# Evaluation of the boundary conditions in CFD modeling of heat transfer in the 3D chevron type plate heat exchanger

Tiina M. Pääkkönen, Markus Riihimäki, Reeta Ylönen, Esa Muurinen, Riitta L. Keiski

University of Oulu, Department of Process and Environmental Engineering,
Laboratory of Mass and Heat Transfer Processes,
P.O.Box 4300, FI-90014 University of Oulu, Finland

Key words: boundary condition, CFD, heat transfer, modeling, plate heat exchanger

## INTRODUCTION

### *Backround*

New heat exchanger geometries are traditionally developed by the trial and error method using some kind of heuristics. Theoretical predictions of the thermal efficiency of plate heat exchangers would facilitate design of new heat exchangers. Also accurate prediction of reactions, heat transfer and fluid flow in different heat exchanger geometries would help to minimize fouling of heat exchanges by geometrical changes.

Fouling, deposition of unwanted material on the heat transfer surface, diminish the heat transfer and increase the pressure drop. Because of deposited material the energy efficiency of the heat exchanger is lowered. The flow resistance, caused by fouling layer, increases the pressure drop, since pumping power must be added. Because of these factors, energy consumption and operation costs of the heat exchangers, which are due to over sizing, additional cleaning costs and process shut downs, are growing. By decreasing fouling of heat exchangers, energy demand and hence climate effects, like carbon dioxide emissions, caused by energy production can be reduced. Also other environmental effects are reduced since need of chemicals used to cleaning of heat exchangers and amount of unusable plates are decreased.

Accurate heat transfer modeling is an essential part of fouling models, because temperature has a considerable effect on many fouling mechanisms. Without physically correctly defined boundary conditions neither heat transfer nor fouling can be modeled reliably. Selection of boundary conditions is complicated especially in the case of complex geometry of corrugated heat exchangers where distribution of the local heat transfer coefficient fluctuates ([1], [2]).

### *Objectives*

Reliable heat transfer modeling in corrugated plate heat exchangers is complicated because temperature changes in both sides of the heat transfer surface. On this account capability of simulation software's built-in boundary conditions were tested in order to find out their constraints.

Thus the objective of this work was to model fluid flow and heat transfer in the corrugated 3D plate heat exchanger geometry with a commercial computational fluid dynamics (CFD) program, Fluent 6.1, and to find the most realistic heat transfer boundary conditions for a plate heat exchanger and to evaluate the limitations of different boundary conditions. For the verification of the model flows with Reynolds numbers between 1650 and 3100 were investigated and results of those simulations were compared to the experimental correlations.

## MATERIALS AND METHODS

### *Structure of the heat exchanger*

The plate heat exchanger studied was chevron type (M15-M) with the corrugation angle of 60 ° and made by Alfa Laval. The plate heat exchanger consists of several thin, corrugated plates, which are compressed together and sealed with gaskets. In every second plate the corrugated herringbone pattern goes upwards and in every second downwards and hence complicated passages are formed between plates. A corrugated flow channel generates vortices even with low flow velocities. In that case mixing of the fluid is increased and thus heat transfer becomes more effective. In the cannels warm and cold flows alternate and heat transfers through the plates by conduction. Heat transfer by forced convection also exists due to the fluid flow. Heat transfer by radiation can be neglected because temperature in the studied plate heat exchanger is quite low (max. 378 K).

Accurate modeling of the whole plate heat exchanger with CFD was not feasible because of limited computational capacity. For the modeling a small part of the heat exchanger structure, which describes the physical phenomena to be modeled, should be selected. Other authors (Zettler et al. [2], Mehrabian et al. [3], Ciofalo et al. [4]) in the previous studies have used one or two waves in their geometries and periodic boundary conditions to make fluid flow fully developed. However, in our case heat transfer does not have periodic nature, because of changing temperature of the plates, and that is why periodic boundary conditions could not be used. In this case a bigger geometry, which also includes the change of wave direction, was used to ensure fully developed flow. Thus, a flow channel between two plates of dimensions of 0.14 m x 0.07 m and with several waves was chosen to modeling (Figure 1). The geometry was meshed with 408 000 unstructured elements since structured mesh was not possible to generate because of the very complex geometry. For the construction material physical properties of titanium and for the fluid properties physical properties of water were used. Temperatures and other parameters needed in modeling were taken from an industrial case, where the cool process fluid (T = 353 K) is heated with the warm district heat water (T = 363K). Information of fluids needed in calculation was obtained by thermal analysis of the plate heat exchanger [5].

