www.czasopisma.pan.pl www.journals.pan.pl
Chemical and Process Engineering 2017, 38 (4), 567-576
DOI: 10.1515/cpe-2017-0045

MODIFICATION OF A RECUPERATOR CONSTRUCTION WITH CFD METHODS

Wojciech Ludwig*[1], Daniel Zając[2]

[1]Wroclaw University of Science and Technology, Faculty of Chemistry, Department of Chemical Engineering, Norwida 4/6, 50-373 Wroclaw, Poland

[2]Opole University of Technology, Faculty of Mechanical Engineering, Department of Environmental Engineering, Mikołajczyka 5, 45-271 Opole

The purpose of the work was initial modification of the construction of a commercially produced heat exchanger – recuperator with CFD (computational fluid dynamics) methods, based on designs and process parameters which were provided. Uniformity of gas distribution in the space between the tubes of the apparatus as well as the pressure drop in it were taken as modification criteria. Uniformity of the gas velocity field between the tubes of the heat exchanger should cause equalization of the local individual heat transfer coefficient values and temperature value. Changes of the apparatus construction which do not worsen work conditions of the equipment, but cause savings of constructional materials (elimination or shortening some parts of the apparatus) were taken into consideration.

Keywords: recuperator, modification, CFD, gas flow

1. INTRODUCTION

Finned tube heat exchangers are devices being very commonly used in chemical and power industry, as well as in a number of technological processes. Most often, they are part of another device designed for carrying out required process (Yang et al., 2015). When designing a heat exchanger, it is necessary to calculate the flow resistance of the fluid flowing through the device (Pal et al., 2016). Values of the resistances affect proper selection of the required pressure level for pumping the medium through the exchanger and thus, the selection of pumping device. Application of increased flow velocities intensifies the heat exchange process; the convective transfer coefficient is being increased, which allows reduction of the device's dimensions, thereby reducing the capital costs. However, the increase of velocity means increased resistance to flow, which leads to higher consumption of energy ensuring adequate pressure, and this can even imply the need for buying bigger pumping device, which generates higher operating and capital costs. The problem of calculating flow resistance in inter-pipe space is sophisticated, due to the complex nature of the flow (Wen et al., 2015). Besides the above-mentioned, ensuring uniform flow of the medium in the tube bundle is an extremely important task from the perspective of utilizing the widest possible heat exchange surface area, and prolonging the lifetime of pipes. In case of improper flow distribution of the working fluid stream, some of the pipes in bundle can be insufficiently surrounded with the fluid flowing at uniform velocity, which reduces the overall efficiency of the heat exchanger. The second problem of irregular fluid flow distribution is formation of dead zones, and this, in case of high fluid temperatures, can increase the risk of thermal damage to individual pipes (Goodarzi and Nouri, 2016).

*Corresponding authors, e-mail: wojciech.ludwig@pwr.edu.pl

Recuperators are the types of heat exchangers being frequently used as an element of heat recovery systems in a variety of technological processes. Location for this type of devices and the available space for mounting them are most often limited by the specific process requirements. It is extremely important that the described exchangers will be possibly characterized by small dimensions and low weight. These characteristics enable installing them in locations inaccessible for traditional units, that often have oversized constructions. Examples of such difficult locations may be ceilings with a small load capacity, some areas of chimneys or even gas transport channels (Gil et al., 2015).

After the review of the current state of knowledge on flow dynamics in heat exchangers, it can be concluded that no work on inlet and outlet unit design optimization deals with the subjects of minimizing flow resistances and distribution of the fluid in particular rows in a tube bundle. There is a lack of research works dealing with inlet and outlet shape optimization in heat exchangers and that is the reason why it is necessary to carry out numerical experiments. One should perform experimental verification of results obtained from numerical calculations in order to optimize the shape of the exchanger parts, so as to reach the assumptions made in the research work.

The purpose of the study was to develop an initial concept of recuperator design modification. The conceptual works were focused on determining new geometrical parameters to ensure uniform gas flow in the inter-pipe space or to reduce the size and weight of the applied device. Computational fluid dynamics (CFD) was used during research, which at current state of computer development is a very useful alternative to classical methods of optimization and design, especially for one phase flows (Jaworski, 2005; Chang et al., 2015).

2. FLOW AND DESIGN ASSUMPTIONS FOR CFD ANALYSIS

The data base for CFD calculations were: a design of the device being manufactured (Fig. 1) and limit values of gas operating parameters in the inter-pipe space, which were supplied by the producer: temperature: 500 – 100 ^{o}C, velocity 2 - 6 m/s, overall pressure drop 100 - 500 Pa. Air was the medium that flowed through the apparatus.

