FACTA UNIVERSITATIS
Series: **Mechanical Engineering** Vol. 11, N^{o} 2, 2013, pp. 169 - 180

3D NUMERICAL SIMULATIONS OF THE THERMAL PROCESSES IN THE SHELL AND TUBE HEAT EXCHANGER

UDC 66.04:536.2

Mića Vukić[1], Goran Vučković[1], Predrag Živković[1], Žarko Stevanović[2], Mladen Tomić[3]

[1]University of Niš, Faculty of Mechanical Engineering, Niš, Serbia
[2]Institute "Vinča", Belgrade, Serbia
[3]School of Higher Technical Professional Education Niš, Serbia

Abstract: *A shell and tube heat exchanger (STHE) is one of the most often used apparatuses in chemical industry. One of the main goals of the STHE manufacturers is to improve their exploitation reliability and efficiency. Two approaches to the STHE design improvement are possible: experimental investigation, which is very expensive and time-consuming because of the shell side complex geometry, and numerical investigations. Numerical simulations can be used to check the old and to develop new and more efficient STHE designs. In this paper, the results of the numerical investigations of fluid flow and heat transfer in the laboratory experimental STHE are presented. Numerical simulation has been performed by using the PHOENICS code. The tube bundle has been modeled by using the concept of porous media. Standard k-ε turbulence model is used.*

Key Words: *Heat Exchanger, Local Heat Transfer, Numerical Investigation*

1. INTRODUCTION

Shell and tube heat exchangers are apparatuses in which heat exchange between hotter and colder fluid is performed. The fluid flowing through the tubes is called a tube fluid, and the fluid flowing around the tube bundle is called a shell side fluid.

The baffles, placed in the shell side space, are providing a cross flow direction of the shell side fluid; hence, a more intensive heat exchange between fluids could be realized. Besides, the baffles are tube bundle carriers, which helps them decrease deflection in the horizontal as well as vibrations in the horizontal and vertical apparatuses.

Received November 20, 2013
Corresponding author: Mića Vukić
Faculty of Mechanical Engineering, A. Medvedeva 14, 18000 Niš, Serbia
E-mail: micav@masfak.ni.ac.rs

STHEs usually have a combined fluid flow, which means that there is parallel in one, and a counterflow in the other part of the exchanger. If the STHE is with a so-called „full tube bundle“, the shell side fluid flows through the baffle cuts along the tubes.

On the shellside, there is not just one stream. In addition to the main cross-flow one, four leakage or bypass streams exist as a result of design type baffle to tubes, baffle to shell and tube bundle to shell gaps (tube–to–baffle hole leakage stream, bundle bypass stream, pass–partition bypass stream and baffle–to–shell leakage stream).

Basically, one can conclude that heat transfer between fluids in the STHEs is highly influenced not only by thermal and flow quantities, such as inlet temperatures and velocities, but also by baffle cut size, baffle spacing, size of inlet and outlet zones and number of baffles [5, 7, 8, 9, 10, 11].

Considering the fact that the flow visualization and a detailed measurement of the shellside turbulent characteristics in the heat exchanger are very difficult, calculated pressure, velocity and temperature fields, as well as turbulent characteristics fields (e.g. field of turbulent kinetic energy dissipation), can be of great significance in solving this, very complex, turbulent thermal – flow processes in the shell and tube heat exchangers, when it comes to further investigation and designing of reliable and efficient shell and tube heat exchangers [7, 8].

2. MATHEMATICAL MODEL

Numerical solution of heat transfer, fluid flow and other processes can begin when the laws that govern these processes are mathematically expressed, mainly using partial differential equations. Partial differential equation describing some of the mass, momentum or energy transport phenomena, is actually expressing the principle of conservation for a certain transport property.

Turbulent heat transfer mathematical model in the STHE (Fig. 1) where heat is exchanged between the shell side and the tube side fluid, is formulated with the following assumptions: the flow fields are stationary, three-dimensional, incompressible, viscous, turbulent with large Re numbers in the defined boundaries. The thermo-physical properties of the process fluids, in the concrete case are considered to be constant. The heat loss outside the exchanger, and the radiation are not taken into consideration.

Fig. 1 Laboratory STHE with cross segmental baffles

The resistance to heat transfer through the tube wall as well as the fouling resistance are also neglected.

The flow characteristics of both fluids have a direct influence on the efficient heat exchange and pressure drops in both the shell and tube bundle. The approaches of the distributed resistance and the porosity concept have been used in modeling both the process of heat transfer and the STHE geometry.