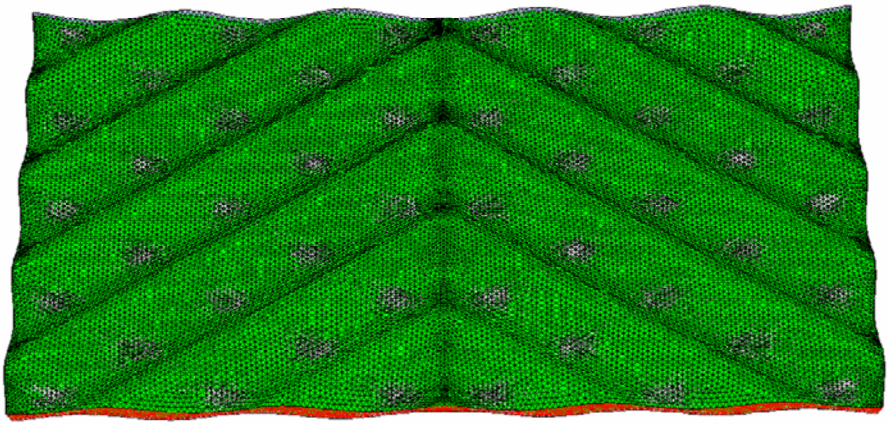

Figure 1. 3-dimensional herringbone pattern plate heat exchanger geometry, size of 0.14 m x 0.07 m x 0.008 m, with a computation mesh.

### *Selection of the flow model and the boundary conditions*

Modeling of the fluid flow at the geometry was based on the Navier-Stokes equations. Reynolds number based on the mean hydraulic diameter of the flow channel is between 1600 and 3100. According to previous studies ([2], [3]) the flow field is neither fully laminar nor turbulent and is thus on transition zone between those Reynolds numbers and that is why different results were obtained with direct numerical simulation and turbulence model. Thus both laminar and turbulence models were tested in this study. For turbulence modeling RNG k-ε turbulence model was used since it is the most suitable turbulence model for quite small Reynolds numbers, when flow is not necessarily fully turbulent [6].

Solving of the flow field is based on general Navier-Stokes equations (1) and (2), from which (1) is continuity equation, (2) momentum equation and (3) shear stress tensor vector of momentum equation. Temperature field of the geometry is solved using energy equation (4). [6]

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{u}) = S_m \ , \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \bar{u}) + \nabla \cdot (\rho \bar{u} \bar{u}) = -\nabla p + \nabla \cdot \bar{\bar{\tau}} + \rho \bar{g} + \bar{F} \ , \tag{2}$$

$$\bar{\bar{\tau}} = \mu \left[ \left( \nabla \bar{u} + \nabla \bar{u}^T \right) - \frac{2}{3} \nabla \cdot \bar{u} I \right] , \tag{3}$$

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot \left( \bar{u} (\rho E + p) \right) = \nabla \cdot \left( k_{eff} \nabla T - \sum_j h_j \bar{J}_j + \left( \tau_{eff} \cdot \bar{u} \right) \right) + S_h \, . \tag{4}$$

At the inlet boundary velocity of inlet flow was defined and at the outlet boundary the pressure was set. For the plate walls and sides of the geometry no-slip boundary conditions were used. For the walls thermal boundary conditions were also used in order to model heat transfer.

In this study different heat transfer boundary conditions of the computational fluid dynamics (CFD) program, Fluent 6.1, were tested and their applicability in modeling complicated heat transfer geometry was studied and discussed. Build-in boundary conditions of Fluent 6.1 available for this case are Convection, Heat flux and Constant wall temperature.