Because the modeling of heat transfer in the test device was not envisaged, a number of simplifications were made in the geometry of the CFD model. The spaces inside the tubes were omitted, as well as their gas inflow areas. Such fastening elements as flanges, screws, etc. were not included. The calculation domain has been limited to the main body of the device with a short section of the inlet and outlet pipes (Fig. 2).

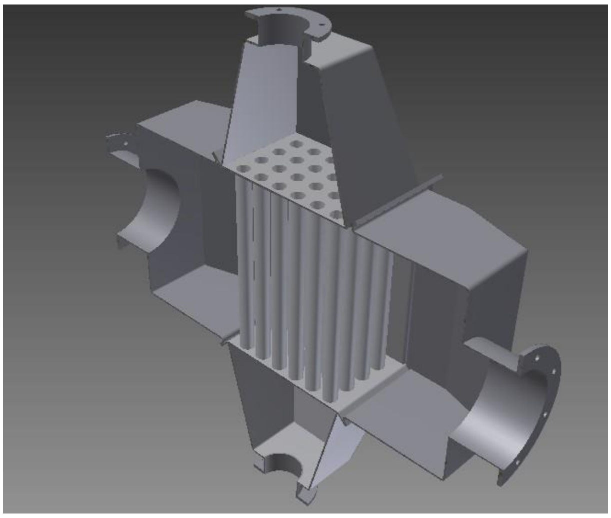

Fig. 1. Pictorial drawing of the produced recuperator design

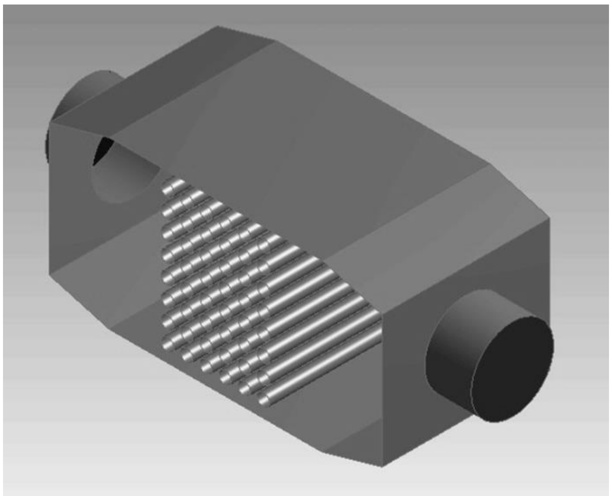

Fig. 2. The simplified geometry of the recuperator used in the CFD model

The research began with tests on numerical meshes of different densities and cell distributions. Finally, a hybrid mesh, mostly with tetrahedral cells was applied inside the device. Moreover, in order to maintain proper density and adequate values of the wall functions in the boundary zone, a high density five-layer mesh with hexahedral cells was introduced (Fig. 3).

A test for checking solution independence on mesh density was carried out. For the mesh with three times greater density (three times more cells), the values of volume integrals of velocity, pressure, k and ε changed on average by about 10%, which was considered as a satisfactory value for quality calculations.

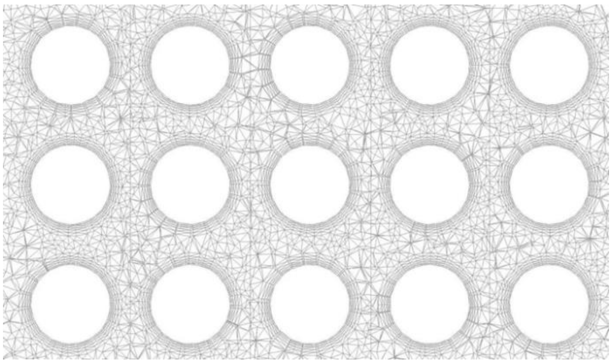

Fig. 3. Part of longitudinal section of numerical mesh in the pipe area used in standard recuperator

Table 1. Basic parameters of the numerical mesh used in calculations of the standard device

Parameter of the mesh	Value
Number of cells	2259620
Number of nodes	787718
Minimum volume of the cell	1.723365×10^{-9}
Maximum volume of the cell	8.623437×10^{-7}
Maximum skewness	0.95

Based on the values of the Reynolds number, it was found, that the turbulent flow was present in the device, for the description of which standard k-ε model was used. The initial stage of the research study also covered the optimization for solver settings (Ansys Fluent 15). Finally, a set of values was gained

(Table 2). It allowed obtaining a stable solution at the lowest number of iterations (1000). Calculations were continued until scaled residuals reached the values below 10^{-4} and the volume integrals of pressure, velocity, k and ε were constant. The impact of temperature on the hydrodynamics of flow was taken into account by changing physical properties of the air, constant in the entire volume of the devices. The air density and viscosity for a given temperature were computed from the ideal gas and Sutherland equations, respectively.

