EPJ Web of Conferences 114, 02074 (2016)
DOI: 10.1051/epjconf/201611402074

Optimization principle of operating parameters of heat exchanger by using CFD simulation

Jozef Mičieta[1,a], Vondál Jiří[2], Jozef Jandačka[1], Richard Lenhard[1]

[1]University of Žilina, Faculty of Mechanical Engineering, Department of Power Engineering, Univerzitná 1, 010 26 Žilina, Slovakia
[2]Brno University of Technology, Institute of Process and Environmental Engineering, Technická 2896/2, 616 69 Brno, Czech Republic

Abstract. Design of effective heat transfer devices and minimizing costs are desired sections in industry and they are important for both engineers and users due to the wide-scale use of heat exchangers. Traditional approach to design is based on iterative process in which is gradually changed design parameters, until a satisfactory solution is achieved. The design process of the heat exchanger is very dependent on the experience of the engineer, thereby the use of computational software is a major advantage in view of time. Determination of operating parameters of the heat exchanger and the subsequent estimation of operating costs have a major impact on the expected profitability of the device. There are on the one hand the material and production costs, which are immediately reflected in the cost of device. But on the other hand, there are somewhat hidden costs in view of economic operation of the heat exchanger. The economic balance of operation significantly affects the technical solution and accompanies the design of the heat exchanger since its inception. Therefore, there is important not underestimate the choice of operating parameters. The article describes an optimization procedure for choice of cost-effective operational parameters for a simple double pipe heat exchanger by using CFD software and the subsequent proposal to modify its design for more economical operation.

1 Introduction

Heat exchangers belong to the most important elements of energy facilities and they are used in a wide field of industry [1]. They are devices for heat transfer among heat carrying fluids. They can be classified according to numerous criteria e.g. according to way of use (heaters, coolers, evaporators, condensers, etc.), design arrangement (heat transfer between two or more fluids, etc.), way of the heat exchange (without or with phase change) or according to contact of one and other fluid (mixer-heat exchanger – it does not have a heat exchange surface and the fluids are mixed together; regenerator – it have one heat exchange surface, which is alternatively flowed around by hot and cold fluid stream and the exchangers use heat accumulation (it is also referred to as direct transfer type); finally there is recuperator, which is referred to as indirect transfer type, because a wall separates the fluid streams) [2].

The technical level of heat exchanger construction significantly adjudicates on the effectiveness and economic return of investments. Minimization of costs are therefore an important aim both for the designers, but mainly for users. Optimal design and operation of the heat exchanger require understanding of heat transfer issues, design and operational requirements. The traditional approach to the design of the heat exchanger is based on an iterative process. It contains gradually changed design parameters until a satisfactory solution for assigned specification is reached [3]. However, these methods, besides that they are time consuming, they do not guarantee optimum economic solution. The calculation process can be supported by specific software and with some constrains also by CFD (Computational Fluid Dynamics) software. The paper shows the optimizing principle of the operating parameters, which is based on the condition of minimal operating cost.

2 Design of heat exchanger

The mathematical description of some occurring processes in the heat exchanger is so complex that there is not possible do it without similarity theory and practical experience [4]. The structural design of the heat exchanger is essentialy dependent on the experience of the engineer and the use of computational software is major advantage from timesaving point of view. Typically, the first there is selected geometry of the device. Subsequently, there are defined value of the design variables according to the specifications and several provided mechanical and thermodynamic parameters in order to obtain a satisfactory heat transfer coefficient. Individual engineer`s options are according to customer requests and depending on the iteration

[a] Corresponding author: jozef.micieta@fstroj.uniza.sk

procedure until an acceptable design of the heat exchanger complies to the specification with satisfactory compromise from efficiency point of view [3].

2.1 Sizing of heat exchanger

Calculating method of heat exchanger depends on case whether new heat exchanger is designed (primary result are design and size of the heat transfer surface) or control calculation (size of heat transfer surface is known and usually there are calculated the output or input temperature, transferred heat flux or other operating parameters). The choice of a suitable type of heat exchanger precedes the sizing of heat exchanger, which is dependent on many parameters [5]. The next paragraphs are focused on the design calculation. Thermal calculation of the heat exchanger has a number of phases:

- Gathering of initial data: kind and physical properties of hot – heating fluid (index 1) and cold – heated fluid (index 2), flow rates and temperatures of both fluids, optionally heat exchanger performance, requirements of the pressure and permissible pressure drop.
- A draft of heat exchanger type, its layout (parallel flow, contraflow, cross-flow) and dimensions.
- Thermal calculation, which is comprised of heat balance of the heat exchanger.
- Technical and economical optimization and specification of the parameters in order the heat exchanger keeps economic requirements.

