# CFD Modelling of Hydrodynamics and Heat Transfer in Channels of a PHE

Ioannis A. Stogiannis[a], Spiros V. Paras*,[a], Olga P. Arsenyeva[b],
Petro O. Kapustenko[b]

[a]Chemical Engineering Department, Aristotle University of Thessaloniki, Greece
[b]National Technical University "Kharkiv Polytechnic Institute", Kharkiv, Ukraine
paras@auth.gr

Plate Heat Exchangers (PHEs) are one of the most efficient types of contemporary heat exchangers with intensified heat transfer. They are commonly used in process industries due to their compactness, lower weight and cost, smaller space for installation and servicing compared to conventional shell-and-tube heat exchangers. Heat transfer in PHEs takes place in channels of complex geometry formed by corrugated plates placed abutting. The flow in such channels can be very complicated due to breakup and reattachment of the boundary layer, secondary flows and the small hydraulic diameter of the flow passages. The aim of this work is to compare a well-established and validated CFD code both with results obtained from an experimental PHE model using a corrugated plate commonly used in industrial applications and with existing correlations. The results show that CFD simulation can predict heat transfer rate and fluid flow behaviour in a range of Re numbers (8,900 to 27,650), with discrepancies up to 1 % and 6 % in terms of outlet temperature and pressure drop respectively. Modelling of the flow inside a single corrugation also allows for computation of the wall shear stress distribution which can be very useful in PHE applications where fouling is of particular importance.

## 1. Introduction

Heat recuperation is of primary importance for efficient energy usage as it leads to reduction of fossil fuel consumption and greenhouse gas emissions. New challenges arise by integrating renewables, poly-generation and CHP units with traditional sources of heat in industrial and communal sector. In this kind of applications there is a requirement to consider minimal temperature differences in heat exchangers of reasonable size (Fodor et al. 2010) conditions that can be satisfied by a Plate Heat Exchanger (PHE). Heat exchangers of this type are rapidly replacing conventional shell-and-tube heat exchangers in various process industries due to their compactness, higher reliability and operability. Heat transfer in PHEs takes place in channels of complex geometry formed by corrugated plates placed abutting. The flow in such channels can be very complicated due to flow separation and reattachment of the boundary layer, which results in heat transfer enhancement. Detailed and accurate description of the flow and temperature fields in PHE channels is very difficult to experimentally obtain. Most of the authors, who investigated pressure drop and heat transfer in PHEs, have presented their data as empirical correlations of averaged, over the whole channel, friction factor and film heat transfer coefficients. A number of authors made attempts to generalise these correlations. Martin (1996) tried to generalise all the data for hydraulic resistance by a semi-empirical mathematical model. He obtained a relation, which in implicit form expresses the dependence of hydraulic resistance coefficient on Reynolds number and geometry parameters of the plate corrugation. In this case the deviation of the results calculated by the relation from experimental data reported by other authors rises up to 50 % and more. Dović et al. (2009) in their correlation of heat transfer and pressure drop obtained a similar result in terms of accuracy. Since in both aforementioned studies several types of commercial corrugated plates, with various geometrical characteristics have been used to fit the semi-empirical models, it seems to be the main reason for the high deviation of the proposed correlations from the experimental data.

The plate surface of an industrial PHE (Figure 1) consists of the main corrugated field (4) and the zones of flow distribution on the inlet (2) and outlet (5) area. Arsenyeva et al. (2011) proposed models for calculating the pressure drop along the main corrugated field of a PHE channel by using data on friction factor reported by various researchers. For the prediction of film heat transfer coefficient the modified Reynolds analogy of heat vs. momentum transfer was proposed, the validity of which was confirmed by the comparison with experimental data of heat transfer in PHE channels available in the literature. In order to use that analogy, an average wall shear stress in the channel is required. Experimental data of friction pressure drop for PHE channels with various corrugation geometries are used to estimate the wall shear stress used in the analogy proposed (Arsenyeva et al. 2011).
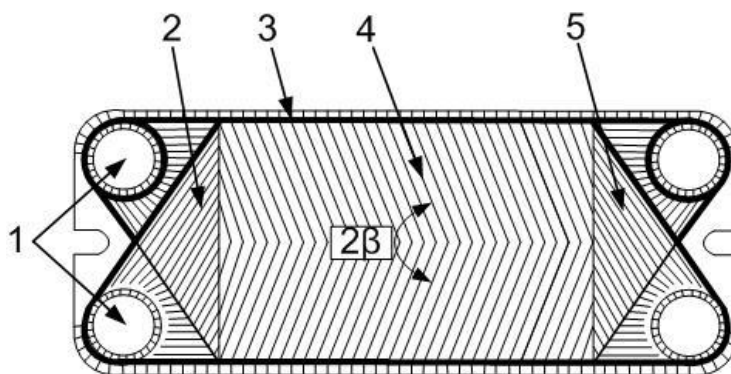