Table 2. Solver parameter constants in all simulations

Simulation parameter	Value
Solver	3D, pressure based
Turbulence model	k-ε standard constants
Boundary layer description	standard wall function
Boundary condition at the inlet	velocity inlet
Boundary condition at the outlet	pressure outlet
Boundary condition for walls	wall, no slip, no heat exchange
Relaxation coefficients	all 0.1
Discretization	second order upwind

The velocity at the inlet of the device was set in such a way that the mean velocity calculated for the empty inter-pipe space was in the range listed in hydrodynamics specifications (2 - 6 m/s).

3. RESULTS IN THE STANDARD APPARATUS

At the beginning of the analysis of the standard equipment work a number of simulations were done for changing temperature of gas in the device (100 – 500 °C). Velocity and pressure fields were obtained for extreme but possible gas velocities in the space between the tubes (2 - 6 m/s). In the entire range of the air flow velocity, the pressure drop was below the permissible value (500 Pa) (Fig. 4), but non-uniform flow of gas around the tubes of the heat exchanger occurred (Figs. 5 and 6).

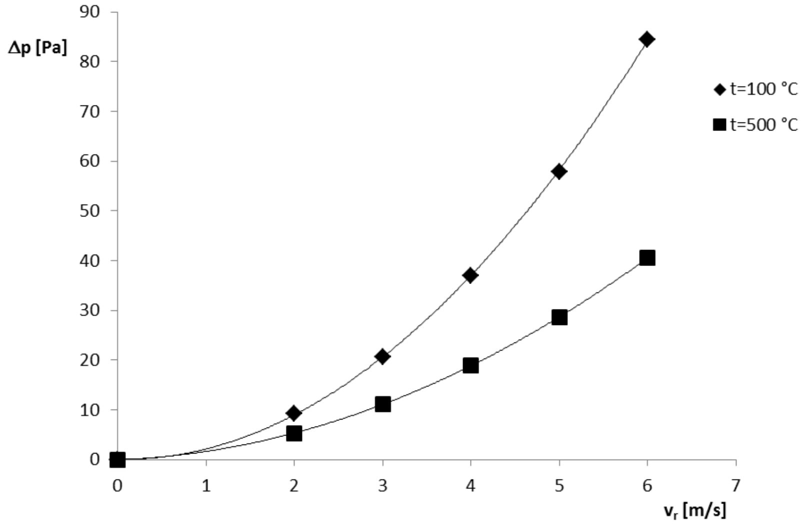

Fig. 4. Pressure drop in the standard device as a function of the average velocity in inter-pipe space

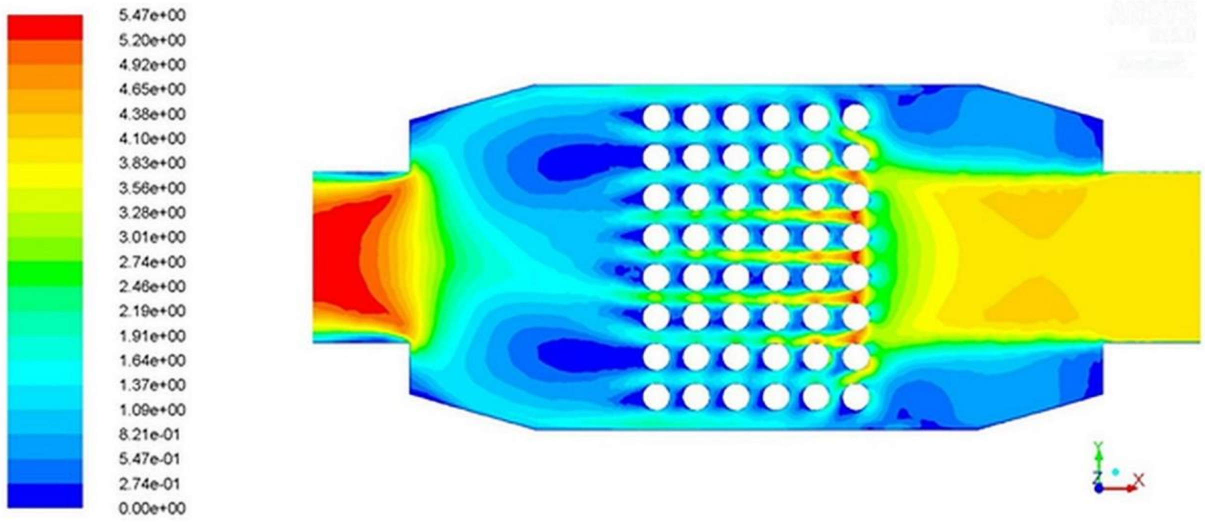

Fig. 5. Air velocity (in m/s) contours inside the longitudinal section of the standard apparatus (t = 100°C, v_r = 2 m/s, inlet on the right)

