is formed by only ***one*** corrugated plate comprised of fourteen equal sized and uniformly spaced corrugations as well as a flat plate and it is used for pressure drop measurements and flow visualization. Details of the plate geometry are presented in ***Table 1***. This model was chosen in an attempt to simplify the complexity of the original plate heat exchanger and to reduce the computational demands.

The geometry studied in the CFD simulations (similar to the test section) is shown in ***Figure 2***. The Reynolds numbers examined are 400, 900, 1000, 1150, 1250 and 1400, which are based on the distance between the plates at the entrance (*d*=10mm), the mean flow velocity and the properties of water at 60°C. In addition to isothermal flow, heat transfer simulations are carried out for the same Reynolds numbers, where hot water (60°C) is cooled in contact with a constant-temperature wall (20°C). 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.

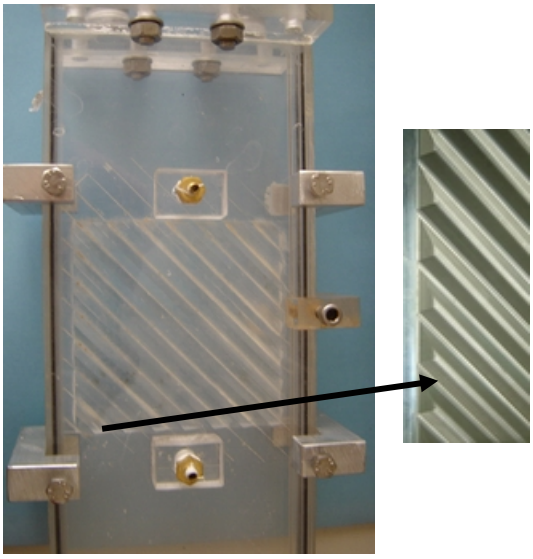


Figure 1. Simplified model and detail of the corrugated plate

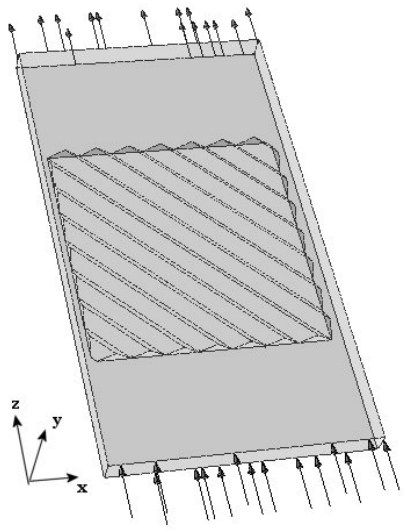


Figure 2. CFD model

A commercial CFD code, namely the *CFX® 5.6* code developed by *AEA Technology*, was employed to explore its potential for computing detailed characteristics of this kind of flow. In general, the models used 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 selected by performing a grid dependence study, since the accuracy of the solution greatly depends on the number and the size of the cells[13]. The resulting mesh was also inspected for inappropriate generated cells (e.g. tetrahedral cells with sharp angles) and fixed, leading to a total number of 870,000 elements. The *SST* model was employed in the calculations for the reasons explained in the previous chapter. The mean velocity of the liquid 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. 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®5.6* 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[10],[13].

Plate length	0.200 m
Plate width	0.110 m
Maximum spacing between plates	0.010 m
Number of corrugations	14
Corrugation angle	45°
Corrugation pitch	0.005 m
Corrugation width	0.014 m
Plate length before and after corrugations	0.050 m
Heat transfer area	2.7×10^{-2} m^2

Table 1. Simple channel's plate geometric characteristics.

The results of the present study suggest that fluid flow is mainly directed inside the furrows and follows them (***Figure 3a***). This type of flow behavior is also described by Focke & Knibbe[7], who made visual observations of the flow between two superposed corrugated plates (***Figure 3b***). They confirm that the fluid, after entering a furrow, mostly follows it until it reaches the side wall, where it is reflected and enters the anti-symmetrical furrow of the plate above, a behavior similar to the one predicted by the CFD simulation. It seems that, in both cases, most of the flow passes through the furrows, where enhanced heat transfer characteristics are expected.

