

NUMERICAL MODELING OF A SHELL-AND-TUBE HEAT EXCHANGER

МОДЕЛНИ ИЗСЛЕДВАНИЯ НА КОЖУХОТРЪБЕН ТОПЛООБМЕНЕН АПАРАТ

Ass. PhD. Raynov P.
UFT-Plovdiv, Bulgaria
p_rainov@uft-plovdiv.bg

Abstract: The aim of this work is numerical modeling of the hydrodynamics and heat transfer of a shell-and-tube heat exchanger. For the purpose of the study a 3D model with geometric dimensions corresponding to real was created. The simulations under the same boundary conditions as experiment were carried out. The independence of solution by the density and the shape of the mesh were investigated. For verification the experimental values for fluid temperatures at the outlets from the apparatus were used. The simulations of different operation modes in the apparatus were carried out. A modification in the geometry with the aim of raising the temperature on the cold fluid at the outlet was made. Results on vectors, velocity and temperature distribution in the apparatus were obtained. On the basis of the obtained results some design changes of the apparatus in order to improve the hydrodynamics have been proposed. The obtained results can be successfully used in the design, optimization and constructing of this type apparatus, as well as in the educational process.

Keywords: CFD, MODELING, HEAT EXCHANGER, HYDRODYNAMICS, ANSYS FLUENT

1. Introduction

The aim of this work is numerical modeling of the hydrodynamics and heat transfer of a shell-and-tube heat exchanger. The general appearance of the real apparatus is shown in Figure 1.



Fig. 1 Experimental apparatus.

For the purpose of the study a 3D model with geometric dimensions corresponding to real was created. The created 3D model is shown in Figure 2.

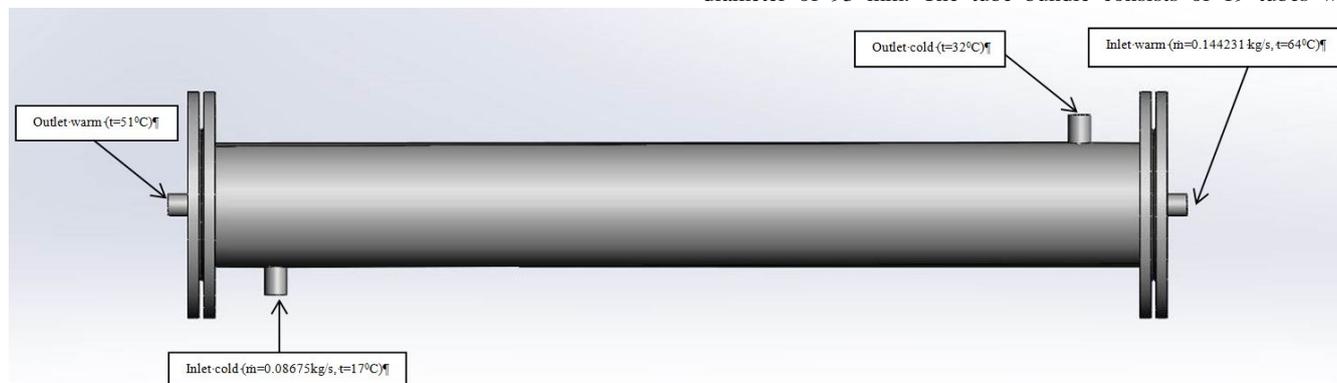


Fig. 2 3D model with experimental results.

The independence of solution by the density and the shape of the mesh were investigated. The simulations under same boundary conditions as experiment were carried out. The experimental values of main parameters are shown in Figure 2. For verification the experimental values for fluid temperatures at the outlets from the apparatus were used. In Table 1 are shown the relevant deviations.

Table 1: Deviations from the experiment.