Among two most common methods for calculation of heat exchanger is included LMTD method (Logarithmic Mean Temperature Difference), which is based on the known inlet and outlet temperatures of both media. However, if some temperature is unknown, there is necessary to interpolate it. The second method, which is based on heat exchanger efficiency for transferring a certain amount of heat (NTU – Number of Transfer Units) is preferable to LMTD method. NTU method is suitable for the control calculation and comparison of various types of heat exchangers [6]. In the paper, there is below, partly mentioned only LMTD method, but NTU method is not described in detail.

2.2 Thermal balance of the heat exchanger

The basic equation for thermal calculating is enthalpy balance of heat exchanger. For illustration, we consider a simple recuperator, concretely the double pipe heat exchanger (Figure 1). For the layout of the heat exchanger in Figure 2 is the conservation of energy for the heat exchanger expressed

$$\sum \dot{H}' = \sum \dot{H}'' + \sum \dot{Q}_{\mathrm{loss}} \quad \text{(W)}, \tag{1}$$

where $\dot{H}$ enthalpy flow, W; $\dot{Q}_{\mathrm{loss}}$ is heat loss to the environment, W; and superscripts $'$ and $''$ mark the inlet and outlet of fluid in the heat exchanger. When the heat loss in common systems with insulation does not exceed 5% [7], it is possible to simplify calculations by neglecting of heat loss or it can be estimated. Furthermore, equation (1) can be itemized to individual enthalpy flows as equation

$$\dot{H}_1' + \dot{H}_2' = \dot{H}_1'' + \dot{H}_2'' + \dot{Q}_{\mathrm{loss}} \quad \text{(W)}, \tag{2}$$

which for two-phase flow of fluid after treatment and expression through $\dot{m}$ mass flow, kg m^{-3}; and h the specific enthalpy, J kg^{-1} has the form

$$\dot{m}_1 \left(h_1' - h_1''\right) = \dot{m}_2 \left(h_2'' - h_2'\right) + \dot{Q}_{\mathrm{loss}} \quad \text{(W)}. \tag{3}$$

If the total heat flux is denoted $\dot{Q}$, W simple modification of equation (3) gives the next expression (4) for the enthalpy balance of one fluid. Its heat flux is given by the enthalpy loss of hotter fluid. The heat flux causes an increasing enthalpy of the heated fluid and increasing enthalpy of environment around the heat exchanger (heat loss to the environment).

$$\dot{m}_1\, p_{c_1} \left(T_1' - T_1''\right) = \dot{m}_2\, p_{c_2} \left(T_2'' - T_2'\right) + \dot{Q}_{\mathrm{loss}} \quad \text{(W)}, \tag{4}$$

where c_{p} specific heat capacity, J kg^{-1} K^{-1} is considered at main temperature of fluid (hotter or colder). For the control calculation, there are three unknown variables: outlet temperatures T_1'', T_2'' and $\dot{Q}$ heat output, W. On the contrary, at design calculations, there are given these temperatures and the heat performance. There is necessary to calculate the heat exchange surface S, m^2 according to the equation for heat transfer calculation

$$\dot{Q} = k\, S\, \Delta T_{\mathrm{mean}} \quad \text{(W)}, \tag{5}$$

where k heat transfer coefficient, W m^{-2} K^{-1} and the heat flux $\dot{Q}$, W can be estimated from enthalpy balance, which is expressed by the previous equation (4). The temperatures of both liquids are during fluid flow through the heat exchanger changed, thereby there is changed also the temperature difference between them. ΔT_{mean} is the mean temperature gradient, which is defined by integrals

$$\Delta T_{\mathrm{mean}} = \frac{1}{S} \int \left(T_1 - T_2\right) \mathrm{d}S \quad \text{(K)}, \tag{6}$$

where the temperature difference $\left(T_1 - T_2\right)$ is the local value at the location. Mean temperature gradient, which is valid for the heat exchanger, must be obtained by integration of equation (6) over the entire surface of the heat exchanger. The result of this integration is the equation for the medium logarithmic temperature difference (it is valid not only for a parallel flow, but also for contraflow), which is marked LMTD and defined by equation

$$\Delta T_{\mathrm{ln}} = \frac{\Delta T'' + \Delta T'}{\ln \dfrac{\Delta T''}{\Delta T'}} \quad \text{(K)}, \tag{7}$$

where $\Delta T'$ and $\Delta T''$ are the extreme temperature differences (gradients) at the inlet and at the outlet of heat exchanger $\left(\Delta T' > \Delta T''\right)$, which mean inlet and outlet temperature differences of the fluid flows. If there is changed the temperature gradient along the heat transfer surface a little, there is possible to calculate an average temperature gradient as the arithmetic average of the two extreme temperature differences (gradients)

$$\Delta T_{\text{mean}} = \frac{1}{2}\left(\Delta T'' + \Delta T'\right) \qquad \text{(K).} \qquad (8)$$