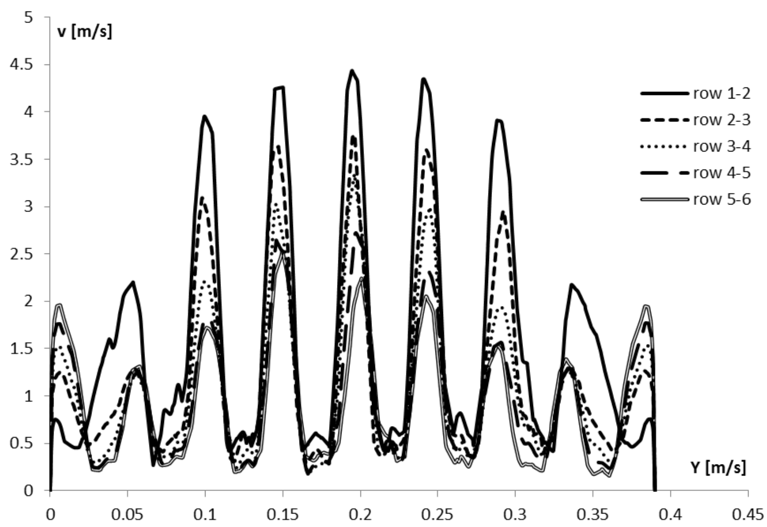

Fig. 6. Gas flow velocity in the cross section of the produced recuperator between different pipe rows in the heat exchanger (v_r = 2 m/s, t = 100 °C)

Inter-pipe flow velocities are the highest in rows opposite the inlet pipe (Fig. 5). At these areas, velocities are often above the assumed maximum values (in front rows, they are more than doubled) (Fig. 6). The greater the distance from the inlet tube, the lower the air velocity in a given row. This is caused by poor inlet air stream flow distribution, which does not cover the entire inter-pipe cross-section. As the distance from the inlet of the device in subsequent inter-pipe gaps in the exchanger is increasing, the gas velocity field is becoming more uniform. The gas passes from rows opposite the inlet tube to rows located closer to the solid walls of the device (Figs. 5 - 6).

The examined device is characterized by irregular gas flow around pipes of the heat exchanger. Air stream flowing from the narrow inlet pipe into the wide chamber of the equipment has not got space to increase its cross section. Almost the whole of air flows through the width of exchanger comparable to the diameter of the inlet pipe. Therefore, in further sections of the research study several design approaches are suggested to eliminate or to reduce the impact of the abovementioned adverse phenomena.

4. HEAT EXCHANGER MODIFICATION PROPOSITIONS AND CFD ANALYSIS

Based on the performed CFD analysis, work was undertaken for formulating modification proposals for the recuperator: the change of inlet and outlet pipe diameter, decreased length of inlet and outlet section, systems of baffles extending the inlet air stream on all rows of pipes. Among all of the tested designs, five were selected, which are characterized either by improvement in gas flow parameters (more homogenous inter-pipe air flow in the heat exchanger) or by causing weight reduction of the units without flow quality-decrease, when compared to the standard device.

Table 3. Comparison of flow parameters for different modification propositions for recuperator designs (v_r = 6 m/s, t = 100 °C, average velocity for profile between the 1st and 2nd row of pipes)

No.	Short description	Δp [Pa]	w [m/s]	Figure
	Standard device d_r = 200 mm	84	4.9	5
1	Best solution, considering pressure drop and uniform flow throughout the device with extended diameter of inlet pipe of d_r = 300	33	2.9	7
2	Size reduction of the unit's body (without the connecting pipes and flanges) from 782 mm to 582 mm by removing the element with fixed cross-section area	88	5.4	8
3	Size reduction of the unit's body (without the connecting pipes and flanges) from 782 mm to 500 mm by removing the element with variable cross-section area	90	5.1	9
4	Adding flow guide in perpendicular direction to pipes of the exchanger	89	4.4	10
5	Adding two flow guides in perpendicular and parallel direction to pipes of the exchanger	106	4.1	11