In this paper, the time averaged conservation equations: of mass, momentum and energy, on the shell side of STHE, in the referent curvilinear coordinates are given with the following simple form [1]:

$$\frac{\Delta}{\Delta x^{(j)}}\left[\rho U^{(j)}\Phi-\left(\Gamma_{\Phi}g^{(jm)}\frac{\partial\Phi}{\partial x^{(m)}}-\overline{\rho\phi' u'^{(j)}}\right)\right]=S_{\Phi}+S_{\Phi}^{ex} \tag{1}$$

The averaged conservation equations are formally equal to the "instant" ones only by replacement of instant values with the average ones; as a result of averaging they have extra terms, namely, so-called Reynolds turbulent stresses in the momentum conservation equation and turbulent flux of scalar in the equation for scalar, of the form $-\overline{\rho\phi' u'^{(j)}}$. There is a set of turbulent models connecting the Reynolds turbulent stresses and the turbulent scalar fluxes with the mean flow characteristics closing the set of equations. One of the frequently used turbulent models is a two-equation k-ε turbulent model. It is based on introducing effective diffusivities for transport quantities under consideration, which are the sum of molecular and turbulent diffusivities.

The terms and coefficients that form expression (1) depend on the conservation equation and should be specified for each conservation equation. Diffusion coefficient Γ_{Φ} and standard source term S_{Φ}, depending on the nature of dependent variable Φ, are given in Table 1.

Table 1 Diffusion coefficient and standard source term

	Φ	Γ_{Φ}	S_{Φ}
Mass	1	0	0
Momentum	$U^{(i)}$	μ_{eff}	$-g^{(ji)}\frac{\partial P}{\partial x^{(j)}}-[\rho U^{(j)}U^{(i)}-(\tau^{(jm)}-\rho\overline{u'^{(j)}u'^{(m)}})]\begin{pmatrix} i \\ m\ j \end{pmatrix}$
Energy	$c_{p,o}T_o$	$\frac{\mu_{eff}}{\sigma_h}$	0
Turbulent kinetic energy	k	$\frac{\mu_{eff}}{\sigma_k}$	$\mathbf{P}-\rho\varepsilon$
Dissipation rate for k	ε	$\frac{\mu_{eff}}{\sigma_{\varepsilon}}$	$C_1\frac{\varepsilon}{k}\mathbf{P}-C_2\rho\frac{\varepsilon^2}{k}$
$\mathbf{P}=g_{(ik)}[\tau_{tur}^{(ij)}-2/3\cdot g^{(ij)}(\rho k+\mu_{tur}\nabla_{(m)}U^{(m)})]\nabla_{(j)}U^{(k)}$			

At the right hand side of equation (1) S_{Φ}^{ex} is an extra source term. In this paper, special attention will be paid to S_{Φ}^{ex}. The extra source term expresses: flow resistance, heat and mass exchange between the fluids. The extra source term in the momentum equation takes into account the flow resistance caused by the tube bundle in the STHE

shell (distributed resistance). Expressions of local hydraulic resistance are taken from Ref. [2, 8].

Extra source term in the energy equation for the shell side fluid can be given as [7]:

$$S_{T_o}^{ex} = K_{\infty}(T_o - T_c) \tag{2}$$

Volume overall heat transfer coefficient K_{co} [W/m^{3}K], reduced to the volume unit that is enveloped by the heat transfer surface, is defined as:

$$K_{co} = \frac{1}{\frac{1}{h_c^v} + \frac{1}{h_o^v}} = \frac{4}{d_s\left(\frac{1}{h_c} + \frac{1}{h_o}\right)} \tag{3}$$

where h_c and h_o [W/m^{2}K] are the heat transfer coefficients scaled to the heat exchange surface unit.

In this paper, in numerical experiments for determination of the volume overall heat transfer coefficient, two approaches are used. Firstly, the average volume overall heat transfer coefficient is determined by generalizing our experimental results, and, secondly, the procedure for determination of the local volume overall heat transfer coefficient is developed and built in the CFD-code, which can thus build all the needed fields. In this procedure, the local Re numbers for the fluid on the shell side are defined in correlation to the external tube diameter and the resulting fluid velocity. In order to determine the thermal-flow fields in the STHE shell as well as the temperature field for the tube fluid, a set of differential equations has to be solved together with the algebraic (constitutive) relations and the corresponding unambiguity conditions.