The mean arithmetic gradient is always greater than the mean logarithmic temperature difference. For $\left(\Delta T'/\Delta T''\right) > 0{,}5$ is the deviation less than 4%. In contrast to the pure parallel flow or contraflow, which includes also the case of double pipe heat exchanger, the calculation of the mean temperature gradient in the heat exchangers with the cross-flow or a combination flow of both fluid is difficult. Therefore, for the case of conventional practice, the results are presented graphically or by equations. By use of these parameters

$$P = \frac{T_2'' - T_2'}{T_1' - T_2'} \quad (-); \qquad P = \frac{T_2'' - T_1'}{T_2' - T_2''} \quad (-) \qquad (9)$$

is determined corrective coefficient F. It is used for multiplying ΔT_{ln} for case of the contraflow adjustment. The correction factor F is for the structural layout of the degree of deviation from the maximum possible LMTD value.

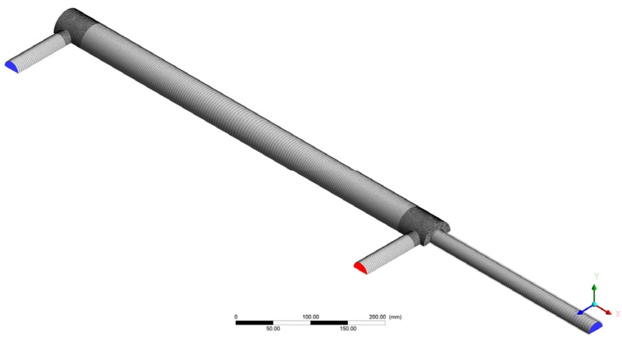

Figure 1. Model of heat exchanger with computational mesh.

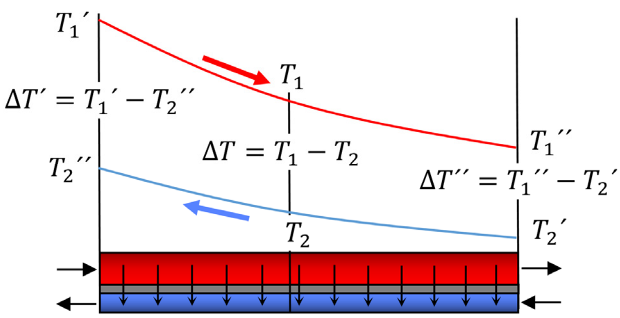

Figure 2. Temperature progress in parallel flow arrangement of heat exchanger.

The reciprocal value of the heat transfer coefficient $1/k$ is the total thermal resistance of the heat transfer from hot to cold fluid, which is expressed by the equation (10). Total thermal resistance R_c is calculated according to serial alignment. Thus, it is sum of the individual thermal resistances, which are caused by heat transfer on the both sides of the wall, the heat resistances of layers of heat transfer wall and the heat resistance of fouling on both sides. The total thermal resistance of wall which is composed from different layers, is solved

$$R_c = \frac{1}{k} = \frac{1}{\alpha_i} + \sum_{j=1}^{n} \frac{\delta}{\lambda_j} + \frac{1}{\alpha_e} \qquad (\text{m}^2\ \text{K W}^{-1}), \qquad (10)$$

where subscript i and e represent the internal and external heat exchange surface. Heat transfer coefficient α, W m^{-2} K^{-1} can be determined by similarity theory in the form of Nusselt number for the design of heat exchanger, the hydrodynamic conditions and the fluid characteristic. The determination of heat transfer coefficient is quite large task, but there is enough literature sources for individual assessment of different cases.