At the Convection boundary condition the values of the outer fluid heat transfer coefficient and temperature are defined. The program calculates heat flux to the wall according to the definition with Equation (5) [6]:

$$q = h_f\left(T_w - T_f\right) + q_{rad} = h_{ext}\left(T_{ext} - T_w\right). \tag{5}$$

While using the Heat flux boundary condition an appropriate value for the heat flux at the wall surface is defined, when Fluent uses Equation (6) to calculate the surface temperature of the wall, where as fluid-side heat transfer coefficient is computed based on the local flow-field conditions [6]:

$$T_w = \frac{q - q_{rad}}{h_f} + T_f\,. \tag{6}$$

At the Constant wall temperature boundary condition the temperature of the wall is defined. According to those definitions program calculates the temperature field in the geometry with Equation (7) [6]:

$$q = h_f\left(T_w - T_f\right) + q_{rad}\,. \tag{7}$$

With the geometry used none of the three alternative thermal boundary conditions describes exactly the physical situation in the plate heat exchanger.

## RESULTS AND DISCUSSION

### *Numerical simulations with different Reynolds numbers and flow models compared with experimental data*

To validate the flow model the fluid flow at the corrugated heat exchanger geometry was modelled with five different flow velocities, which indicate Reynolds number of 1650, 2020, 2370, 2470 and 3100. Both straight simulation and RNG k-ε turbulence models were used to all flow velocities. The Fanning friction factor, which indicates the pressure drop in the geometry, was calculated from the results and plotted as a function of Reynolds number. The simulated results were compared with the experimental correlation found in the literature. Those experimental correlations were done with slightly different Alfa Laval plate heat exchanger geometries than the one used in this study. In spite of that those experimental correlations were used to evaluate the quality of the simulation results. Geometrical parameters of different heat exchanger plates are shown in the Table 1.

Table 1. Geometrical parameters of the plates.

| Plate type | Amplitude (m) | Corrugation angle | Wave length (m) |
|---|---|---|---|
| M15 | 0.0040 | 60° | 0.0140 |
| P01* | 0.0012 | 60° | 0.0103 |
| M3* | 0.0012 | 60° | 0.0103 |
| M6* | 0.0010 | 60° | 0.0110 |

*Reference [2].

Comparison between the experimental correlations and the simulations are shown in Figure 2. It can be seen that both simulation models under predict the fanning factor and thus the pressure drop. However, it should be noticed that in correlation of Zettler (M3) also distributors and ports were included in the measurements. In the correlation of Alfa Laval (P01) channel, which includes the distribution sector, between plates was considered. In that case the difference compared to the simulations is smaller. The best agreement between the correlations and simulations was achieved when comparing simulations to the correlation of Tribbe (M6) in which only corrugated section was considered and the dimension of the wavelength was closest to the simulated geometry. It can be noted that the wave length of the correlation of Tribbe is larger, compared to the other correlations, which usually reduces the pressure drop [2]. It should be noted that also an increase in amplitude usually reduces the pressure drop. Thus, the differences between the experimental correlations and simulations can be explained with the differences of geometrical parameters.

Difference between the laminar and the RNG k-ε model is quite small, but the laminar model under predicts more the Fanning friction factor. The difference between laminar and turbulent models may indicate that the flow is quite perturbed in the corrugated flow field and inadequate for the laminar model. This result is consistent with the results of Ciofalo et al. [4].