3. NUMERICAL INVESTIGATION

Solving the set of partial differential equations with the PHOENICS software package is based on using the finite volume method. Effective flow side surfaces A_{cf} and cell volume V are proportional to geometrical quantities A_{cf}^* and V^*, where the proportion coefficients are the porosity factors that are denoted as β with corresponding subscripts for sides and volumes. This principle is known as the porous media concept where the porosity factor takes values $\beta = 0.0 - 1.0$. The chosen face or whole cell is completely blocked for $\beta = 0$ (when the cell is at the wall), and completely free for $\beta = 1$. The porosity factors can be different for each cell; therefore, with complex geometrical configurations additional dependent variables can be introduced.

For solving the general discretization equation SIMPLE procedure is used of Patankar and Spalding [3] which uses staggered grid and combines the continuity equations with momentum ones for pressure determination.

All the numerical experiments have been made for the same initial fluid parameters as in real experiments [7].

First discretization of the integration domain in BFC coordinates has been made by grid generator in the SATELITE mode using Q1 file. By testing a large number of different density grids, with and without matching the frames, a unique nonuniform grid

for all numerical experiments is used, with number of cells in $x^1 \equiv x$, $x^2 \equiv y$ and $x^3 \equiv z$ directions, respectively, $24 \times 24 \times 128$, where z coordinate is pointed in direction of the shell axis (Fig. 2). The grid uniqueness is represented in the aspect that for all the numerical experiments the same grid lines in the cross section are used (grid density in x-y plane is adjusted to different sizes of the baffle cut), while the grid density, in z - direction, is adjusted to the number, position and thickness of segmental baffles in the shell. In the cells that are on the segmental baffles, the fluid flow is blocked unlike the heat transfer which is free.

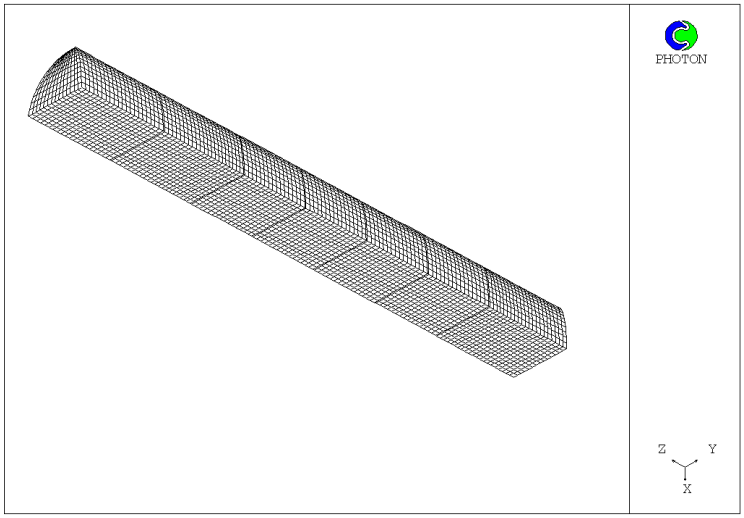

Fig. 2 Segment of numerical grid (plane x = 12, y = 15, z = 128)

Then by using the porosity concept, in Q1 file and GROUND file the STHE geometry modeling is carried out. Considering the position and symmetry of the tube bundle in the shell, and the prechosen numerical grid, eight constant porosity factor fields have been defined in the cross section. Determination of the extra source terms of the distributed resistance and volume heat transfer coefficients has been made by linearization technique of the special source terms. The corresponding subroutines are written in GROUND and Q1 file [5]. For turbulence modeling on the shell side, a standard k-ε model has been used [8].

Firstly, a series of numerical experiments with the average volume heat transfer coefficient is made for verifying both mathematical and numerical models. Then a series of experiments with local heat transfer coefficient on the shell side is performed. Naturally we have to emphasize that in all the numerical experiments the average value of the heat transfer coefficient on the tube side has been used, determined on the bases of the real experiments. This is absolutely justifiable by the fact that neither the volume flow rate nor the fluid temperature at the tube bundle inlet varied.

In order to verify mathematical and numerical models, the numerical results are compared to the experimental ones [7]. We have compared the values of the fluid tempera-

tures at the outlet for both fluids as well as the fluid temperatures inside the shell (in the shell axis).

Fig. 3 shows the comparison between the experimental and the numerical values of the heating fluid temperature by using the average and the local volume overall heat transfer coefficient in the shell axis for the case of one segmental baffle $N_p = 1$, with baffle cut of BC = 22%, without leakage flow between the baffle and the shell, for the heating fluid flow rate of $V_t = 4m^3/h$ and the initial temperature of $t_{1t} = 60°C$.