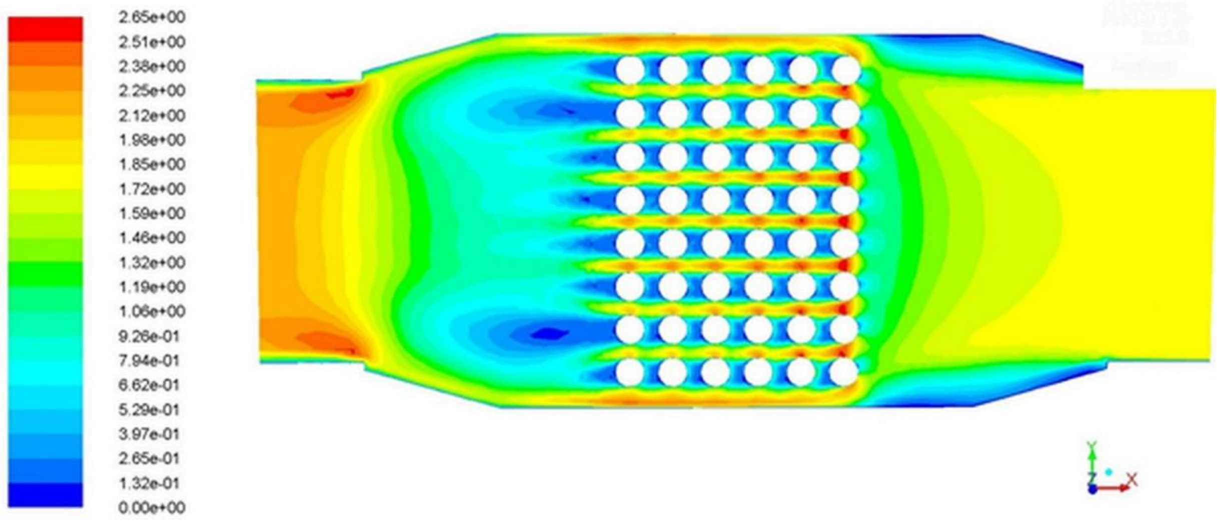

Fig. 7. Air velocity (in m/s) contours inside the longitudinal section of the modified apparatus, proposition 1, (d_r = 300 mm, t = 100 °C, v_r = 2 m/s, inlet on the right)

After increasing the outer diameter of the inlet pipe from 200 to 300 mm (proposition 1, Table 3) contours of the gas flow are becoming harmonized already in the first inter-pipe gaps (Figs. 7 and 12).

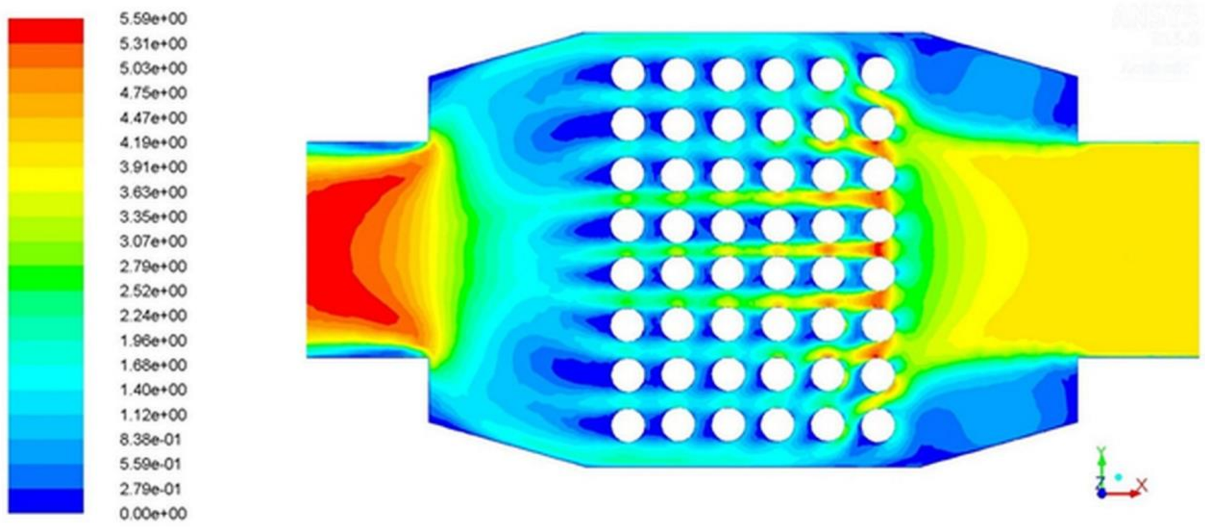

Fig. 8. Air velocity (in m/s) contours inside the longitudinal section of the modified apparatus, proposition 2, (t = 100 °C, v_r = 2 m/s, inlet on the right)