From the economy view, for design calculation, there is important the determination of the parameters that directly affect investments and operating costs. Investments depend on the size of the heat transfer surface and on the complexity of production (type) of the heat exchanger, which is reflected in the price for m^2 heat transfer surface. Operating costs include energy, maintenance and repair services. The increasing velocity of hot or cold fluid causes that the heat transfer coefficient α increases, too. Besides, it also allows to decrease investment costs due to the size reduction of the heat transfer surface, but simultaneously operating costs rise significantly, due to increased pressure loss. In design calculations, there have to be determinate the flow velocity (flow rate) so that the total costs, which can be expressed as the sum of investment and operation, were minimal.

2.3 Hydraulic calculation

In order to sizing of devices for circulation of working fluid, there is necessary the calculation of the flow resistance (pressure loss) in the heat exchanger. The total pressure drop of the fluid flow Δp_c is equal to the sum of friction losses Δp_t and the loss of local resistance Δp_ζ.

$$\Delta p_c = \Delta p_t + \Delta p_\zeta = \left(\lambda \frac{L}{d} + \sum \zeta\right) \frac{v^2}{2} \rho \qquad \text{(Pa)} \qquad (11)$$

There are λ coefficient of friction, –, depending on the Reynolds number and the wall roughness; L the considered length of channel through which fluid flows, m; d equivalent diameter (cross section of the channel), m; ξ shape coefficient of resistance, –, which primarily depends on the shape and less on the character of flow (Reynolds number); v mean velocity of flow, m s^{-1}; and ρ density of the fluid flow, kg m^{-3} [8]. P_e total demand for electric power of the circulating pump, W is determined by equation

$$P_e = \Delta p_c \dot{V} = \Delta p_{c_1} \dot{V}_1 + \Delta p_{c_2} \dot{V}_2 = \Delta p_{c_1} \frac{\dot{m}_1}{\rho_1} + \Delta p_{c_2} \frac{\dot{m}_2}{\rho_2}. \qquad (12)$$

3 Task of the heat exchanger calculation

Operating efficiency of heat exchanger is established on the electricity costs associated with forced circulation of heat transfer liquid and on assumption of known incomes from distribution of heat. Capital costs are neglected in this case, because there was considered a heat exchanger in operation and it could be affected only its operation setting. There was necessary to establish flows of both media so that the operation of the device was as economically as possible.

For heat exchanger was set the cold (heated) water with temperature of 30 °C flows to the straight tube and through the pipe interspace flowed the hot (heating) water with inlet temperature of 110 °C. Input parameters for the boundary conditions were flows of heat transfer fluids or amended geometry of the heat exchanger. The model of flow and heat transfer in the heat exchanger`s geometry was simulated in a commercial CFD software Ansys Fluent 15. For evaluation of the parameter`s impact was changed only one parameter while others remained constant. The main assessment detail from CFD analyses was the performance of heat exchanger, which was expressed by the ratio between incomes from the heat distribution and operating costs for circulation of heat transfer fluid. In the calculation were considered the cost of electricity 150 € MWh^{-1} [9] and income from heat distribution 100 € MWh^{-1} (approximately 27.80 € GJ^{-1}) from heat distribution [10, 11, 12].

3.1 The model of the heat exchanger and the principle of optimization

Double pipe heat exchangers are often used in special operating conditions such as low flow volume, for large temperature differences, at desired residence time, for fast temperature changes or at high pressure operation and if there is a requirement for pure parallel flow or counterflow [2]. The heat exchanger consists of simple concentric arrangement of the two pipes with different diameter. One of the fluid flows in the inner tube and the second into the created annulus. In this model case, there was not assumed a phase change of heat transfer fluid (water). Double pipe heat exchanger is one of the simplest, therefore it was selected for example. The optimization process was focused on the adjustment of cost-effective flow rate of heat transfer fluids for given dimensions of the heat exchanger and the inlet temperatures of the both streams.