Figure 1: Schematic drawing of PHE plate: 1 inlet and outlet; 2, 5 zones for flow distribution; 3 rubber gasket; 4 main corrugated field.

To confirm this assumption and obtain such a correlation experimentally would be an extremely difficult task. On the other hand, a Computational Fluid Dynamics (CFD) code has been proved a reliable tool for modelling heat transfer and fluid flow inside the PHE channels. The aim of this study is to critically compare the numerical results of a well-established and previously validated CFD code with experimental results of industrial PHE as well as with data from published correlations.

## 2. . Experimental part

The test section consists of four plates depicted in Figure 2 (1,000 mm long by 220 mm wide). They are modulated with corrugations of triangular form (pitch 18 mm, height 5 mm) and which are inclined at an angle of $\beta=45^{o}$ with respect to the channel axis. The plates are clamped together between thermally insulated plates (2) and form three identical channels (Figure 3), tightened by bolts.
In the central channel steam is supplied, which condenses on the walls of the plates. Water is the cold medium flowing in the two outer channels. Consequently, one of the water channel walls is heated by the saturated steam that flows in the central channel, while the other walls are thermally insulated. The inlet and outlet water temperature was measured by copper-constantan thermocouples with accuracy of $\pm 0.1$ $^{o}$C, while its flow rate was measured by a calibrated orifice meter. The flow rate of the condensate was calculated by measuring the quantity collected over a period time in a calibrated vessel. The discrepancies in heat balance for both sides did not exceed $\pm 5$ %.

## 3. CFD modelling

The CFD simulation for heat transfer and pressure drop modelling is focused on a single water channel whose one side has a boundary condition of fixed temperature that is not constant but changes with the plate length (Figure 4a). As the pressure of the saturated steam (which flows outside the channel) decreases (due to friction losses) its saturation temperature also decreases. Using steam tables, the saturation temperatures were estimated for the experimental pressures provided. Thus, a linear relation between temperature and the total length of the plate was assumed and this temperature was imposed as boundary condition on one plate wall while all other walls are considered adiabatic. In the present calculation, the CFD code (ANSYS 14.0) employs a high resolution advection scheme for the discretization of the momentum equations, while for the pressure-velocity coupling SIMPLEC algorithm was used. Relevant work (Kanaris et al. 2006) concerning PHE has proven that the SST turbulence model is the most appropriate for simulating the flow inside this type of conduit. The grid used is an unstructured one