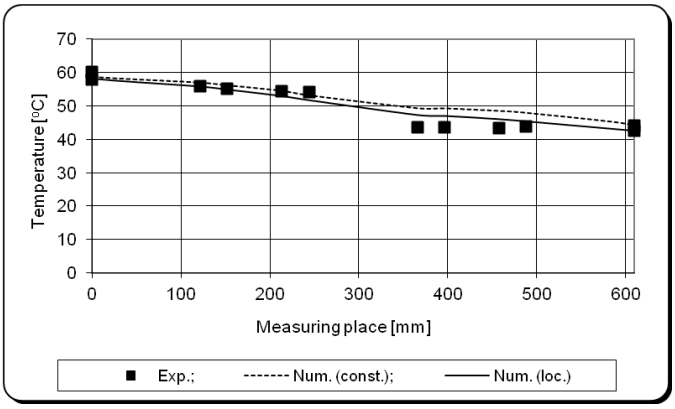

Fig. 3 Variation of heating fluid temperature along the STHE's axis

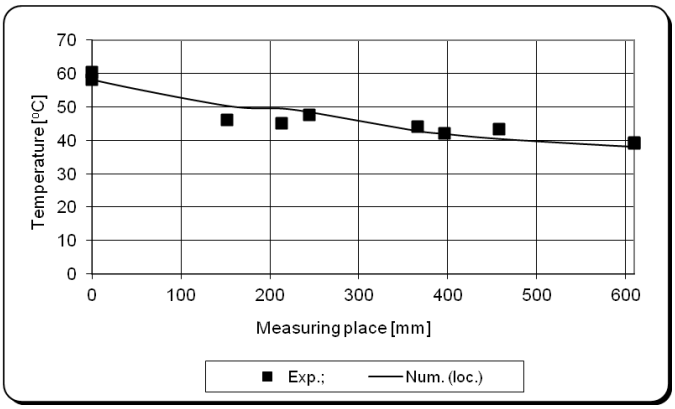

Fig. 4 Variation of heating fluid temperature along the STHE's axis

Fig. 4 shows the comparison between the experimental and the numerical values of the heating fluid temperature by using the local volume overall heat transfer coefficient in the shell axis for the case of three segmental baffle $N_p = 3$, with baffle cut of BC = 22%, with leakage flow between the baffle and the shell, for the heating fluid flow rate of $V_t = 3m^3/h$ and initial temperature of $t_{1t} = 60°C$.

Fig. 5 shows the comparison between the experimental and the numerical values of the heating fluid temperature by using the average and the local volume overall heat transfer

coefficient in the shell axis for the case of five segmental baffle $N_p = 5$, with baffle cut of BC = 22%, with leakage flow between the baffle and the shell, for the heating fluid flow rate of $V_t = 3m^3/h$ and the initial temperature of $t_{1t} = 60°C$.

Fig. 6 shows the comparison between the experimental and the numerical values of the heating fluid temperature by using the local volume overall heat transfer coefficient in the shell axis for the case of five segmental baffle $N_p = 5$, with baffle cut of BC = 32%, with leakage flow between the baffle and the shell, for the heating fluid flow rate of $V_t = 3m^3/h$ and the initial temperature of $t_{1t} = 60°C$.