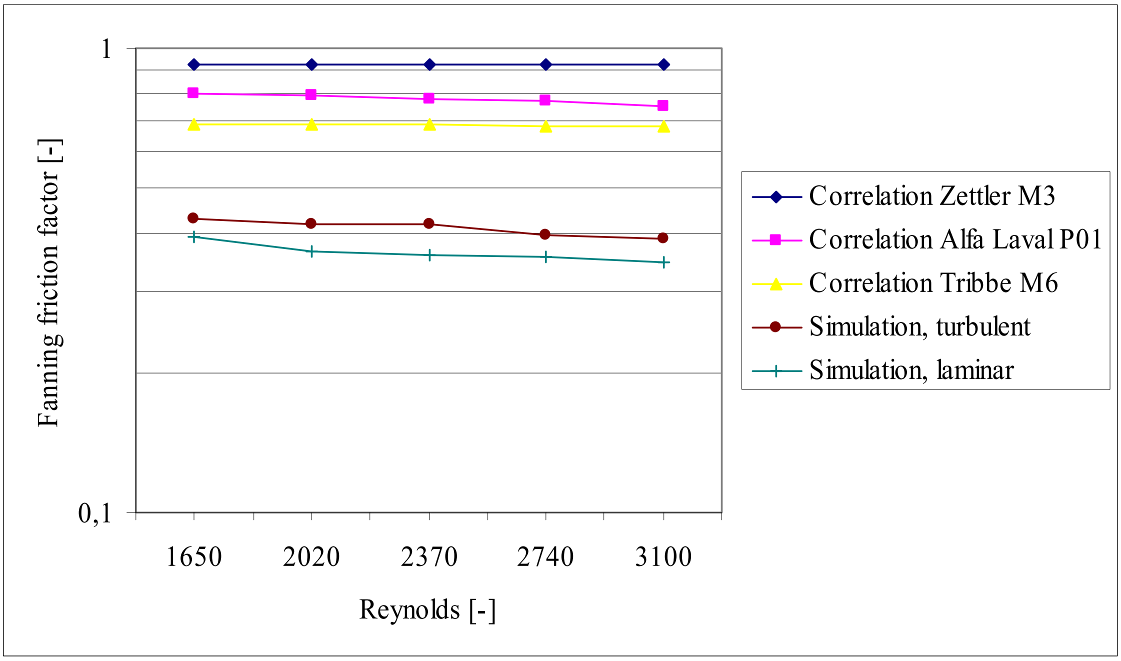


*Figure 2. Comparison of the Fanning friction factors of the experimental correlations [2] and the CFD simulations with different Reynolds numbers.*

### *Evaluation and selection of the heat transfer boundary conditions*

While using the Constant wall temperature boundary condition, the temperature of the plate was defined to be 353 K, the same as the average temperature of the process fluid flowing outside of the geometry. The heat flux through the wall and the temperature field in the geometry have been computed. The temperature defined constant at walls is definitely an inaccurate assumption in this case of heat exchanger because temperature of the fluid on the other side and thus also heat flux changes during the time. Temperature of the plate has also local variations caused by corrugations.

With the Heat flux boundary condition the heat flux had to be defined as a constant at wall. In this study empirical Nusselt number correlation with the overall mass and heat balances was used to estimate the heat flux [7]. The local heat transfer coefficient and furthermore the flux vary spatially at the wall. Thus, the constant overall heat transfer coefficient is not an exact approximation. Furthermore to design a new heat exchanger geometry it would be beneficial to calculate the heat flux in order to find out performance of the structure, not to define it.

While using the Convection boundary condition the temperature and the heat transfer coefficient of surroundings need to be defined as constants. The surrounding fluid has spatial temperature variations similar as flow inside the geometry. However, the heat flux to the wall and the heat transfer coefficient are computed for the fluid inside the modeled channel which means that its dependence on temperature and flow rate is being taken into account. Figure 3 presents the temperature contour of the geometry when heat transfer was calculated with the Convection boundary condition.

