CFD Modelling of Turbulent Hydrodynamics and Heat Transfer: Application for PHE Channels

I.A. Stogiannis, **S.V. Paras**, **O. Arsenyeva***, **P. Kapustenko***
Aristotle University of Thessaloniki, Chemical Engineering Department,
Univ. Box 455, GR 54124 Thessaloniki, Greece
paras@auth.gr
* National Technical University “Kharkiv Polytechnic Institute”,
21 Frunze Str., 61002 Kharkiv, Ukraine
kap@kpi.kharkov.ua

Abstract

Plate Heat Exchangers (*PHEs*) are increasingly used in process industries due to their compactness, as well as higher reliability and operability compared to conventional shell-and-tube heat exchangers. Heat transfer in *PHE* 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 experimental results obtained from industrial applications of *PHE* 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 (3,200 to 9,450), with discrepancies up to 1% and 6% in terms of outlet temperature and pressure drop respectively.

1. Introduction

Heat recuperation is of primary importance for efficient energy usage with consequent reduction of fossil fuel consumption and greenhouse gas emissions. New challenges arise by integrating renewables, polygeneration and *CHP* units with traditional sources of heat in industrial and communal sector. In this kind of application there is a requirement to consider minimal temperature differences in heat exchangers of reasonable size [1]. Such conditions can be satisfied by a Plate Heat Exchanger (*PHE*). Heat exchangers of this type are rapidly adapted by various process industries because of their compactness, higher reliability and operability replacing conventional shell-and-tube heat exchangers. Heat transfer in *PHE* 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 the enhancement of the heat transfer. The detailed and accurate description of the flow and temperature fields in *PHE* channels is very difficult to obtain experimentally. 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. The survey of those correlations can be found in a book by Wang et al. [2] and papers by Khan et al. [3], Dovic et al. [4]. A number of authors made attempts to generalise these correlations. Martin [5] 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. [4] in their correlation of heat transfer and pressure drop obtained a similar result in terms of accuracy. Since in both aforementioned studies [4,5] 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. [6] 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 [6].

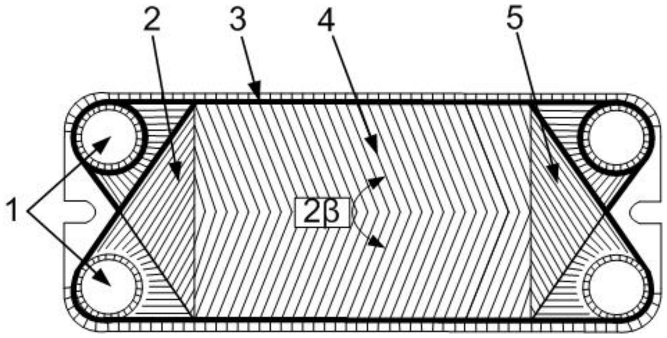

Figure 1. Schematic drawing of PHE plate: 1 – inlet and outlet;
2, 5 – zones for flow distribution; 3 – rubber gasket; 4 – main corrugated field.

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*** (1000 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.

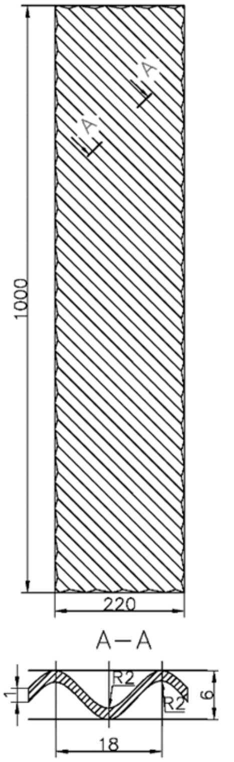

Figure 2. Schematic of the tested plate.