consisting of tetrahedral and prism elements. In order to facilitate the boundary layer calculations, a layer of prism elements is imposed on the vicinity of the walls. A grid dependence study was also performed for choosing the optimum grid size.
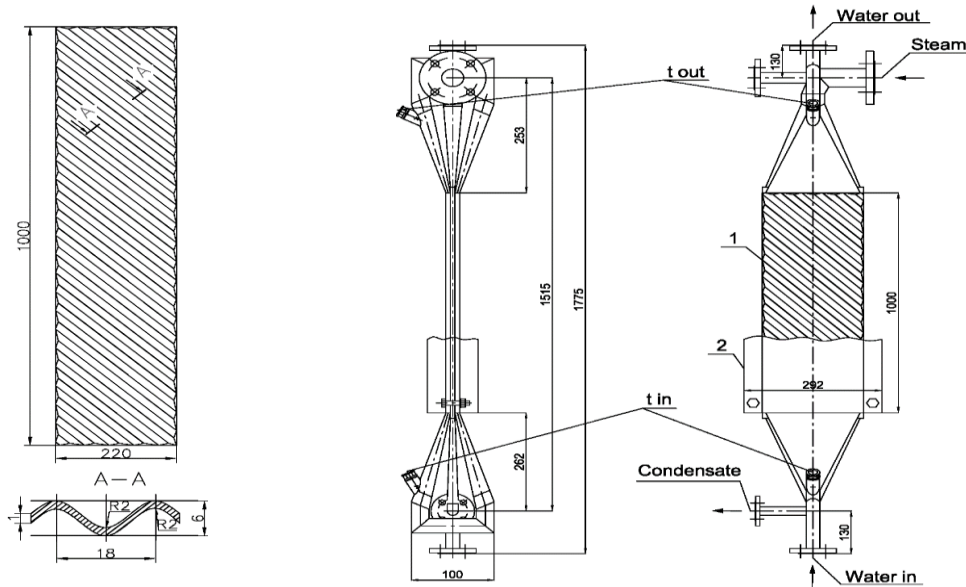


Figure 2: Schematic of the tested plate.

Figure 2: The experimental model of plate condenser (all sizes in mm): 1 tested model; 2 thermally insulated clamping plates.

Besides the simulation of the corrugated flow channel as a whole Ciofalo et al. (1996) proposed CFD models based on a Representative Elementary Unit (REU), i.e. the smallest volume that can be considered as the repeating building block of the channel. In this case, to resolve the complex configuration inside the channel a segment of the flow passage is used as the simulation geometry. This approach allows for significantly smaller mesh elements and consequently for improved computation of the secondary flows and re-attachment of the boundary layer (Mehrabian and Poulter, 2000). Periodic boundary conditions were imposed between the two inlets and outlet faces while on the other walls of the REU no-slip condition was used (Figure 4b). For the translational periodicity interface, the solver implements a mass flow rate by adjusting the pressure change until the mass flow rate converges to the specified boundary condition (ANSYS, 2011). The REU computational geometry is employed in an effort to predict not only an average wall shear stress value but also a detailed wall shear stress distribution on the corrugated wall, useful for PHE applications where fouling is of particular importance.
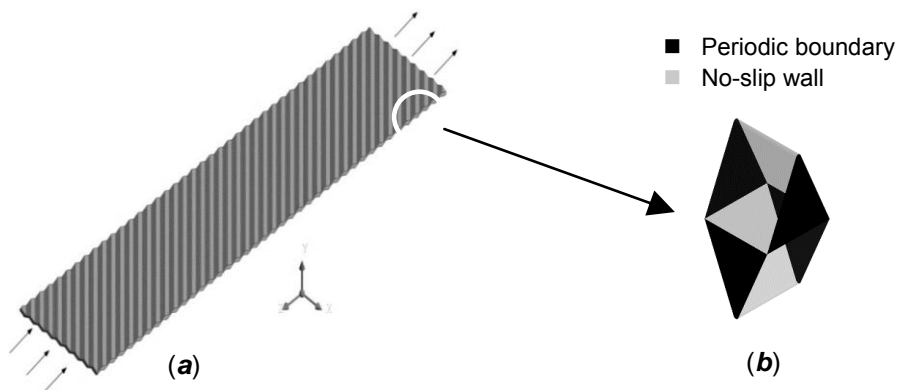


Figure 3: Computational geometry used for the CFD simulations: a) the full water channel, b) the Representative Elementary Unit and periodic boundary conditions.

## 4. Results and discussion

In this paper the results concern four experiments with different water flow rates (Table 1). CFD simulations results are found to be in fairly good agreement with the experimental ones for a range of Reynolds numbers (8,900 to 27,650). The calculated outlet water temperature deviates from the experimentally measured by 1.1 % at maximum. The pressure drop obtained using the CFD code was compared to the one calculated by empirical correlation for corrugated field of such channels (Arsenyeva et al. 2011), in which the accuracy of the correlation was estimated by a mean-square error of ±6.5 %. The maximal discrepancy 5.5 % is well inside the limits of accuracy for probability of 95 % (Figure 5a).