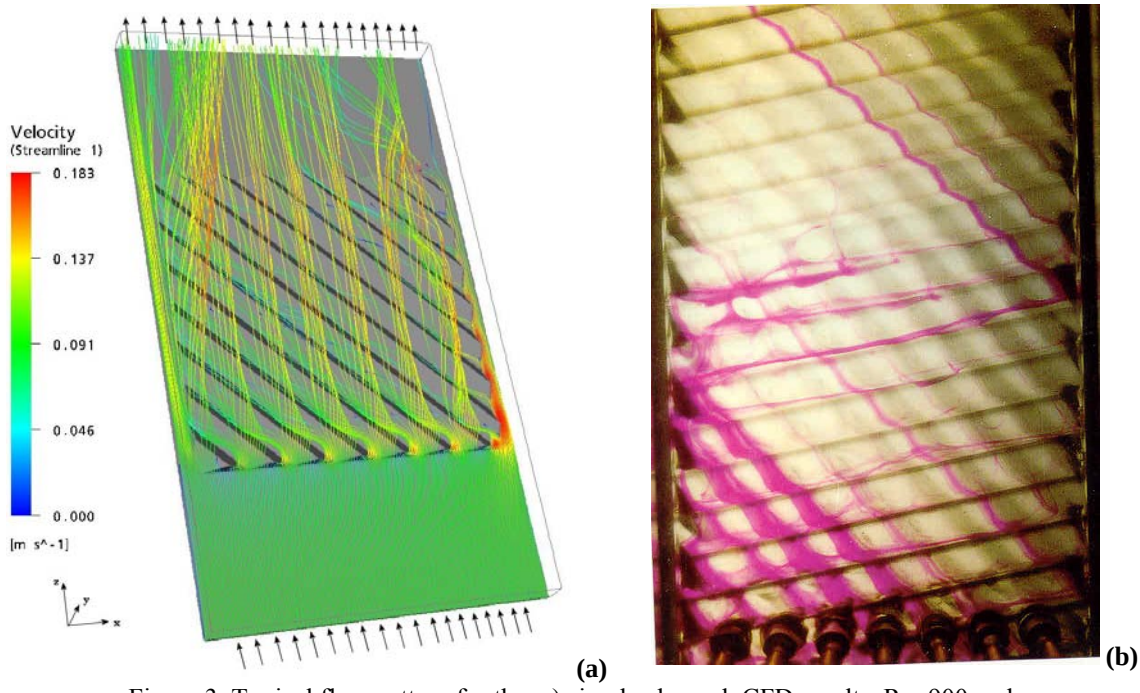


Figure 3. Typical flow pattern for the: a) simple channel, CFD results, Re=900 and b) Flow visualization by Focke & Knibbe[7], Re=125

Figure 4 shows details of the flow inside a furrow for the simple model, 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[7], who performed visualization experiments in similar geometries, also describe this kind of swirling flow.

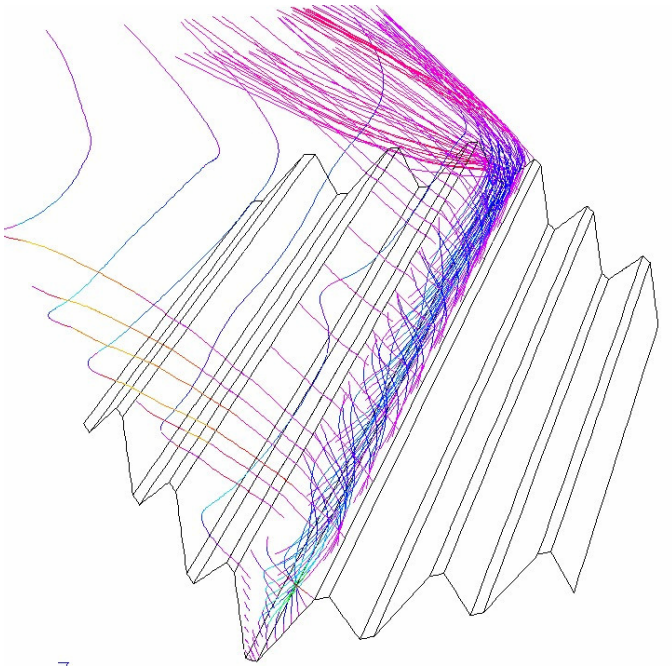


Figure 4. Swirling flow inside a furrow; Re=900

The values of the *z*-component of shear stress (***Figure 5a***) increase with the Reynolds number –as expected– and the maximum value occurs at the crests of the corrugations. It may be argued that, during gas-liquid counter-current flow in such geometries, the 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.[3] seem to confirm the above behaviour.