Parameter	Deviation[°C]
Temperature at the outlet (cold fluid)	0,9
Temperature at the outlet (warm fluid)	0,2
Δt	0,65

The simulations of different operation modes in the apparatus were carried out. A modification in the geometry with the aim of raising the temperature on the cold fluid at the outlet was made. Results on vectors, velocity and temperature distribution in the

apparatus were obtained, as well as some key parameters. Multitude cross sections (longitudinally and transverses) in different typical planes of the apparatus were created, but in this work only those which are of interest were shown. The obtained results can be successfully used in the design, optimization and constructing of this type apparatus, as well as in the educational process. On the basis of the obtained results some design changes of the apparatus in order to improve the hydrodynamics have been proposed. All designations are generally acknowledged in the field of computational fluid dynamics and heat engineering.

2. Materials and methods

The created model shown in Figure 2 is with the same geometric dimensions and construction as the experimental apparatus and a counterflow working scheme was made. It is a one pass as regards to the tube side, and as regards to the shell side. Working fluids (warm and cold) are water. Cold is fed into the shell side and warm is fed the tube side. The shell has an internal diameter of 93 mm. The tube bundle consists of 19 tubes with

diameter $\phi 10/2$ mm and length 700 mm. To create the 3D model shown in Figure 2 a software product SolidWorks was used. For creating the meshes a software product Ansys Mescher was used. For numerical modeling a software product Ansys Fluent was used. After investigation of the independence of solution by density and shape of mesh the optimal shown in Figure 3 was obtained. It consists of 1187551 nodes and 6979399 cells. The simulations on a computer with processor Intel Core I7 and 16 GB RAM were carried out. Performing a simulation with shown mesh and convergence of solution 10^{-4} takes approximately 20 hours. To close the system of differential equations of motion a standard k- ϵ turbulence model was used [1], [4], [5]. The post processing in Ansys Results was carried out. For the purpose of present work a several modeling investigations in different operating modes of the apparatus were carried out. They are the following:

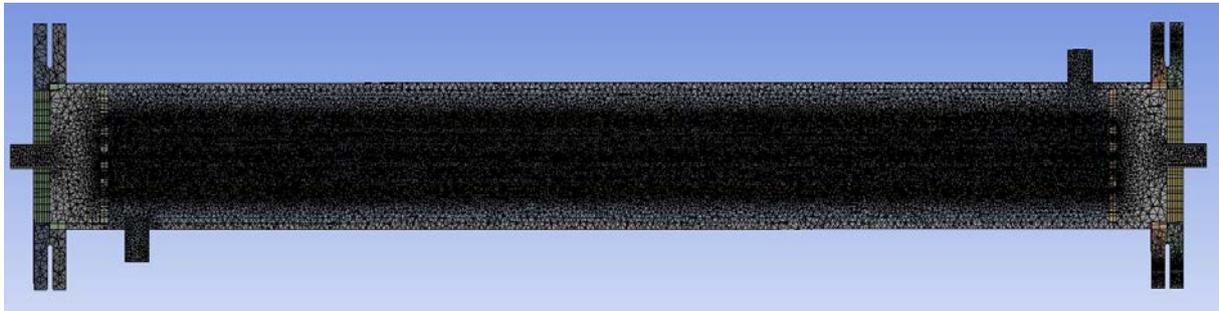


Fig. 3 Optimal mesh.

- Counterflow movement of the working fluids at a mass-flow rate of cold fluid $\dot{m}=0,08675$ kg/s (used for verification) corresponding to experimental apparatus;
- Counterflow movement of the working fluids at a mass-flow rate of cold fluid $\dot{m}=0,1388$ kg/s;
- Parallel flow movement of working fluids (changing the direction of the cold fluid).

In order to examine the influence of baffles onto hydrodynamics, respectively, heat transfer a modification of the model including placement of four screens in the shell side was made. Figure 4 shows the general appearance of the 3D model with baffles.