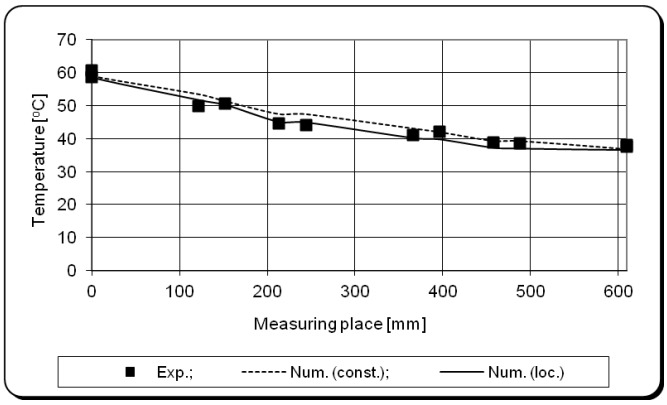

Fig. 5 Variation of heating fluid temperature along the STHE's axis

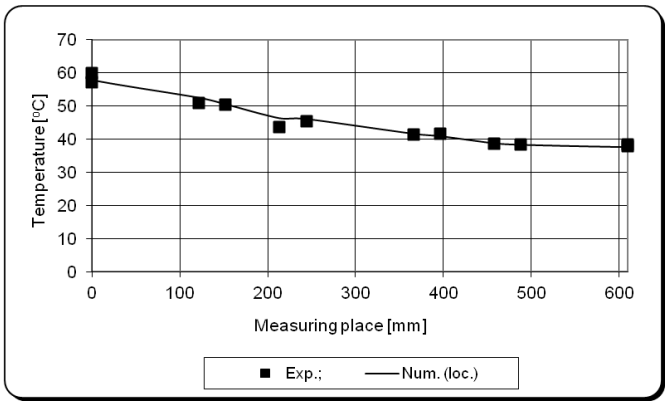

Fig. 6 Variation of heating fluid temperature along the STHE's axis

On the basis of the shown results we conclude that there is satisfactory agreement between the experimental and the numerical results and that the employed numerical model can be successfully applied to the simulation of the thermal-flow processes in the shell and tube heat exchangers. The maximum deviation between the measured and the numerical values of certain temperature for all experiments was 5%.

In the Figs. 8-10 we have the results of numerical simulations for the experiments with five segmental baffles $N_p = 5$, with the baffle cut of BC = 22%, for the heated fluid

parameters: flow rate of $V_h = 9m^3/h$ and the initial temperature of $t_{1h} = 15°C$, heating fluid parameters: flow rate of $V_t = 3m^3/h$ and the initial temperature of $t_{1t} = 60°C$.

Fig. 8 shows velocity field on shellside fluid for case without leakage flow between the baffle and the shell. Fig. 9 shows velocity field on shellside fluid for case with leakage flow between the baffle and the shell.

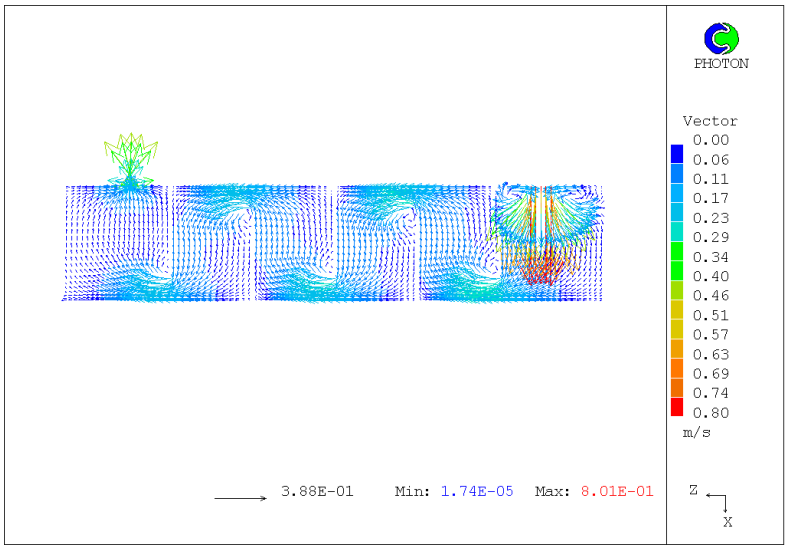

Fig. 8 Velocity field (plane y = 13) – no leakage

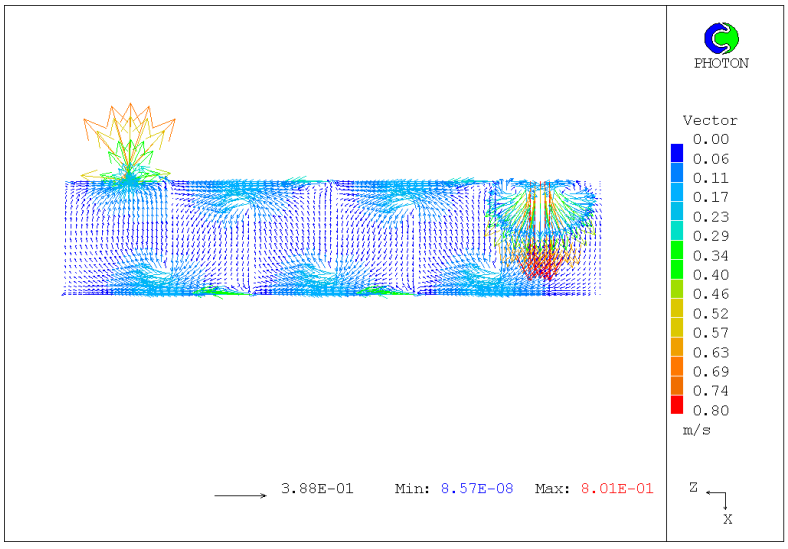

Fig. 9 Velocity field (plane y = 13) – with leakage

Fig. 10 shows velocity field on shellside fluid for case with leakage flow between the baffle and the shell.