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

$$Nu_x = \frac{\dot{q}d}{(T_b - T_w)k} \tag{1}$$

where $\dot{q}$ is the local wall heat flux, d the distance between the plates at the 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* calculated using the total wall heat flux through the *whole* plate and the fluid temperatures at the channel entrance/exit.

The comparison of the values of the above Nusselt numbers shows that they do not differ more that 1%; therefore, the smooth part of the corrugated plate does not seem to influence the overall heat transfer.

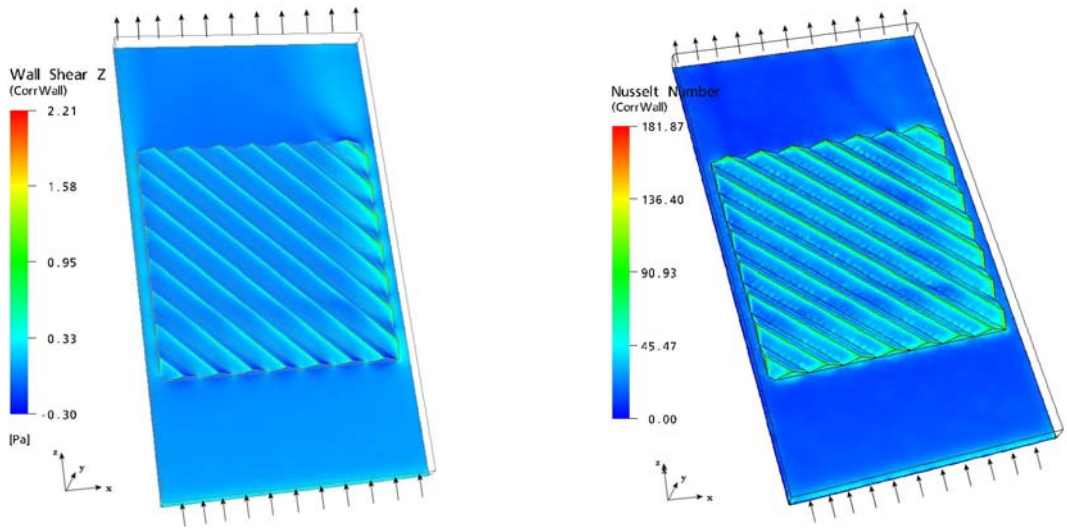
Figure 5b shows a typical local Nusselt number distribution over the corrugated wall for Re=900. All the Reynolds numbers studied exhibit similar distributions. It is noticeable that local Nusselt numbers attain their maximum value at the top of the corrugations. This confirms the strong effect of the corrugations, not only on the flow distribution, but also on the heat transfer rate.

To the best of author's knowledge, experimental values of heat transfer and pressure drop are very limited in the open literature for the corrugated plate geometry, since these data are proprietary[4]. Therefore, the data of Vlasogiannis et al.[5] were used to validate the simulation results. These data concern heat transfer coefficients measurements of both single (Re<1200) and two-phase flow in a plate heat exchanger with corrugated walls and a corrugation inclination angle of 60°. Heavner et al.[14] proposed a theoretical approach, supported by experimental data, to predict heat transfer coefficients of chevron-type plate heat exchangers.

Figure 6 presents the experimental friction factors, obtained from the Plexiglas test section of ***Figure 1***, as well as the CFD predictions for the simple geometry studied, as a function of the Reynolds number. It appears that the experimental values follow a power law of the form:

$$f = m\,\mathrm{Re}^{-n} \tag{2}$$

where m and n constants with values 0.27 and 0.14 respectively. Heavner et al.[14] proposed a similar empirical correlation based on their experimental results on a single pass of a plate heat exchanger with 45° corrugation angle, but with two corrugated plates. In spite of the differences in geometry, it appears that the present results are in good agreement with the experimental data of Heavner et al.[14] (0.687 and 0.141 for the variables m and n, respectively).



(a) **(b)**

Figure 5. Typical results of the CFD simulation for Re=900; distributions of:
(a) z-shear stress component and (b) local Nusselt number

It must be noted that Focke et al.[15] , who also measured heat transfer coefficients in a corrugated plate heat exchanger having a partition of celluloid sheet between the two plates, reported that the overall heat transfer rate is the 65% of the corresponding value without the partition. ***Figure 7*** shows that the mean j-Colburn factor values calculated using the *overall* Nusselt number are practically equal to the 65% of the values measured by Vlasogiannis et al. This holds true for all Reynolds numbers except the smallest one (Re=400). In the latter case the Nusselt number is greatly overpredicted by the CFD code. This is not unexpected, since the *two-equation turbulence* model is not capable to predict correctly the heat transfer characteristics for such low Reynolds number. The CFD results reveal that the corrugations enhance the heat transfer coefficient, whereas the pressure losses due to the augmentation of friction factor f are increased ***(Table 2)***, compared to a smooth-wall plate heat exchanger. Additionally, comparison of the normalized values of Nusselt number and the friction factor, with respect to the corresponding values for the smooth plate (f_{sm}, Nu_{sm}), indicates that as the Reynolds number increases, heat transfer enhancement is slightly reduced, while the friction factor ratio, f/f_{sm} , is increased. This is typical for plate heat exchangers with corrugations[16].