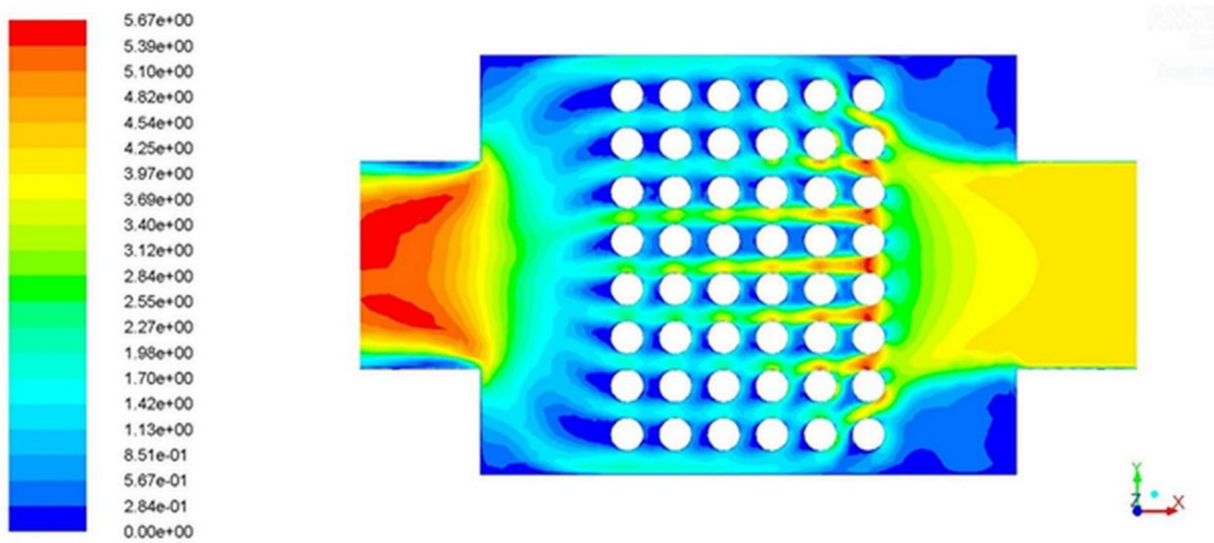

Fig. 9. Air velocity (in m/s) contours inside the longitudinal section of the modified apparatus perpendicular do the pipes, proposition 3, (t = 100 °C, v_r = 2 m/s, inlet on the right)

Local velocities are within the assumed range of values (between 2 and 6 m/s), nearly in the entire volume of the heat exchanger. For the highest gas inlet velocity, the maximum velocity between the 1st and the 2nd rows exceeds slightly (by 8 %) the assumed maximum velocity of 6 m/s; for the lowest inlet velocity in the last layer area (5th and 6th rows), it drops slightly (by 10 %) below the value of 2 m/s. As the distance from the inlet of the apparatus increases, the mean velocity changes a little (by 12 and 8 % for the lowest and the highest gas inlet velocity, respectively).

The inlet and outlet element design changes (propositions 2 and 3, Table 3) do not lead to changes in the profile of inter-pipe flow velocity in the heat exchanger of the examined devices (Figs. 8 - 9, and 12). In all modifications, the distance between the inlet of the device and pipes of the exchanger was so small that the gas stream had no time to extend above the diameter of inlet pipe. The analyzed modifications slightly increase the pressure drop in the device (by up to 5-7 % for the highest gas inlet velocity).

Modification 4 (Table 3) with the large opening angle of baffles (Fig. 10) extends the air stream in such a way that in the inter-pipe gaps, in the proximity of solid walls of the device, velocity increases. However, behind the baffle a dead zone with low gas velocity is formed, which causes velocity drop in the inter-pipe gaps behind the barrier (Figs. 10 and 12). This design causes slight increase in pressure drop (6% for $v_r = 6$ m/s).