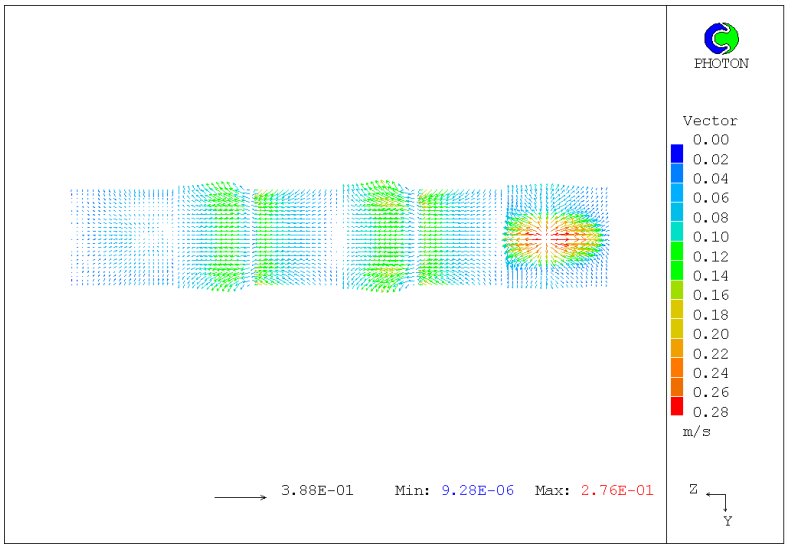

Fig. 10 Velocity field (plane x = 8) – with leakage

Figs. 11-15 present the results of numerical simulations for the experiments with one segmental baffles N_p = 1, with the baffle cut of BC = 22%, for the heated fluid parameters: flow rate of $V_h = 9m^3/h$ and the initial temperature of t_{1h} = 15°C, heating fluid parameters: flow rate of $V_t = 3m^3/h$ and the initial temperature of t_{1t} = 60°C.

Fig. 11 shows velocity field around segmental baffle, for case with leakage flow between the baffle and the shell.

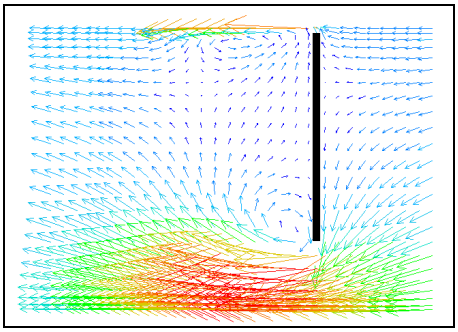

Fig. 11 Recirculation zone behind baffle – with leakage

Fig. 12 shows temperature field for tubeside fluid, for the case without leakage flow between the baffle and the shell.

Fig. 13 shows temperature field for shellside fluid, for the case without leakage flow between the baffle and the shell.

Fig. 14 shows temperature field for shellside fluid, for the case with leakage flow between the baffle and the shell.

Fig. 15 shows heat transfer coefficient field around segmental baffle for shellside fluid, for the case with leakage flow between the baffle and the shell (plane z-x).