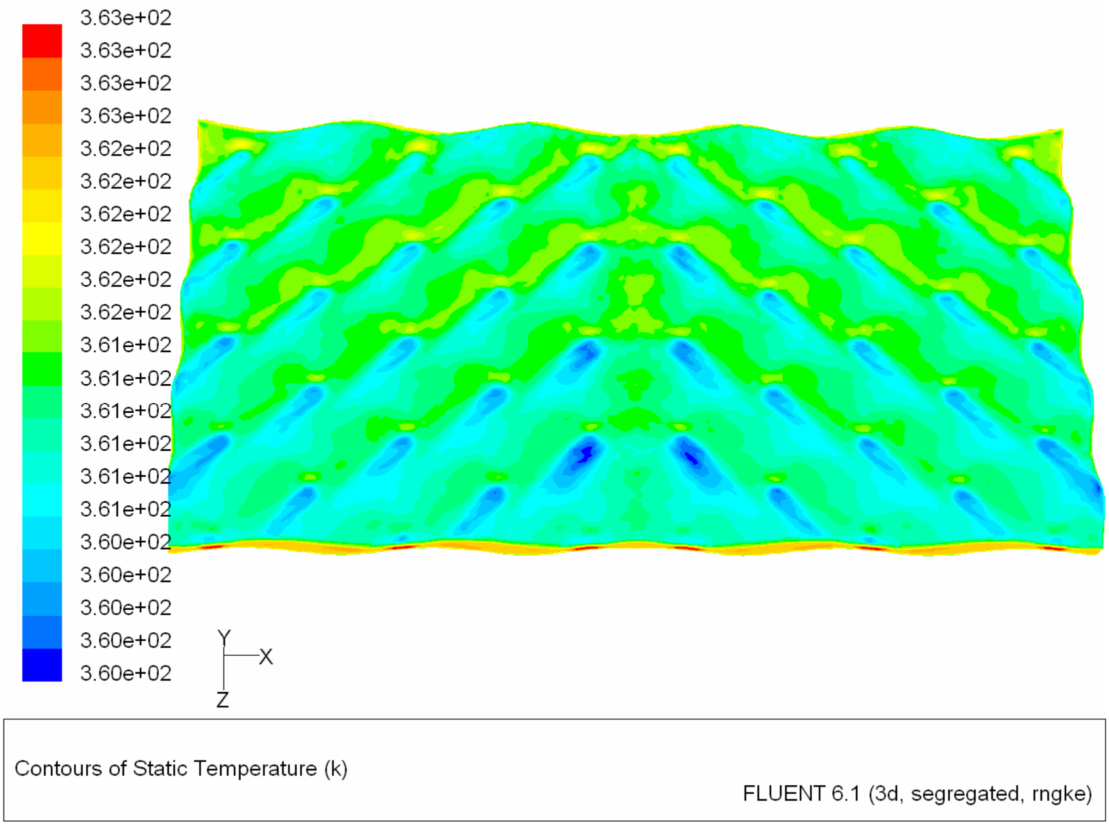


*Figure 3. Temperature field when the heat transfer boundary condition was defined as Convection.*

The results of the heat transfer simulations calculated with different boundary conditions are collected to Figure 4. Values of the experimental heat transfer correlation are also plotted at Figure 4. Calculation of the experimental correlation is based on the industrial plate heat exchanger [8]. The experimental total heat flux was calculated by using total heat transfer coefficient (Eq. 8) and temperature difference in the studied plate heat exchanger. Nusselt number for the average heat transfer coefficients of water (Eq. 9) and process fluid were calculated from Marriot's [9] Nusselt number correlation for plate heat exchangers (Eq. 10). For the outer fluid, which is the process fluid, average values were estimated from process data and settled as a constant both in the correlation and in the simulations where needed.

$$q_{tot} = \frac{Q}{A} = U_{tot} \Delta T_{lm} = \left( \frac{1}{\frac{1}{h_w} + \frac{1}{h_l} + \frac{\delta}{k}} \right) \Delta T_{lm} \tag{8}$$