Re	Nu_{vlasog}	65% Nu_{vlasog}	Nu_{all}	Nu_{sm}	$\frac{Nu_{ave}}{Nu_{sm}}$	$\frac{f}{f_{sm}}$
400	*13.2*	*8.6*	*20.5*	-	-	-
900	*38.0*	*24.7*	*27.3*	*9.4*	*2.9*	*12.4*
1000	*41.2*	*26.8*	*28.6*	*10.2*	*2.8*	*12.8*
1150	*44.2*	*28.7*	*28.8*	*11.0*	*2.7*	*13.5*
1250	*46.8*	*30.4*	*30.9*	*11.7*	*2.7*	*13.9*
1400	*49.5*	*32.2*	*32.0*	*12.5*	*2.6*	*14.5*

Table 2. Experimental values, calculated Nusselt numbers and normalized values of Nu and f

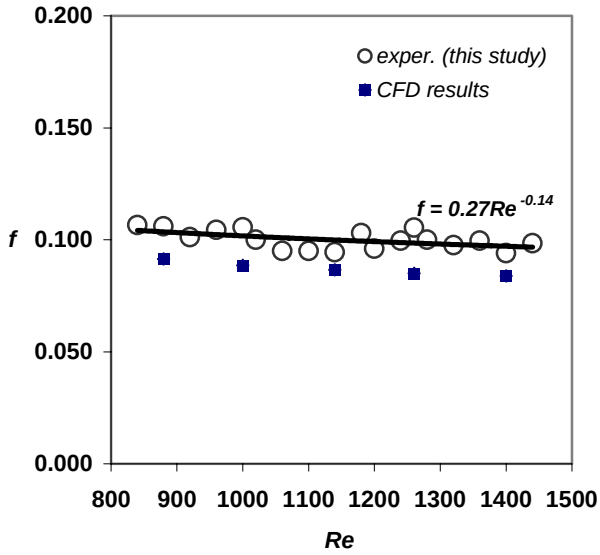


Figure 6. Comparison of friction factor predictions (CFD) with experimental data

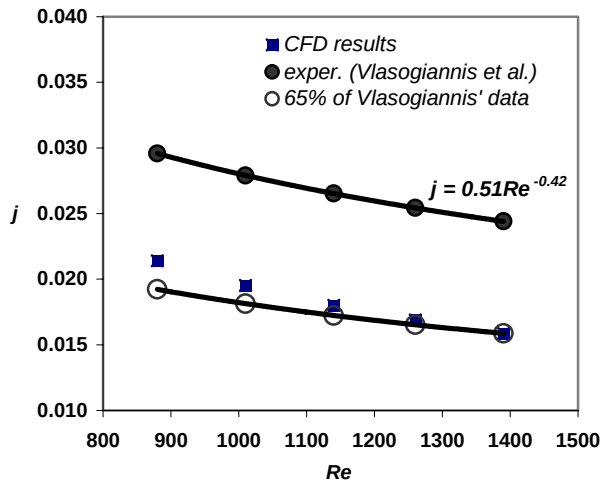


Figure 7. Comparison of *j*-Colburn factor predictions (CFD) with experimental data