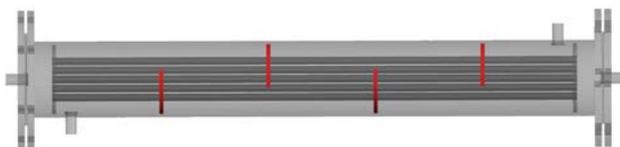


Fig. 4 3D model with baffles.

The baffles in the heat exchangers used to increase the velocity and changing the direction of movement of the fluid, a consequence of which there is provided a crosswise streamlined of the tube bundle, wherein the heat transfer coefficient is increased 2-3 fold compared to streamline of the longitudinal bundle. The cropped part of the segment baffles constitutes 15-40% of the diameter in the shell. The recommended distance between the baffles in the shell side a crosswise streamlined of the tube bundle is:

$$(1) \quad l = (40 \div 60) \cdot d_2$$

It is estimated that the optimum distance is:

$$(2) \quad l = D_1 / 2$$

Experience suggests that the two recommendations are not contradictory, but complementary [2].

It is well known that the distance, length, number and type of baffles significantly influence on the hydrodynamics respectively heat transfer in these apparatus. In the literature is a dedicated dependency for these typical dimensions and are indicated equations with the aid of that can define these construction dimensions [3].

Nevertheless, this remains a complex technical-economic task and for each concrete model is desirable to be approached individually.

3. Results and discussion

Table 2: Numerical results.

Parameters Model	Q[W]	ΔP [Pa]	t_{out}^{cold} (Area-Weighted Average)[$^{\circ}C$]
Counterflow $\dot{m}=0,08675$ kg/s	5119	260	31,1
Counterflow $\dot{m}=0,1388$ kg/s	5954	635	27,25
Parallel flow $\dot{m}=0,08675$ kg/s	4720	260	30
Counterflow $\dot{m}=0,08675$ kg/s with baffles	8568	268	40,6

Some numerical results for the various study models are shown in Table 2. From the obtained results it can be seen that increasing the mass-flow rate of the cold fluid with 37,5% lead to decreasing the temperature at the outlet with $3,85^{\circ}C$, increasing the heat transfer of the apparatus by 14% and increase the hydraulic losses by 59%. The Parallel flow movement of working fluids showed a decrease in heat transfer by 7,8% and reduced the temperature by $1,1^{\circ}C$. In the model with baffles the heat transfer is increased by 40,2%, the temperature was increased by $9,5^{\circ}C$ and the hydraulic losses increased by 3%.

Figure 5 shows the velocity vectors on the right side of the apparatus in a longitudinal section along the axis of the apparatus in $\dot{m}=0,08675$ kg/s. Can clearly see the presence of two pronounced circulation zones. There is also the presence of a circulating flow between the nozzle and the tube sheet.

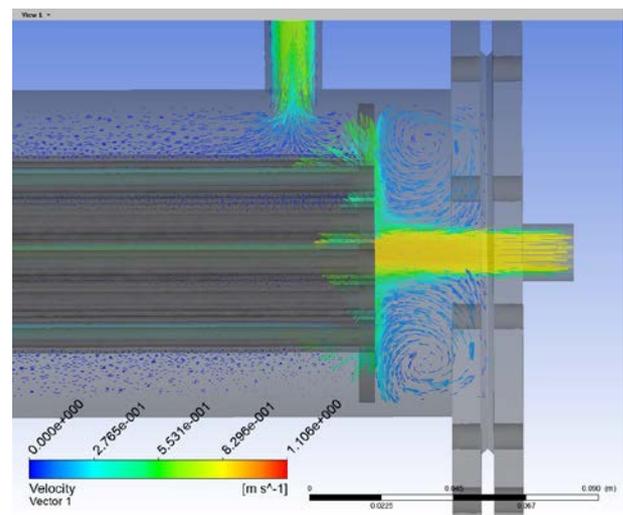


Fig. 5 Velocity vectors on the right side of the apparatus in $\dot{m}=0,08675$ kg/s.