$$h = \frac{Nu \cdot k}{D_h} \tag{9}$$

$$Nu = 0.28 Re^{0.56} Pr^{0.4} \tag{10}$$

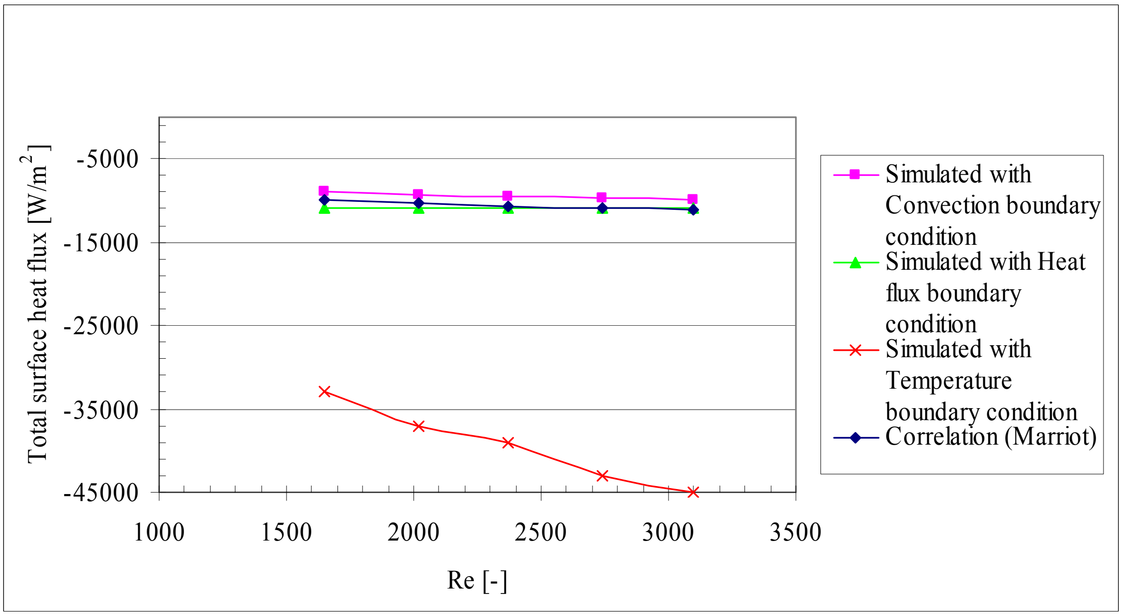


*Figure 4. Comparison of the total heat flux of the Marriot's correlation and the CFD simulations with Convection, Heat flux and Constant wall temperature boundary conditions as a function of Reynolds numbers.*

It can be seen from Figure 4 that the heat transfer model calculated by Convection boundary condition only slightly underestimates the heat flux.The error is not very large, only about 10 %. Simulations calculated by Constant wall temperature boundary condition overestimates the heat flux a lot. This is probably due to assumption of constant temperature of the outer fluid. In reality the temperature changes when the heat transfers to the other side of the plate. With the Heat flux boundary condition the simulated heat flux is constant as expected, because it was defined to the program. In this case this boundary condition is thus not feasible. In conclusion, the Convection boundary condition gives the most realistic model for heat transfer in the corrugated plate heat exchanger.

### *Verification of the heat transfer model*

The heat transfer model was verified by comparing experimental data [2] and simulations. The experimental data was achieved at the inlet temperature of 300 K where as in this study the inlet temperature was 363 K. In order to neglect the influence of temperature $Nu/Pr^{0.33}$ was plotted as a function of Reynolds number (Figure 5). The Nusselt number was calculated by using the correlation of Zettler [2] $Nu = 0.38Re^{0.65}Pr^{0.33}$.

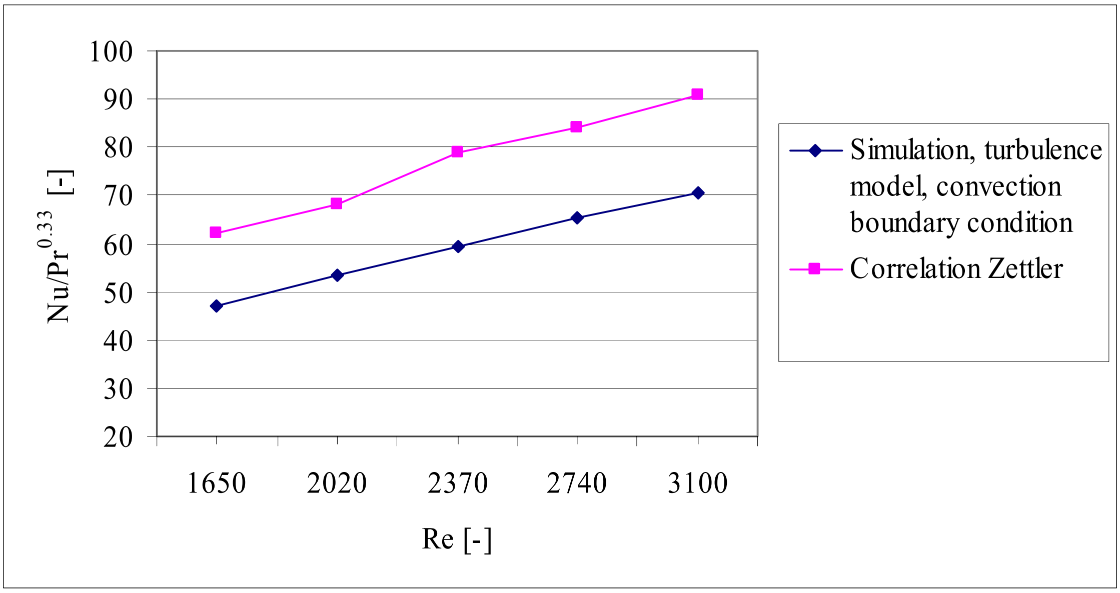


*Figure 5. Comparison of CFD simulated and experimental $Nu/Pr^{0.33}$ in the function of Reynolds numbers.*

From Figure 5 it can be seen that the simulations under predict the results when compared to the experimental correlation. However, the difference is only about 20 %. Error may derive from the different structural material, which furthermore changes the value of Prandtl number. Consequently the heat transfer model with the Convection boundary condition seems to be quite acceptable.