*Table 1. Experimental data and comparisons with CFD simulations – without inlet and outlet zones effect.*

| Experiment number | 1 | 2 | 3 | 4 |
|---|---|---|---|---|
| Water flow rate, kg/s | 0.596 | 0.772 | 0.833 | 0.283 |
| Average velocity in channel, m/s | 0.56 | 0.73 | 0.79 | 0.27 |
| Reynolds number | 17,750 | 25,450 | 27,650 | 8,900 |
| Inlet steam temperature, °C | 113.4 | 107.4 | 102.4 | 101.1 |
| Outlet steam temperature, °C | 108.4 | 104.9 | 102.1 | 101 |
| Inlet water temperature, °C | 82.9 | 97.7 | 98.6 | 94.4 |
| Measured outlet water temperature, °C | 95.6 | 101.8 | 100.5 | 98.6 |
| Calculated outlet water temperature (CFD), °C | 96.6 | 101.8 | 100.4 | 98 |
| Difference between experimental and calculated values, % | -1.1 | 0 | 0.1 | 0.6 |
| Measured pressure drop (Arsenyeva et al. 2011), kPa | 5.29 | 8.53 | 9.81 | 1.34 |
| Calculated pressure drop (CFD), kPa | 5.17 | 8.95 | 10.39 | 1.35 |
| Difference between experimental and calculated values, % | -2.3 | 4.7 | 5.6 | 0.3 |

On the other hand the results of pressure drop calculated by correlations proposed by Martin (1996) and for similar corrugated PHE plates give the resulting pressure drops about two times higher (Table 2). Another correlation by Kanaris et al. (2009) gives similar results in terms of pressure drop. This difference is mainly attributed to the inlet and outlet distribution zones (Figure 1), which are included in the aforementioned correlations but they have not be taken into account in the first approach of CFD modelling (Table 1). It is obvious that the distribution zones greatly contribute to the total pressure drop value in a PHE channel. To estimate the influence of flow inlet and outlet zones on pressure drop in a PHE channel, CFD simulations were also performed by blocking half of the channel cross sections at inlet and outlet. In this case (Figure 5b) the total pressure drop in the channel has been almost doubled, i.e. it is close to the one calculated by the generalised correlations for similar industrial PHE channels. The employment of the generalised correlation should be done under the consideration that they are validated to a specific range of Re number and their use out of this range can be unsatisfactory in terms of accuracy. We can conclude that for the investigated PHE channel the pressure drop in distribution zones on channel inlet and outlet can constitute about 50 % of total pressure drop. On the other hand, the outlet water temperature predicted by CFD in the channel of half blocked inlet and outlet changed rather little compare to the channel with free inlet and outlet (Table 2). A possible explanation would be the significantly augmented heat transfer in the undulated surface of the main flow field which dominates the total heat transfer rate in a PHE. This leads to the conclusion that in a PHE channel the distribution zones have a lower impact on heat transfer than on pressure drop. Wang et al. (2007) have reported similar impact of the flow maldistribution due to inlet effects on the overall thermal performance of a PHE. Thus, special attention must be paid during the design of the aforementioned zones, since their contribution to pressure loss is considerably higher than their influence on heat transfer enhancement.

*Table 2. Experimental data and comparisons with CFD simulations – inlet and outlet zones effect.*

| Experiment number | 1 | 2 | 3 | 4 |
|---|---|---|---|---|
| Pressure drop (Kanaris et al. 2009), kPa | 9.51 | 15.33 | 17.66 | 2.37 |
| Pressure drop (Martin, 1996), kPa | 8.81 | 14.21 | 16.36 | 2.19 |
| Pressure drop (CFD), kPa | 10.41 | 16.87 | 19.53 | 2.59 |
| Calculated outlet water temperature (CFD), °C | 97.3 | 98.6 | 102.4 | 100.6 |