Figure 6 shows the velocity vectors perpendicular to the section shown in Figure 5 in $\dot{m}=0,08675$ kg/s.

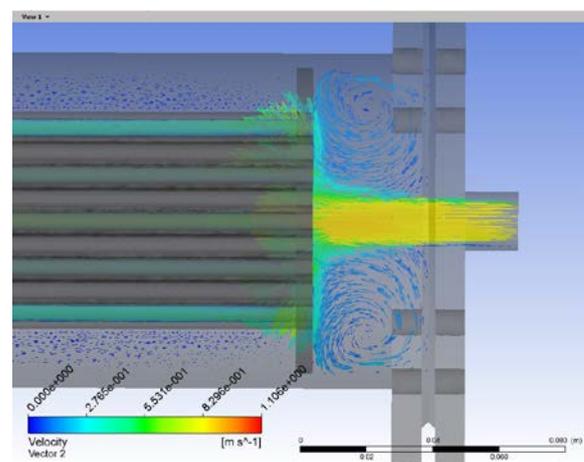


Fig. 6 Velocity vectors on the right side of the apparatus perpendicular in $\dot{m}=0,08675$ kg/s.

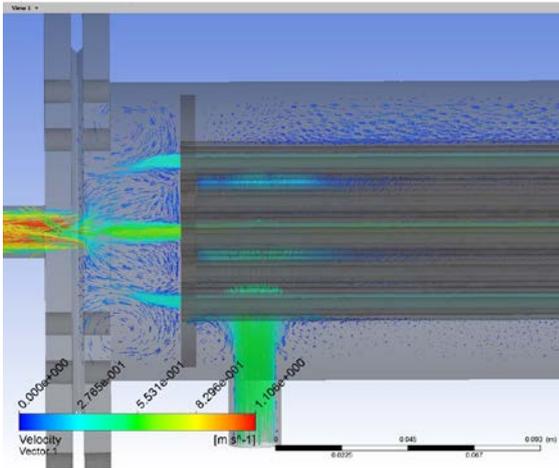


Fig. 7 Velocity vectors on the left side of the apparatus in $\dot{m}=0,08675$ kg/s.

It can be distinguished again two pronounced circulation areas. It could be argued that the entire space in the inlet plenum is filled with circulating flow.

Figure 7 shows the velocity vectors on the left side of the apparatus in a longitudinal section along the axis of the apparatus in $\dot{m}=0,08675$ kg/s.

Figure 8 shows the velocity vectors perpendicular to the section shown in Figure 7 in $\dot{m}=0,08675$ kg/s.

Here again there are two pronounced circulation zones, but with a smaller size and pushed to the wall by the the incoming flow of the tubes in the outlet plenum. There is also the presence of a circulating flow between the nozzle and the tube sheet.

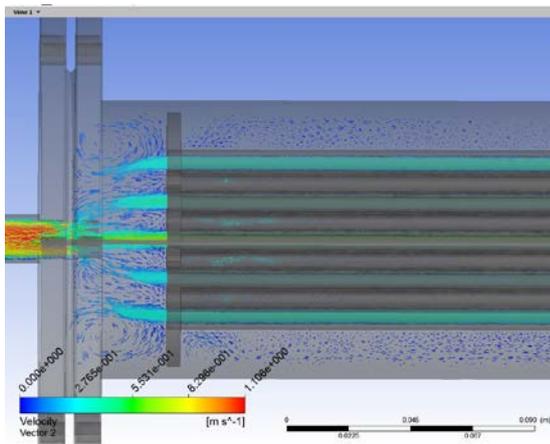


Fig. 8 Velocity vectors on the left side of the apparatus perpendicular in $\dot{m}=0,08675$ kg/s.