3 STUDY OF A HEAT EXCHANGER CHANNEL

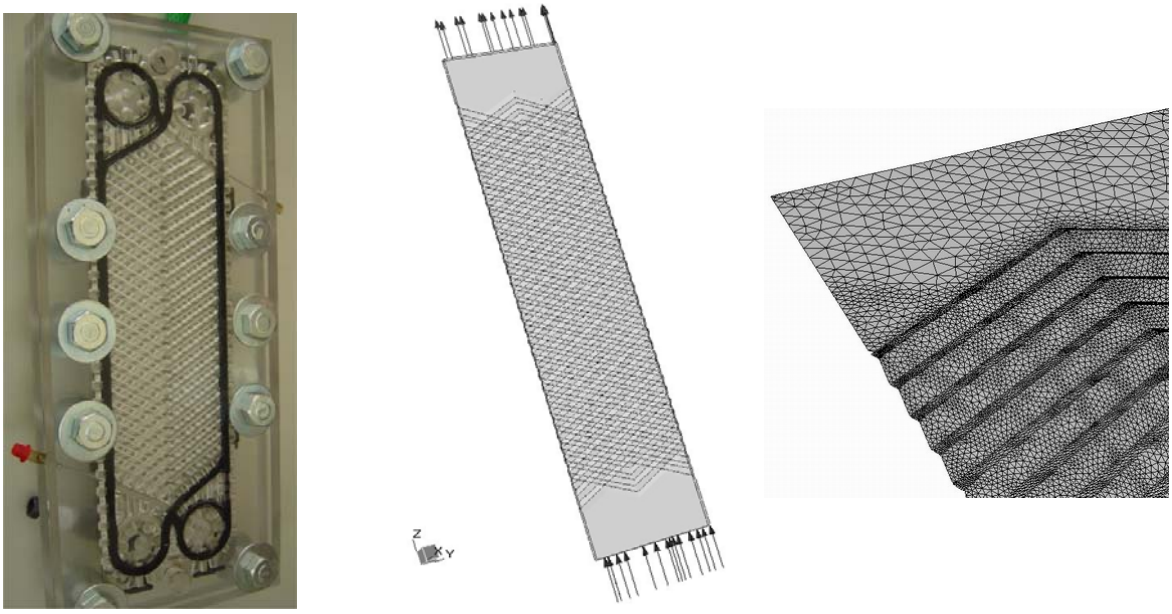
The results for the simplified geometry confirm the validity of the CFD code and strongly encourage the simulation of a module (pass) consisting of two corrugated plates of a compact heat exchanger (***Figure 8a***). In order to quantitatively evaluate the results of this simulation, the experimental setup of Vlasogiannis et al.[5] was used as the design model (***Figure 8b***). Due to the increased computational demands, an AMD AthlonXP 1.7GHz workstation with 1GB RAM was used. The geometric characteristics of the new model are presented in ***Table 3.***

Plate length	0.430 m
Plate width	0.100 m
Mean spacing between plates	0.024 m
Corrugation angle	60°
Corrugation area length	0.352 m

Table 3. Geometric characteristics of the model with two-corrugated plates

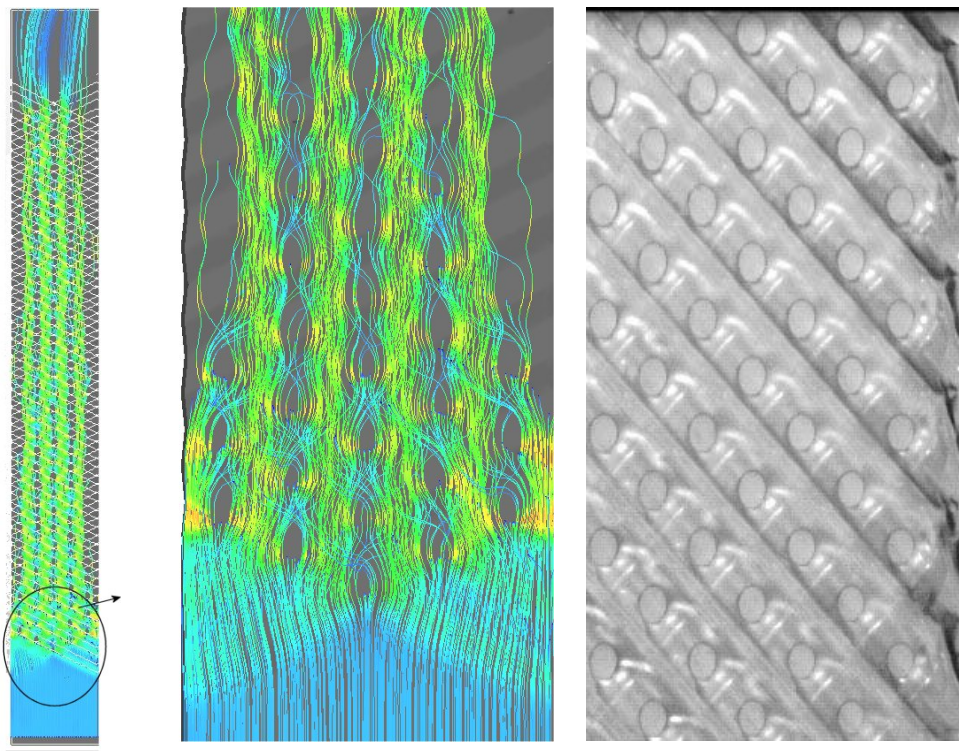
Preliminary results of the present study, which is still in progress, are shown in ***Figure 9***. It is obvious that the herringbone design promotes a symmetric flow pattern (***Figure 8b***). Focusing on the left half of the channel