The goal of paper is not to describe the settings of the model in detail because it is a very large task. Therefore, the description of the model is mentioned only briefly. Since the model heat exchanger was symmetric, there was appropriate to model only half of the heat exchanger in order to speed up the calculation. From an actual geometry could be removed insignificant volumes as heat exchanger walls, there was left only partition - heat exchanger surface and volumes of liquids. The heat transfer thorough the outer walls was included in the model. But on other hand the outer wall was not physically modelled, because temperature field in the direction of the wall thickness was not significant. Simplified computational domain was then filled by mesh, which was near the walls and in areas with expected intensive flow smoother, but on other hand, in the direction, in which there were not expected substantial changes of flow were mesh cells more stretched. Finally, this approach contributes to more accurate and less time consuming calculation. Therefore, the mesh quality was acceptable. For solution, there was implemented the Realizable k-epsilon turbulent model with Enhance wall treatment that describes the flow near the walls of heat exchanger [13]. Boundary condition on inlet was defined by mass flow rate in normal direction to boundary. Outlet boundary condition was pressure-outlet type. Calculation also includes heat loss to the environment and for roughness was assumed that height of roughness peaks does not exceed the hydrodynamic boundary layer. For chosen, above-mentioned boundary conditions at inlet were specified input values and the calculation was made subsequently. When tasks were converged, the results were plotted in temperature, speed and pressure profiles. The flow was drawn by velocity vectors and streamlines. Subsequently, the required data were shown in a table and exported for further analysis of costs in the spreadsheet program Excel. Electricity costs did not include efficiency of pumps, since there were other parameters more or less estimated. The results of this process were graphs of various analysed parameters. In the graphic data (Figure 5, Figure 6), there was found the maximum of efficiency curve which shows the optimum flow.

3.2 Data evaluation

For determination of mentioned variables were decisive data about pressure loss for calculation of consumed energy and heat fluxes in the heat exchanger for determination of the heat output. In order to determine the most economically advantageous operating parameters (flow) of fluid was sought global maximum of reference indicator (efficiency - the ratio of energy prices) by use of regression analysis. Several values, which were calculated by numerical calculation in CFD software, were for properly selected input parameters approximated by a polynomial function [14]. The emphasis in approximation of values was given on the accuracy of the maximum values. Greater variation in the limit values of the reference interval was not important then, because the approximation function was decisive for determination of the maximum. The approximation function in the limit values describes mostly the trend line.

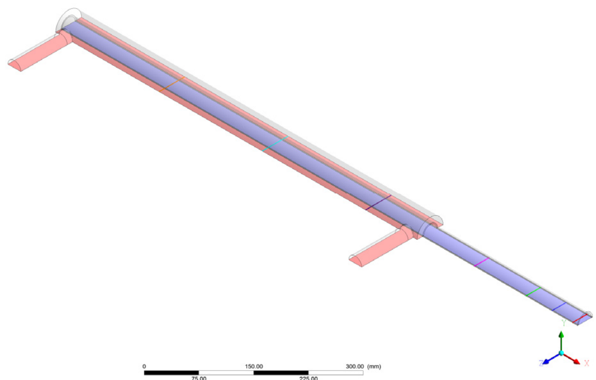

Figure 3. Lines for analysed velocity profiles.

Figure 3 shows lines in given distance from inlet, where were drawn velocity profiles. They are shown in next Figure 4.

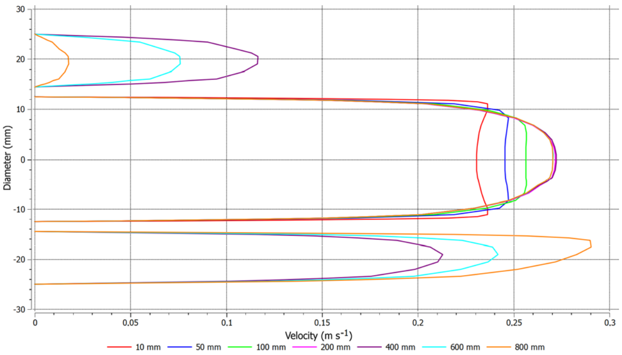

Figure 4. Velocity profiles (m s^{-1}) in symmetry plane.

From shape of velocity profiles is visible their tendency to form the parabolic profile, since in distance of about 8 times the pipe diameter leads to stable profile.

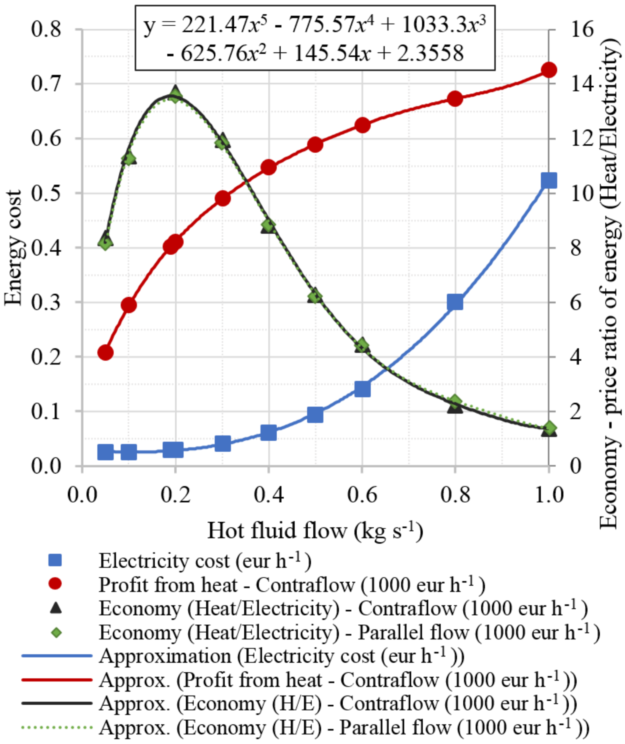

Figure 5. Chart for determination of flow rate (kg s^{-1}) of heating (hot) fluid.