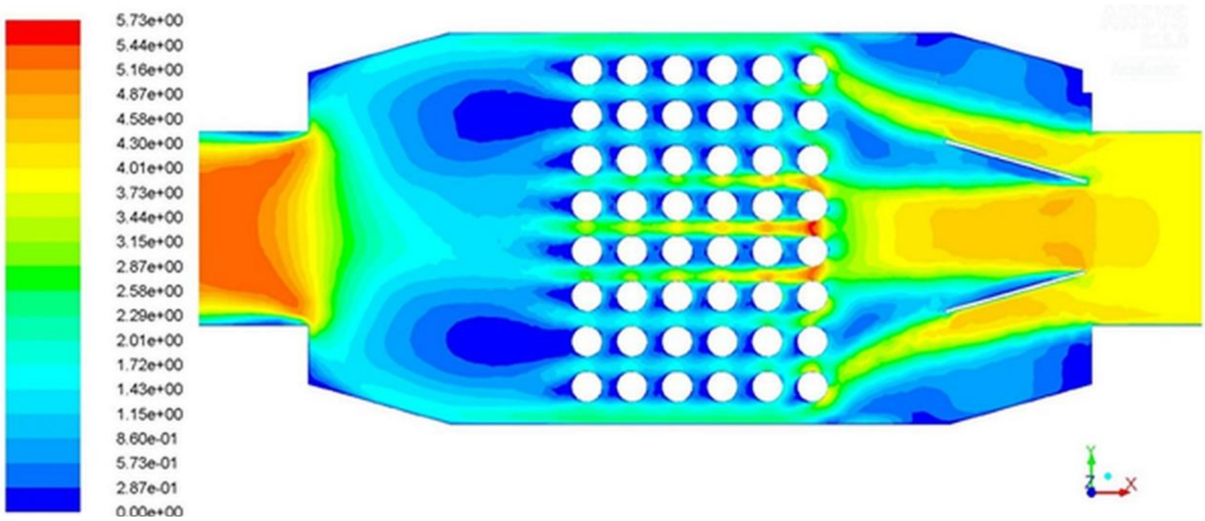

Fig. 10. Air velocity (in m/s) contours inside the longitudinal section of the modified apparatus perpendicular to the pipes, proposal 4, (t = 100 °C, v_r = 2 m/s, inlet on the right)

Modifications 2 - 4 were related with cross-section perpendicular to pipes of the heat exchanger. Therefore, the outcome of air flow guides, extending the stream crosswise and alongside the pipes of the heat exchanger was analyzed. Based on proposition 4, a revised design was proposed by adding two transverse plates of a longer size.

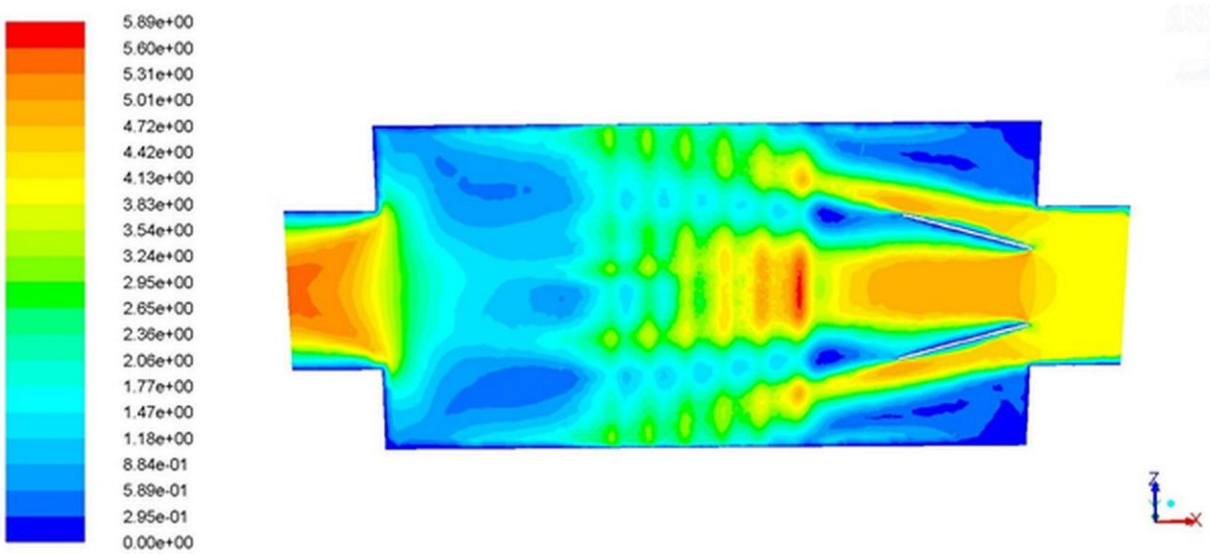

Fig. 11. Air velocity (in m/s) contours inside the longitudinal section of the modified apparatus parallel to the pipes, proposition 5, (t = 100 °C, v_r = 2 m/s, inlet on the right)

The suggested design of baffles leads to changes in air flow, both in parallel and perpendicular cross-section to pipes of the heat exchanger. Behind flow guides there are zones with reduced gas velocity, which is directed toward pipes closed to solid walls of the unit (Figs. 11 and 12). The proposed design causes significant increase of pressure drop in the device (by 26 % for v_r = 6 m/s).