## CONCLUSIONS

Deficiencies were found out in all three studied heat transfer boundary conditions. To model the heat transfer with CFD in the plate heat exchanger is problematic because of the assumptions that have to be made when defining the boundary conditions. Because of the very complex geometry, the values of those parameters are function of the space and can not be defined unambiguously. However, the Convection boundary condition describes most reliably the physical situation of heat transfer in the studied geometry.

## NOMENCLATURE

| | |
|---|---|
| $F$ | force effecting on the system, for example gravity |
| $h_f$ | heat transfer coefficient of fluid [W/m$^2$·K] |
| $h_w$ | heat transfer coefficient of water [W/m$^2$·K] |
| $h_f$ | heat transfer coefficient of process fluid [W/m$^2$·K] |
| $J_j$ | diffusion flux of component $j$ |
| $k_{eff}$ | effective conductivity |
| $k$ | heat conductivity [W/m·K] |
| $p$ | pressure [Pa] |
| $q$ | heat flux [W/m] |
| $q_{rad}$ | heat flux of radiation [W/m] |
| $S_h$ | energy source |
| $S_m$ | mass source |
| $t$ | time [s] |
| $T$ | temperature [K] |
| $T_f$ | local fluid temperature [K] |
| $T_w$ | temperature of wall [K] |
| $u$ | flow velocity [m/s] |

### *Greek letters*

| | |
|---|---|
| $\delta$ | thickness of the plate [m] |
| $\rho$ | density [kg/m] |
| $\overline{\overline{\tau}}$ | shear stress tensor |
| $\mu$ | viscosity [kg/m·s] |

### *Subscript*

| | |
|---|---|
| *eff* | effective |
| *ext* | external |
| *f* | fluid |
| *j* | component |
| *h* | energy |
| *lm* | logarithmic mean |
| *m* | mass |
| *rad* | radiation |
| *w* | wall |

## REFERENCES:


[1] Heggs P J, Hallam, R A Walton C (1997) Local transfer coefficient in corrugated plate heat exchanger channels. IChemE, 75, Part A, p. 641–645.

[2] Zettler H U & Müller-Steinhagen H (2001) The use of CFD for the interpretation of fouling data PHEs. The 4th International Conference on Heat Exchanger Fouling. Fundamental approaches & technical solutions. 8.–13.7.2001, Davos, Switzerland

[3] Mehrabian M A & Poulter R (2000) Hydrodynamics and thermal characteristics of corrugated channels: computational approach. Applied Mathematical Modelling, 24, p. 343–364.

[4] Ciofalo M, Stasiek J & Collins M W (1996) Investigation of flow and heat transfer in corrugated passages - II. Numerical simulations. International Journal of Heat and Mass transfer, 39, (1), p. 165–192.

[5] Martin H (1996) A theoretical approach to predict the performance of chevron-type plate heat exchangers. Chemical engineering and processing, 35, (4), p. 301–310.

[6] Anon. (2003) Fluent 6.1 User's guide. Lebanon, Fluent Inc.

[7] Riihimäki M, Muurinen E, Keiski RL (2004) Thermal analysis of a plate heat exchanger in fouling conditions – Analysis of process information and calculations based on design equations. 16th International Congress of Chemical and Process Engineering, 22.–26.8.2004, Prague, Czech Republic.

[8] Pääkkönen T M (2005) Modelling of heat transfer and fluid flow at plate heat exchanger. Diploma thesis, University of Oulu, Department of process and environmental engineering, 134 p.

[9] Marriott (1971) Where and how to use the plate heat exchanger. Chemical engineering, 78, April 5, p. 127–134.