Subsequently, when the flow rate of one fluid was specified (Figure 5), there was implemented the same analysis for the heated fluid in the inner pipe. For this case of the heated fluid, there was set optimum flow from previous calculation (Figure 6). For verification of the calculated optimum flows were CFD simulations realized for these values again.

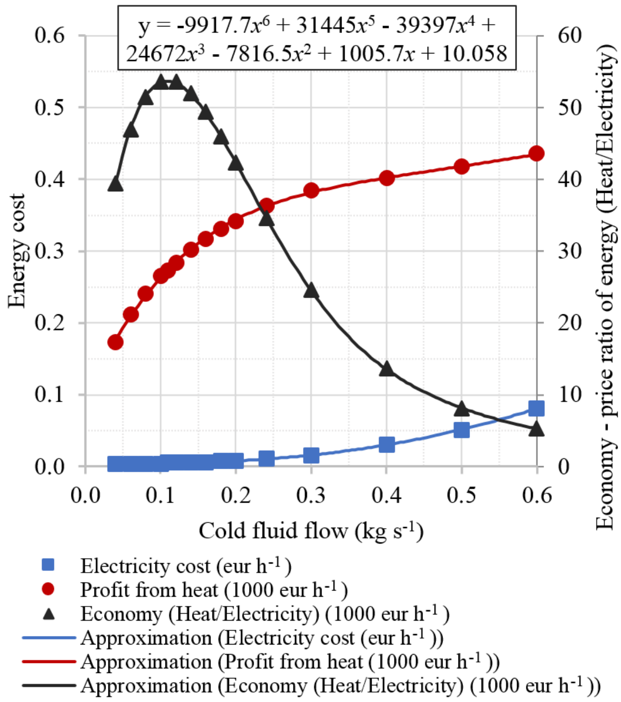

Figure 6. Chart for determination of flow rate (kg s^{-1}) of heated (cold) fluid in parallel flow adjustment.

The results in the case of parallel flow adjustment were almost identical to contraflow adjustment, but the efficiency of the contraflow adjustment should be significantly better. The reason for this discrepancy was very little difference of output temperatures in parallel flow and contraflow adjustment in Figure 7 and Figure 8.

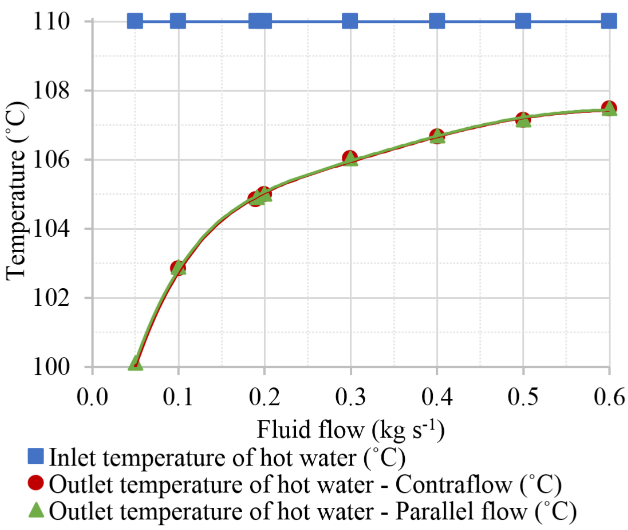

Figure 7. Inlet and outlet temperatures (°C) of heating (hot) fluid in both cases of adjustment.