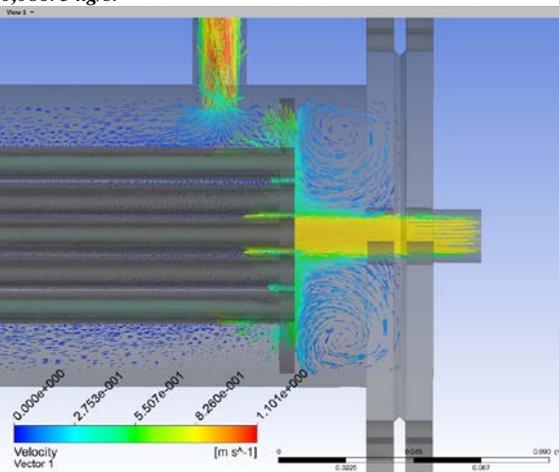


Fig. 9 Velocity vectors on the right side of the apparatus in $\dot{m}=0,1388$ kg/s.

Figure 9 shows the velocity vectors on the right side of the apparatus in a longitudinal section along the axis of the apparatus in $\dot{m}=0,1388$ kg/s.

Figure 10 shows the velocity vectors perpendicular to the section shown in Figure 9 in $\dot{m}=0,1388$ kg/s.

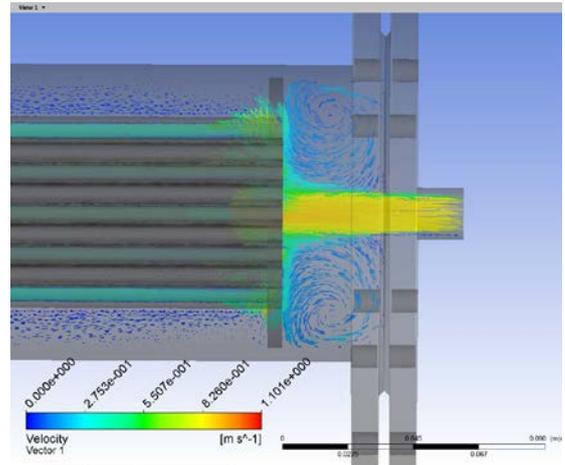


Fig. 10 Velocity vectors on the right side of the apparatus perpendicular in $\dot{m}=0,1388$ kg/s.

In both figures it is seen that increasing the mass-flow rate at the inlet to the shell side is no change in the vector image, which is not at the output, and that is shown in Figure 11 and Figure 12.

Figure 11 shows the velocity vectors on the left side of the apparatus in a longitudinal section along the axis of the apparatus in $\dot{m}=0,1388$ kg/s.

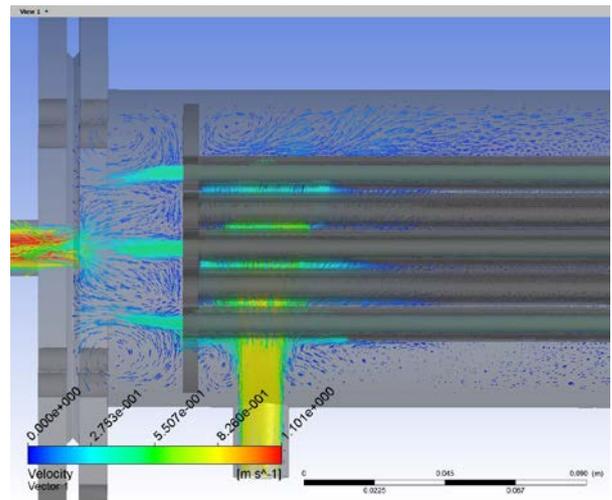


Fig. 11 Velocity vectors on the left side of the apparatus in $\dot{m}=0,1388$ kg/s.