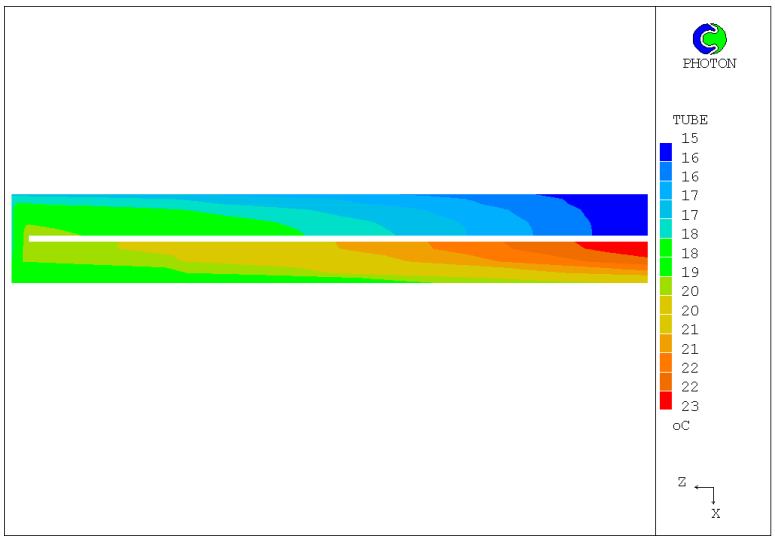

Fig. 12 Temperature field for tubeside fluid (plane y = 12) – no leakage

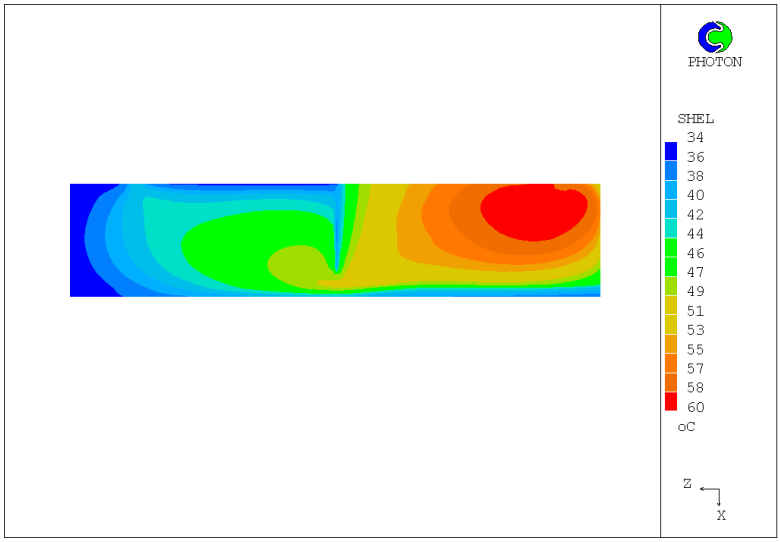

Fig. 13 Temperature field for shellside fluid (plane y = 12) – no leakage

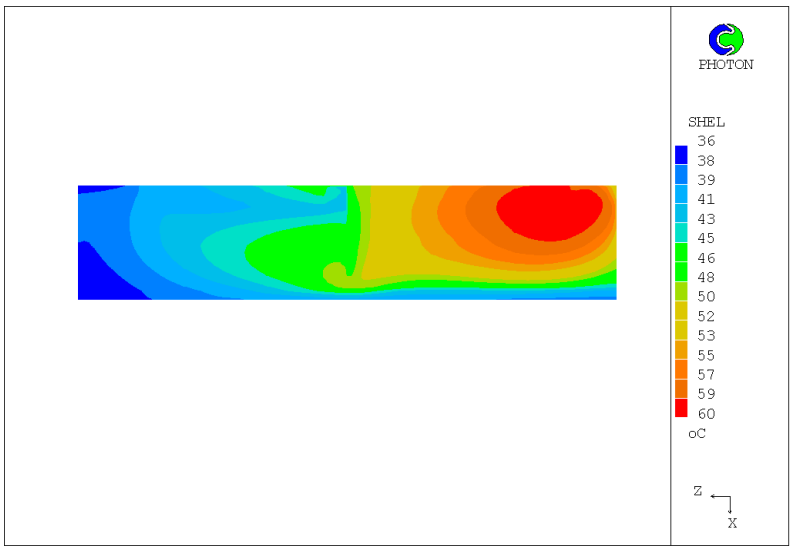

Fig. 14 Temperature field for shellside fluid (plane y = 12) – with leakage

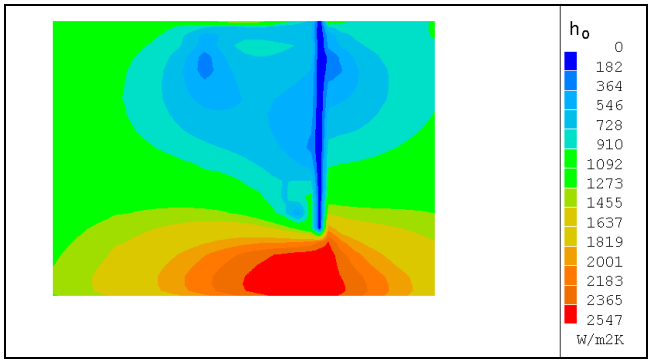

Fig. 15 Heat transfer coefficient field (plane y = 12, z = 45 ÷ 70) – with leakage

4. CONCLUSIONS

In this paper the results of numerical experiments carried out on the basis of PHOENICS software package applied on thermal-flow processes in a shell and tube heat exchanger, are shown. For the illustrative purposes the cross correlations between the experimental and the numerical results are shown. Satisfactory agreement is observed. On the basis of the results shown above we conclude that there is satisfactory agreement between the experimental and the numerical results and that the employed numerical model can be successfully applied to the simulation of the thermal-flow processes in the heat exchangers. The results of the performed numerical simulations show that the STHE's heat exchange strongly depends on the shell side geometry.