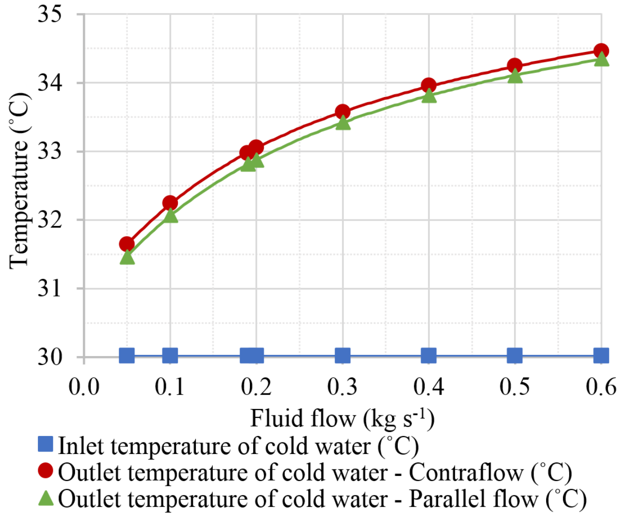

Figure 8. Inlet and outlet temperatures (°C) of heated (cold) fluid in both cases of arrangement.

Figure 9 and Figure 10 display the temperature profiles in the symmetry plane, respectively in the cross-section, approximately in the middle of the heat exchanger.

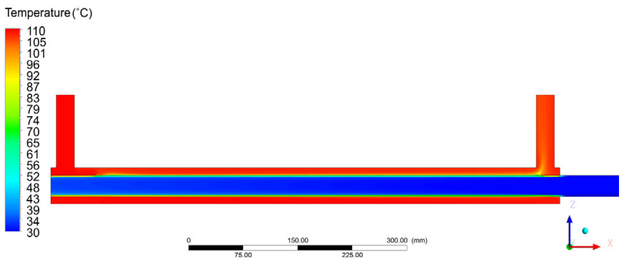

Figure 9. Temperature field (°C) in the cross-section – 24 ribs in the symmetry plane.

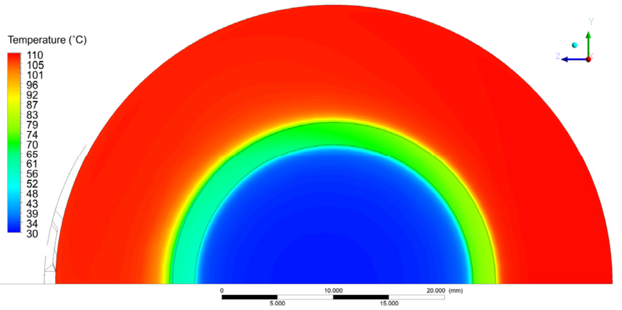

Figure 10. Temperature field (°C) in the cross-section, approximately in the middle of the heat exchanger.

The heat transfer can be increased by adding a suitable number of ribs (Figure 11, Figure 12). However, in this respect, there are structural impacts and costs of materials and manufacturing becoming more significant.

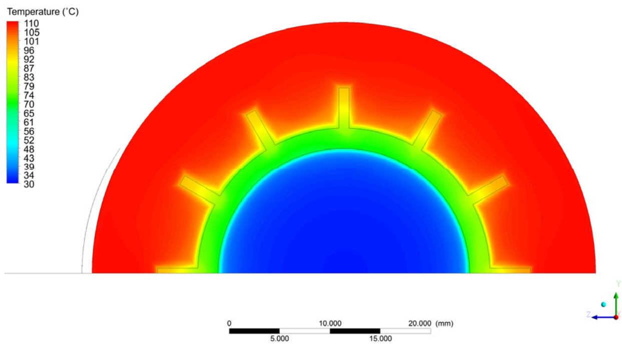

Figure 11. Temperature field (°C) in the cross-section – 12 fins.

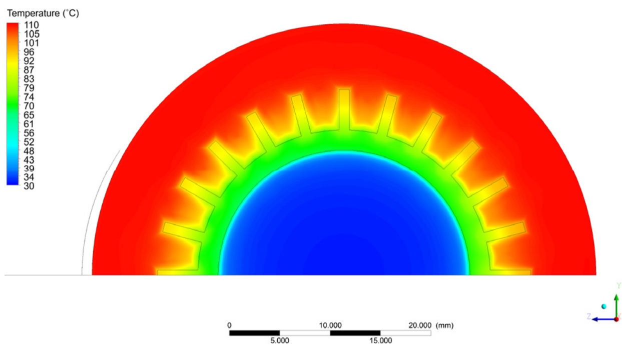

Figure 12. Temperature field (°C) in the cross-section – 24 fins.

There was achieved more guided flow by addition of fins in the longitudinal direction, as illustrate velocity profiles (Figure 13, Figure 14).