(***Figure 9a***), a close-up of the flow streamlines (***Figure 9b***) reveals a "*peacock-tail*" pattern as the liquid flows inside the furrows and over the corrugations. The same flow pattern, which is characteristic for this type of geometry, has also been observed by Paras et al.[3] in similar cross-corrugated geometries (***Figure 9c***), where "dry areas" of ellipsoidal shape are formed around the points where the corrugations come into contact. The effect of fluid properties (e.g. surface tension, viscosity) on the shape and the extent of these areas, which are considered undesirable, will be examined in the course of this study.



(a) **(b)** **(c)**

Figure 8. (a) Module of a corrugated plate exchanger; (b) The CFD model and (c) Detail of the grid distribution over the corrugated wall



(a) **(b)** **(c)**

Figure 9. (a) Streamlines in the left half of the channel; (b) Close up of the flow pattern; (c) Photo of the flow in the cross-corrugated geometry[3]

4 CONCLUDING REMARKS

Although compact heat exchangers with corrugated plates offer many advantages compared to conventional heat exchangers, their main drawback is the absence of a general design method. The variation of their basic geometric details (i.e. aspect ratio, shape and angle of the corrugations) produces various design configurations, but this variety, although it increases the ability of compact heat exchangers to adapt to different applications, renders it very difficult to generate an adequate 'database' covering all possible configurations. Thus, CFD simulation is promising in this respect, as it allows computation for various geometries, and study of the effect of various design configurations on heat transfer and flow characteristics.

In an effort to investigate the complex flow and heat transfer inside this equipment, this work starts by simulating and studying a simplified channel and, after gaining adequate experience, it continues by the CFD simulation of a module of a compact heat exchanger consisting of two corrugated plates. The data acquired from former simulation is consistent with the single corrugated plate results and verifies the importance of corrugations on both flow distribution and heat transfer rate. To compensate for the limited experimental data concerning the flow and heat transfer characteristics, the results are validated by comparing the overall Nusselt numbers calculated for this simple channel to those of a commercial heat exchanger and are found to be in reasonably good agreement. In addition, the results of the simulation of a complete heat exchanger agree with the visual observations in similar geometries.

Since the simulation is computationally intensive, it is necessary to employ a cluster of parallel workstations, in order to use finer grid and more appropriate CFD flow models. The results of this study, apart from enhancing our physical understanding of the flow inside compact heat exchangers, can also contribute to the formulation of design equations that could be appended to commercial process simulators. Additional experimental work is needed to validate and support CFD results, and towards this direction there is work in progress on visualization and measurements of pressure drop, local velocity profiles and heat transfer coefficients in this type of equipment.

5 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, September 24-28.

[4] 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.

[5] 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, pp. 757-772.

[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, 80, 6, pp. 667-673.

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

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

[9] Menter, F., Esch, T. (2001), "Elements of Industrial Heat Transfer Predictions", *16th Brazilian Congress of Mechanical Engineering (COBEM), 26-30 Nov. 2001, Uberlandia, Brazil.*

[10] AEA Technology (2003), *CFX Release 5.6 User Guide,* CFX International, Harwell, Didcot, UK.

[11] Wilcox, D. (1988), "Reassessment of the scale-determining equation", AIAA Journal, 26, 11.

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

[13] Versteeg, H.K., Malalasekera, W. (1995), *An introduction to Computational Fluid Dynamics*, Longman London.

[14] Heavner, R.L., Kumar, H., Wanniarachchi, A.S. (1993), "Performance of an industrial plate heat exchanger: effect of chevron angle", *AIChE Symp. Ser. No. 295, Vol. 89, Heat Transfer,* Am. Inst. Chem. Eng., Atlanta, GA, pp. 262-267.

[15] 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, pp. 1469-1497.

[16] Ligrani, P.M., Oliveira, M.M. (2003), "Comparison of Heat Transfer Augmentation Techniques", AIAA Journal, Vol. 41, 3, pp.337-362.