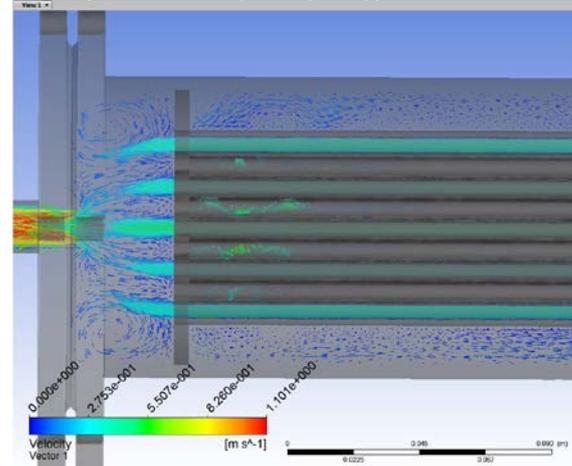


Fig. 12 Velocity vectors on the left side of the apparatus perpendicular in $\dot{m}=0,1388$ kg/s.

Increasing the flow rate gives rise to a circulation of the flow in the zone between the tubes, the shell and the tube sheet at the top of the apparatus. Also is observed increase in the circulating zone between nozzle and tube sheet.

Figure 12 shows the velocity vectors on the left side of the apparatus perpendicular to the section shown in Figure 11 in $\dot{m}=0,1388 \text{ kg/s}$. Can observe that an increasing the flow rate a circulation zone in bottom of the shell near the tube sheet (Figure 8) does not exist.

Figure 13 shows the distribution of velocity along the apparatus by the model used for verification, and Figure 14 by the model with baffles. In the figures clearly evident stagnation zones in inlet plenum and outlet plenum caused by circulating flows shown in Figure 5 and Figure 7. In Figure 13 clearly evident zone of increase the velocity in the shell side nears the outlet. Figure 14 shows an increase of the velocity in zones of the gap section of the baffles.

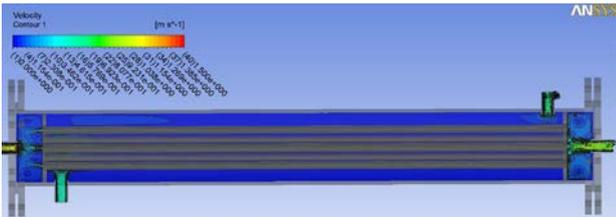


Fig. 13 Distribution of velocity along the apparatus by the model used for verification.

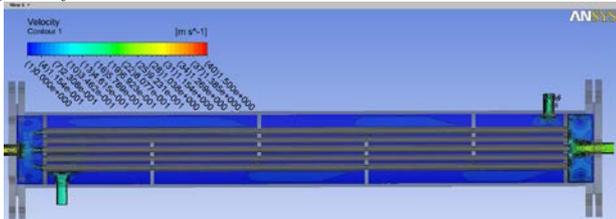


Fig. 14 Distribution of velocity along the apparatus by the model with baffles.

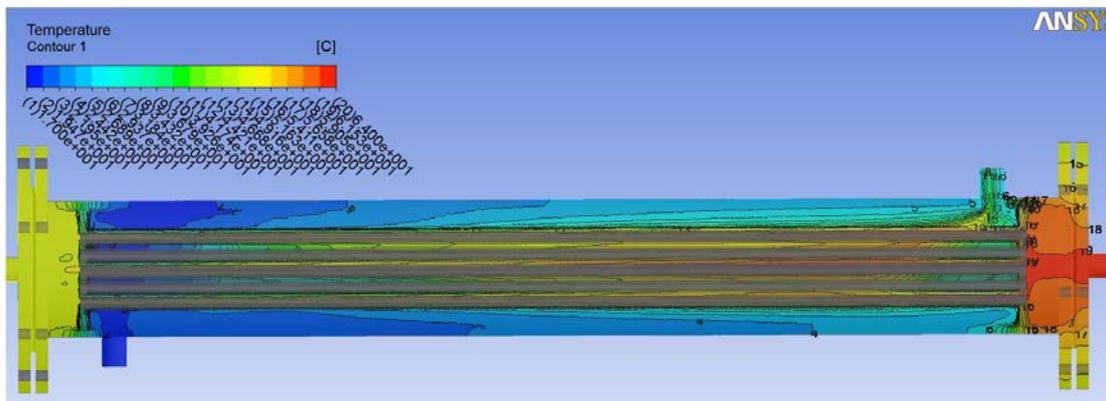


Fig. 15 Distribution of temperature along the apparatus by the model used for verification.