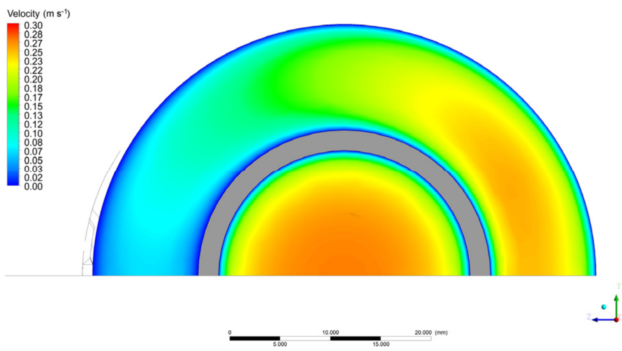

Figure 13. Velocity field (m s^{-1}) in the cross-section – without fins.

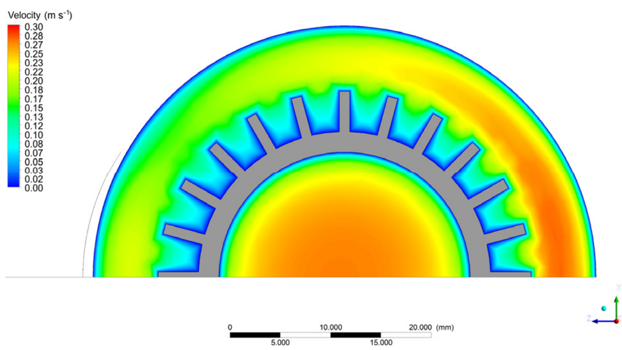

Figure 14. Velocity field (m s^{-1}) in the cross-section – 24 fins.

4 Conclusions

The calculation was unable point out more pronounced difference between parallel flow and contraflow arrangement of flows. Mentioned output temperatures showed that the contraflow arrangement is only a little advantageous as the parallel flow. The optimal flow rate of heated fluid was determined, for both flow arrangement, by a maximum of polynomial functions, which are specified by approximation of calculated values. For parallel flow arrangement was determined optimum flow rate approximately 0.191 kg s^{-1} and for the contraflow adjustment was optimal flow 0.190 kg s^{-1}, which are almost the same values. The determined flow rate in annulus space, in case contraflow arrangement of the heat exchanger, was eked by the optimized flow in a straight (inner) pipe (0.109 kg s^{-1}). In the above mentioned processes was determined the best fluid flow from the economic point of view. For a comprehensive analysis should be taken into account also other operating parameters in the calculation. Similarly, it would be appropriate to assess adding fins, which were proposed for the intensification of heat transfer. Then there was necessary to assess besides desired height, width and number of fins also costs of material and manufacture.

Acknowledgments

Moderné vzdelávanie pre vedomostnú spoločnosť/Projekt je spolufinancovaný zo zdrojov EÚ

Podpora kvality vzdelávania a rozvoj ľudských zdrojov v oblasti technického výskumu a vývoja v priestore modernej vedomostnej spoločnosti ITMS 26110230117

The research is supported by European regional development fund and Slovak state budget by the project Research centre of University of Žilina“, ITMS 26220220183.

References

1. G. V. S. Rao, C. J. Rao, N. H. Babu, International J. of Research in Aero. and Mech. **2**, 11 - 16 (2014)
2. K. Jelemenský, iDB Journal **2**, 34 - 36 (2011)
3. A. C. Caputo, M. P. Pacifico, P. Salini, Applied Thermal Engineering **28**, 1151 - 1159 (2008)
4. Z. Ibler, J. Karták, J. Mertlová, *Technický průvodce energetika*, 61 (2002)
5. K. Jelemenský, iDB Journal **3**, 38 - 40 (2011)
6. M. Baláš, *Kotle a výměníky tepla*, 119 (2013)
7. P. Stehlík, J. Kohoutek, J. Němčanský, *Tepelné pochody: výpočet výměníku tepla*, 129 (1991)
8. M. Székyová, K. Fernstl, R. Nový, *Vetranie a klimatizácia*, 422 (2004)
9. Energy online, a. s., www.cenaenergie.sk (2015)
10. I. Buranský, www.energiaweb.sk (2014)
11. P. Laube, energostat.cz (2014)
12. Energostat, www.energostat.cz (2013)
13. J. Hužvár, P. Nemec, Proc. of the Optim. Using Ex.-based Meth. and Comp. Fluid*,* 219 - 224 (2009)
14. A. Kapjor, J. Hužvár, B. Ftorek, M. Vantúch, Communications **16**, 54 - 61 (2014)