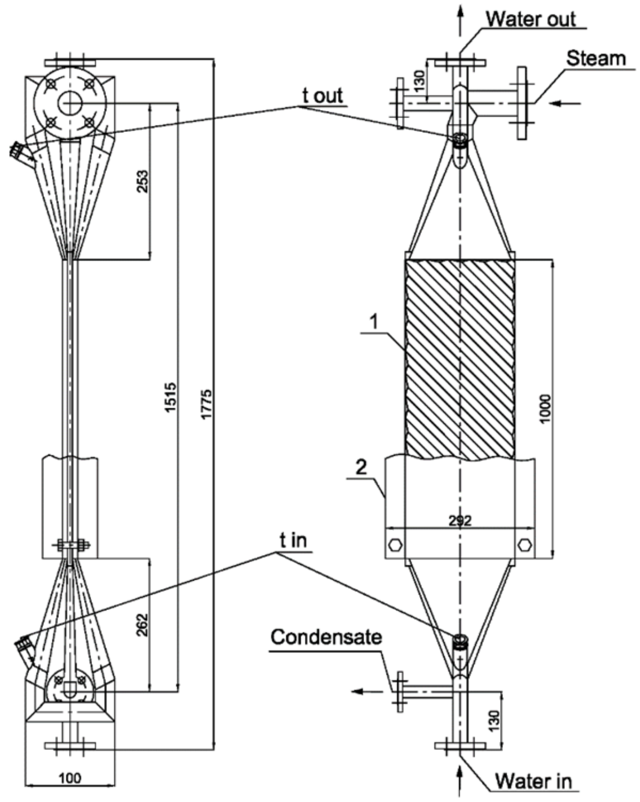

Figure 3. The experimental model of plate condenser (all sizes in mm): 1 – tested model; 2 – thermally insulated clamping plates.

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 ±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 ±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. 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 [7] 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.

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 (3,200 to 9,450). The calculated outlet water temperature deviates from the experimentally measured by 1.1% at maximum.

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

Experiment number	**1220**	**1245**	**1247**	**1251**
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	*6750*	*8750*	*9450*	*3200*
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 [6], kPa	*5.79*	*9.29*	*10.69*	*1.50*
Calculated pressure drop (*CFD*), kPa	*5.97*	*9.18*	*11.31*	*1.47*
Difference between experimental and calculated values, %	*3.0*	*-1.2*	*5.5*	*-2.1*

The pressure drop obtained using the *CFD* code was compared to the one calculated by empirical correlation for corrugated field of such channels [6], 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 4a***). On the other hand the results of pressure drop calculated by correlations proposed by Martin [5] and Kanaris et al. [7] for similar corrugated *PHE* plates give the resulting pressure drops about two times higher (***Table 2***). 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 4b***) the total pressure drop in the channel has been almost doubled, i.e. it is close to the one calculated by the generalised correlations [5,7] for similar industrial *PHE* channels. 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***).

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

Experiment number	**1220**	**1245**	**1247**	**1251**
Pressure drop (correlation [7]), kPa	*10.46*	*16.96*	*19.56*	*2.61*
Pressure drop (correlation [5]), kPa	*9.69*	*15.71*	*18.12*	*2.42*
Pressure drop (*CFD*), kPa	*10.41*	*17.55*	*20.46*	*2.71*
Calculated outlet water temperature (*CFD*), °C	*97.3*	*98.6*	*102.4*	*100.6*

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. 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. In ***Table 2*** there are also presented the results of calculations by empirical correlation of paper [6] with addition of local hydraulic resistance in distribution zones, as it was suggested by Arsenyeva et al. [9]. 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.