REFERENCES

1. I. Demirdžić, *Metod konačnih zapremina za proračun transportnih procesa u turbulentnim tokovima proizvoljne konfiguracije*, Akademija Nauka i umjetnosti Bosne i Hercegovine, Posebna izdanja: Knjiga LXVI, Odeljenje tehničkih nauka: Knjiga 13, Sarajevo, 1984.
2. L. Huang, J. Wen, T. Karayiannis, 1996, *CFD Modeling Of Fluid Flow And Heat Transfer In Shell And Tube Heat Exchanger*, PHOENICS Journal, Vol.9, No.2, pp.181-209.
3. S. Patankar, 1980, *Numerical Heat Transfer And Fluid Flow*, Hemisphere, Washington.
4. H. Rosten, B. Spalding, 1987, *The PHOENICS Equations*, CHAM TR/99.
5. B. Spalding, 2005, *A simplified CFD method for the design of heat exchangers*, Proceedings of HT2005, ASME Summer Heat Transfer Conference, San Francisco, California, USA, July 17-22 2005.
6. W. Shyy, S. Tong, S. Correa, 1985, *Numerical Recirculating Flow Calculation using a Body-Fitted Coordinate System*, Numerical Heat Transfer, Vol. 8, pp.99-113.
7. M. Vukić, 2004, *Experimental and numerical investigations thermal and fluid flow processes in shell and tube heat exchangers*, (In Serbian) PhD Thesis, Niš, Serbia.
8. Stevanović, Ž., Ilić, G., Radojković, N., Vukić, M., Stefanović, V., Vučković, G., 2001, *Design of Shell and Tube Heat Exchangers by using CFD Technique – Part one: Thermo-Hydraulic Calculation*, Facta Univesitatis Series Mechanical Engineering, 1(8) pp. 1091-1105.
9. M. Prithiviraj, M. J. Andrews, *Three-dimensional numerical simulation of shell-and-tube heat exchangers. part II: Heat transfer*, Numerical Heat Transfer, Part A: Applications: An International Journal of Computation and Methodology, Volume 33, Issue 8, 1998.
10. Yong-Gang Lei, Ya-Ling He, Rui Li, Ya-Fu Gao, *Effects of baffle inclination angle on flow and heat transfer of a heat exchanger with helical baffles*, Chemical Engineering and Processing: Process Intensification**,** Volume 47, Issue 12, 2008, pp 2336–2345.
11. Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao, *3D numerical simulation on shell-and-tube heat exchangers with middle-overlapped helical baffles and continuous baffles – Part I: Numerical model and results of whole heat exchanger with middle-overlapped helical baffles*, International Journal of Heat and Mass Transfer, Volume 52, Issues 23–24, 2009, pp 5371–5380.

3D NUMERIČKE SIMULACIJE PRENOSA TOPLOTE U DOBOŠASTOM IZMENJIVAČU TOPLOTE

Dobošasti izmenjivač toplote je jedan od najčešće korišćenih aparata u hemijskoj industriji. Glavni ciljevi proizvođača ovih aparata su povećanje njihove pouzdanosti i efikasnsoti u eksploataciji. Moguća su dva pristupa u cilju poboljšanja dizajna ovih aparata: eksperimentalno istraživanje koje je veoma skupo i dugotrajno, zbog složene geometrije međucevnog prostora aparata i numeričko istraživanje. Numeričkim simulacijama može se proveriti stari dizajn i usavršiti novi, efikasniji dizajn aparata. U ovom radu prikazani su rezultati numeričkog istraživanja termo-strujnih procesa u laboratorijskom dobošastom izmenjivaču toplote. Trodimenzionalne numeričke simulacije turbulentnih termo-strujnih procesa obavljene su korišćenjem softverskog paketa PHOENICS. Za modeliranje geometrije izmenjivača toplote korišćen je koncept poroznosti. Za modeliranje turbulentnih napona korišćen je standardni k-ε turbulentni model.

Ključne reči: *izmenjivač toplote, lokalni prenos toplote, numeričko istraživanje*