In Figure 4b there are also presented the results of calculations by empirical correlation of paper Arsenyeva et al. (2011) with addition of local hydraulic resistance in distribution zones, as it was suggested by Arsenyeva et al. (2013). The coefficient of local hydraulic resistance was taken equal to 38 for both zones together. Such method is also giving good results and allows for separate accounting of the

hydraulic resistance into two important components of the channel of industrial PHE: the main corrugated field and the flow distribution zones.
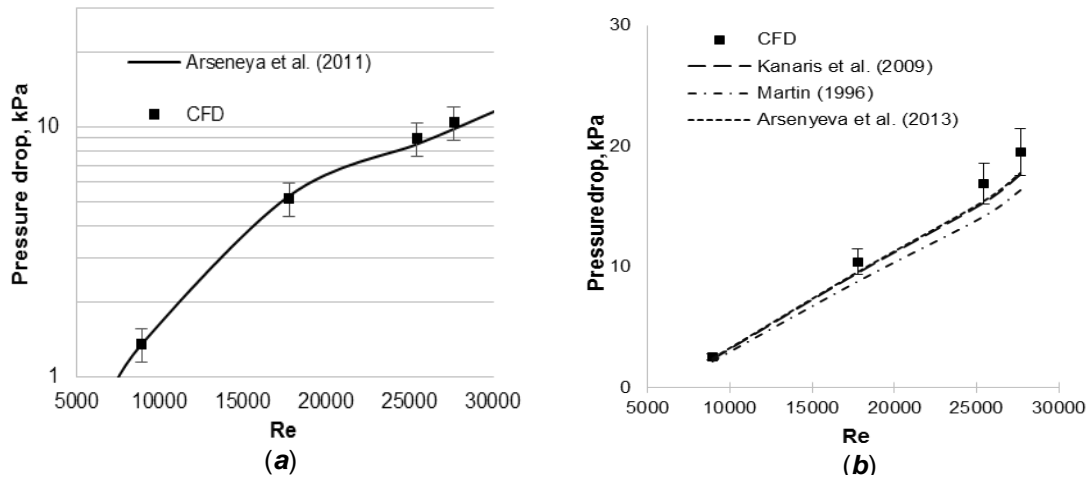


*Figure 4: Comparison of pressure drop CFD results with empirical correlation predictions: a) without inlet-outlet zone effects, b) including inlet-outlet zone effects.*

One of the main factors governing the transfer processes in flow regions close to the channel walls is the wall shear stress. The total pressure loss in a PHE channel of complex geometry consists of the pressure loss due to friction on the wall and of the form drag due to the pressure redistribution on the corrugated channel walls in three dimensional flows. To estimate the contribution of pressure losses due to friction, a semi empirical correlation was proposed (Arsenyeva et al. 2012) based on the analogy of heat and momentum transfers. As local wall shear stresses measurements are difficult to be realised, good estimations can be obtained by applying a validated (in terms of heat transfer and pressure drop) CFD code. In order to get an average shear stress value but also a detailed distribution of wall shear stresses the REU modelling approach has been used. In Table 3 the results of average shear stress calculated both by CFD and by an existing correlation (Arsenyeva et al. 2012) are presented. The deviation between the results obtained by CFD and the proposed correlation is less than ±10.0 %, which can be regarded as a fairly good agreement. Consequently, it is possible to predict the share of pressure losses due to friction in PHE channels, using a CFD code. This can be useful in PHE applications, where fouling is of critical importance as pressure losses due to friction are used to calculate fouling thermal resistance. For the calculation of the fouling thermal resistance a mean wall shear stress over the entire plate is commonly used. Computational results show that fluid flow inside the narrow passages formed by the plate's corrugations is complicated due to large secondary flows. This leads to a wide distribution of the shear stress over the corrugated walls, as shown in Figure 6. Using the CFD results it is possible to locate the regions of locally decreased wall shear stresses and compare them with experimental results of fouling deposition distribution.
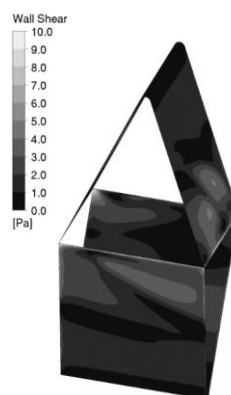