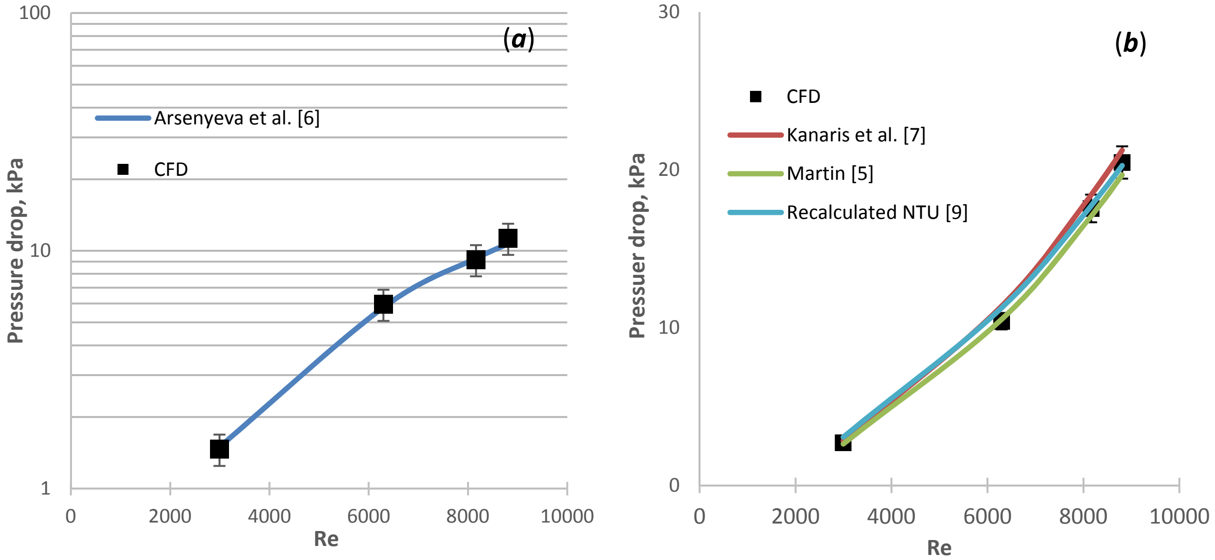

Figure 4. Comparison of pressure drop *CFD* results with empirical correlation predictions: a) without inlet-outlet zone effects, b) including inlet-outlet zone effects.

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 [8] 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 ***Table 3*** the results of average shear stress calculated both by *CFD* and by an existing correlation [8] 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.

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

Experiment number	**1220**	**1245**	**1247**	**1251**
Wall shear stress by correlation of paper [8], Pa	*8.85*	*13.9*	*15.9*	*2.45*
Wall shear stress (*CFD*), Pa	*8.4*	*13.3*	*14.7*	*2.6*
Difference in shear stress calculations, %	*7.3*	*6.4*	*9.3*	*-1.6*

5. Conclusions

In this work, it has been proven that *CFD* is a reliable tool for simulating the operation of a *PHE*. So instead of expensive and time consuming laboratory experiments, *CFD* simulations can be used for accurately predicting heat transfer rates and pressure drop in a *PHE*. Additionally, the influence in the *PHE* performance of specific parts, like the inlet-outlet distributions zones, can be assessed. Wall shear stress distribution for *PHE* channel geometries can be also calculated by the *CFD* code with acceptable deviation from proposed semi-empirical correlations, typically used in the industry.

Acknowledgements: Financial support from the *EC* Project *DISKNET* (*FP7-PEOPLE-2011-IRSES-294933*) is sincerely acknowledged.

References

1. Fodor Z., Varbanov P. and Klemeš J., 2010, Total site targeting accounting for individual process heat transfer characteristics. Chemical Engineering Transactions, 21, 49-54.
2. Wang L., Sunden B., Manglik R.M., 2007, PHEs. Design, Applications and Performance. WIT Press, Southhampton, 270 p.
3. Khan T.S., Khan M.S., Chyu M-C, Ayub Z.H., 2010, Experimental investigation of single phase convective heat transfer coefficient in a corrugated plate heat exchanger for multiple plate configurations. Applied Thermal Engineering, 30 (8-9), 1058–1065.
4. Dović D, Palm B., Švaić S, 2009, Generalized correlations for predicting heat transfer and pressure drop in plate heat exchanger channels of arbitrary geometry. International Journal of Heat and Mass Transfer, 52 (19-20), 4553-4563.
5. Martin H., 1996, Theoretical approach to predict the performance of chevron-type plate heat exchangers. Chem. Eng. Process., 35, 301–310.
6. Arsenyeva O., Tovazhnyansky L., Kapustenko P., Khavin G., 2011, The generalized correlation for friction factor in crisscross flow channels of plate heat exchangers. Chemical Engineering Transactions, 25: 399-404.
7. Kanaris A.G., Mouza A.A., Paras S.V., 2009, Optimal design of a plate heat exchanger with undulated surfaces. Int. J. Therm. Sc., 48**,** 1184-1195.
8. Arsenyeva O.P., Tovazhnyanskyy L.L., Kapustenko P.O., Demirskiy O.V., 2012, Heat transfer and friction factor in criss-cross flow channels of plate-and-frame heat exchangers. Theoretical Foundations of Chemical Engineering, 46 (6), 634-641.
9. Arsenyeva O., Kapustenko P., Tovazhnyansky L., Khavin G., 2013, The influence of plate corrugations geometry on plate heat exchanger performance in specified process conditions, Energy, http://dx.doi.org/10.1016/j.energy.2012.12.034 (*in press*)