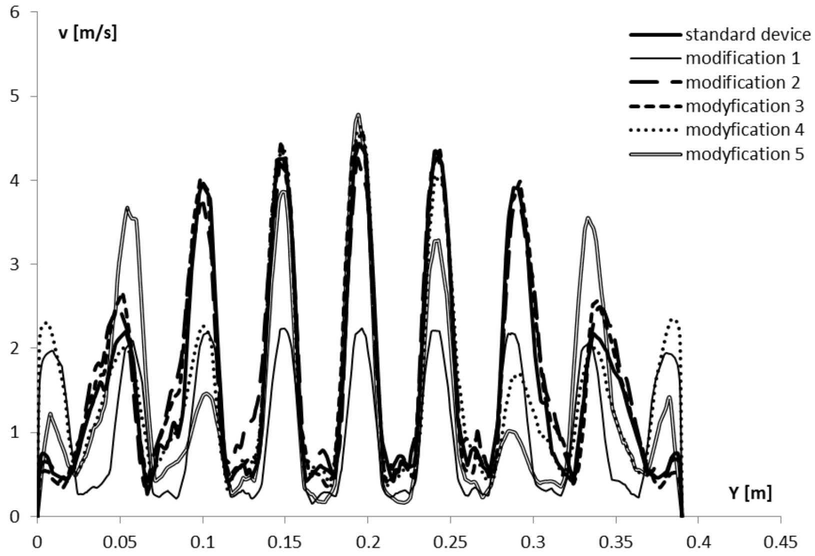

Fig. 12. Gas flow velocity in the cross section between 1st and 2nd row of pipes in the recuperator for different modifications (v_r = 2 m/s, t = 100°C)

5. CONCLUSIONS

Out of the tested modifications, proposition 1 seems to be the best construction, which is characterized by improvement in gas flow parameters: more homogenous inter-pipe gas flow in the heat exchanger and smaller pressure drop (by 60 % for v_r = 6 m/s). In this case one can take into account the increase of apparatus mass. Modifications 2 and 3 do not reduce the quality of the gas flow (velocity profiles and overall pressure drop are comparable to those in the standard recuperator), but they can decrease the mass of the device. Modifications 4 and 5 cause more homogenous inter-pipe gas flow in the heat exchanger and increase pressure drop and mass of the apparatus. Final conclusions on design change can be made only if experimental research tests are implemented with prototypes that will confirm the accuracy of the obtained results.

The studies were funded by the National Centre for Research and Development as a part of the project POIG.01.04.00-16-288/13.

SYMBOLS

d_r	inlet pipe external diameter, m
Δp	overall pressure drop in the apparatus, Pa
t	temperature, °C
v_r	average velocity calculated on the cross section of inter-pipe space, m/s
v	local velocity, m/s
w	average velocity calculated on the cross section of the empty apparatus, m/s
Y	distance from the wall parallel to heat exchanger pipes in recuperator, m

REFERENCES

Chang T.H., Lee C.H., Lee H.S., Lee K.S., 2015. Velocity profiles between two baffles in a shell and tube heat exchanger. *J. Therm. Sci.*, 4, 356-363. DOI: 10.1007/s11630-015-0795-x.

Gil S., Góral J., Horňak P., Ochman J., Wiśniewski T., 2015. Pressurized recuperator for heat recovery in industrial high temperature processes. *Arch. Metall. Mater.*, 60, 1847-1852. DOI: 10.1515/amm-2015-0315.

Goodarzi M., Nouri E., 2016. A new double-pass parallel-plate heat exchanger with better wall temperature uniformity under uniform heat flux. *Int. J. Therm. Sci.*, 102, 137-144. DOI: 10.1016/j.ijthermalsci.2015.11.012.

Jaworski Z., 2005. *Numeryczna mechanika płynów w inżynierii chemicznej i procesowej*, Akademicka Oficyna Wydawnicza Exit, Warszawa.

Pal E., Kumar I., Joshi J.B., Maheshwari N.K., 2016. CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles. *Chem. Eng. Sci.*, 143, 314-340. DOI: 10.1016/j.ces.2016.01.011.

Wen J., Yang H., Wang S., Xue Y., Tong X., 2015. Experimental investigation on performance comparison for shell-and-tube heat exchangers with different baffles. *Int. J. Heat Mass Transfer*, 84, 990-997. DOI: 10.1016/j.ijheatmasstransfer.2014.12.071.

Yang F., Zeng M., Wang Q., 2015. Numerical investigation on combined single shell-pass shell-and-tube heat exchanger with two-layer continuous helical baffles. *Int. J. Heat Mass Transfer*, 84, 103-113. DOI: 10.1016/j.ijheatmasstransfer.2014.12.042.

Received 25 September 2016
Received in revised form 20 October 2017
Accepted 18 November 2017