*Figure 5: Wall shear stress distribution on a REU of a PHE (Experiment no. 1, Re = 8,900).*

Table 3: Comparison of wall shear stress calculated by the CFD and predicted by empirical correlation.

| Experiment number | 1 | 2 | 3 | 4 |
|---|---|---|---|---|
| Wall shear stress correlation (Arsenyeva et al. 2012) , Pa | 7.5 | 11.7 | 13.3 | 2.1 |
| Wall shear stress (CFD), Pa | 7.0 | 11.4 | 12.0 | 2.2 |
| Difference in shear stress calculations, % | 7.3 | 2.0 | 10.1 | -6.3 |

## 5. Conclusions

In this work, it has been proved that CFD is a reliable tool for simulating the operation of a PHE. So instead of expensive and time consuming laboratory experiments, a validated CFD code can be used for accurately predicting heat transfer rates and pressure drop in a PHE with various geometrical characteristics. Additionally, the influence in the PHE performance of specific parts, like the inlet-outlet distributions zones, can be assessed. Moreover, CFD results can be used for the generation of new heat transfer and pressure drop correlations as well as for the assessment of existing empirical correlations found in literature. Wall shear stress for PHE channel geometries can be also calculated by the CFD code and the deviation from a proposed semi-empirical correlation, typically used in the industry, is less than ±10.0 %. Finally, using the CFD code is possible to predict the shear stress distribution on the walls of a corrugated plate and consequently predict areas of increased deposition rate.

## Acknowledgements

## References

Ansys 2011. ANSYS CFX Solver Theory Guide Release 14.0. Ansys Inc. International, Canonsburg, USA.

Arsenyeva O., Kapustenko P., Tovazhnyanskyy L., Khavin G., 2013, The influence of plate corrugations geometry on plate heat exchanger performance in specified process conditions. Energy, 57, 201-207

Arsenyeva O., Tovazhnyansky L., Kapustenko P., Khavin, G., 2011, The Generalized Correlation for Friction Factor in Criss-cross Flow Channels of Plate Heat Exchangers, Chemical Engineering Transactions 25, 399-404.

Arsenyeva O., Tovazhnyanskyy L., Kapustenko, P., Demirskiy O., 2012, Heat transfer and friction factor in criss-cross flow channels of plate-and-frame heat exchangers, Theor. Found. Chem. Eng., 46, 634-641.

Ciofalo M., Stasiek J., Collins M.W., 1996, Investigation of flow and heat transfer in corrugated passages .2. Numerical simulations, Int. J. Heat Mass Transfer, 39, 165-192.

Dović D., Palm B., Švaić S., 2009, Generalized correlations for predicting heat transfer and pressure drop in plate heat exchanger channels of arbitrary geometry, Int. J. Heat Mass Transfer, 52, 4553-4563.

Fodor Z., Varbanov P.S. , Klemeš J.J., 2010, Total site targeting accounting for individual process heat transfer characteristics. Chemical Engineering Transactions 21, 49-54.

Kanaris A.G., Mouza A. A., Paras S. V., 2006, Flow and heat transfer prediction in a corrugated plate heat exchanger using CFD code, Chem. Eng. Technol, 29, 923-930.

Kanaris A.G., Mouza A.A., Paras S. V., 2009, Optimal design of a plate heat exchanger with undulated surfaces, Int. J. Therm. Sci., 48, 1184-1195.

Martin H., 1996, A theoretical approach to predict the performance of chevron-type plate heat exchangers, Chem. Eng. Process. Process Intensif, 35, 301-310.

Mehrabian M.A., Poulter R., 2000, Hydrodynamics and thermal characteristics of corrugated channels: computational approach, Appl. Math. Model., 24, 343-364.

Wang L., Sundén B., Manglik R.M., 2007, Plate heat exchangers: design, applications and performance. WIT Press, Southampton, UK