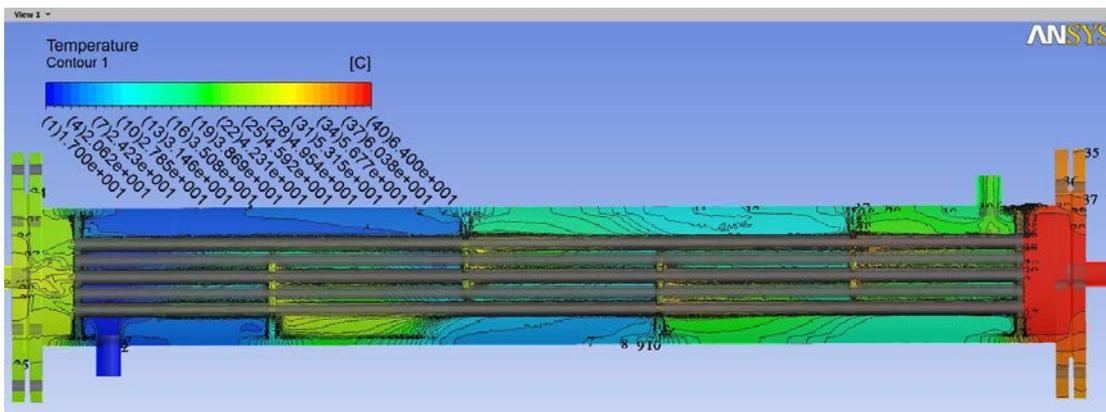


Fig. 16 Distribution of velocity along the apparatus by the model with baffles.

Figure 15 shows the distribution of temperature along the apparatus by the model used for verification, and Figure 16 by the model with baffles. Comparing the two figures shows that the temperature field of the Figure 16 is more uniform due to an increase in the velocity at zones of the gap section by baffles and crosswise streamline of the tube bundle. This in turn is closely associated with an increase of the heat transfer (Table 2).

4. Conclusions

A modification of the model leads to a significant increase in outlet temperature of the cold fluid while she negligible influence on the hydraulic losses. To eliminate or reduce size of the circulation zones in both plenums is required geometric change of headers. To eliminate or reduce size of the circulation zones between the nozzle and the tube sheet is required change of distance between them. The obtained results can be successfully used in the design, optimization and constructing of this type apparatus, as well as in the educational process.

5. Literature

[1] Ангелов, М. Възможности за моделни изследвания на обменните процеси в двуходов топлообменник. Научна конференция с международно участие „Хранителна наука, техника и технологии“, Пловдив, 19-20 Октомври, Научни трудове УХТ, Том LIX, pp 768-773, 2012. (Ангелов, М., Райнов, П., Стоева, Д.)
 [2] Емануилов, А. Теплообменни апарати. Изд на УХТ – Пловдив, 2008.
 [3] Мартыненко, О. Справочник по теплообменникам. Энергоатомиздат, 1987, Перевод с английского. (Мартыненко, О., Михалевица, А., Шикова, В.)
 [4] Launder, B. E. The numerical computation of turbulent flow. Comp. Mech. In Appl. Mech. Engng, 1974, Volume 3, Pages 269-289. (Launder, B. E., and Spalding, D. B.)
 [5] Stoeva, D. Intensification of the heat transfer through corrugated wall. Journal of Faculty of Food Engineering, Volume XIII, Issue 1, pag. 14 – 22, 2014.