FACTA UNIVERSITATIS
Series: **Mechanical Engineering** Vol.1, N° 8, 2001, pp. 1091 - 1105

DESIGN OF SHELL-AND-TUBE HEAT EXCHANGERS BY USING CFD TECHNIQUE – PART ONE: THERMO-HYDRAULIC CALCULATION

UDC 621.039.534:517.538

Žarko Stevanović[1], Gradimir Ilić[2], Nenad Radojković[2], Mića Vukić[2], Velimir Stefanović[2], Goran Vučković[2]

[1]Department of Thermalmechanics, Mechanical Faculty University of Belgrade
[2]Faculty of Mechanical Engineering, University of Niš, FR Yugoslavia

Abstract. *In this paper, an iterative procedure for sizing shell-and-tube heat exchangers according to prescribed pressure drop is shown, then the thermo-hydraulic calculation and the geometric optimization for shell and tube heat exchangers on the basis of CFD technique have been carried out. Modeling of shell and tube heat exchangers for design and performance evaluation is now an established technique used in industry. In this paper, a numerical study of three-dimensional fluid flow and heat transfer in a shell and tube model heat exchanger is described. The baffle and tube bundle was modeled by the 'porous media' concept. Three turbulent models were used for the flow processe. The velocity and temperature distributions as well as the total heat transfer rate were calculated. The calculations were carried out using PHOENICS Version 3.3 code.*

Key words: *shell-and-tube heat exchanger, turbulence modeling, computational fluid dynamics, parametric design optimization*

1. INTRODUCTION

Often in thermotechnical and installations in chemical industry is necessary to replace the damaged or too fouled HE-s. In such situation pressure drops of process fluids are predetermined with available head of the pump aggregates. Also, it should be taken care of not to cause disbalance of the pressure drop in installation net by installing the new projected HE. In other words, we can say that pressure drop in HE is known in advance.

Iterative procedure is based on the fact that for the specified apparatus thermal power (apparatus thermal power is given by project task) exists a hyperbolic relation between overall heat transfer coefficient and heat transfer area:

Received February 17, 2002

$$k = \frac{C_k}{A} \text{ where } C_k = \frac{\dot{Q}}{\Delta t_{sr}} = \frac{\dot{Q}}{\varepsilon(R,P)\cdot \Delta t_{sr}} \tag{1.1}$$

Basic idea in this algorithm is to find such opperation point (k_i,A_i) by searching the area $k = f(A)$ in the interval $A_{min} \le A \le A_{max}$ where $A_{min} \to 0$, and $A_{max} \to \infty$,which corresponds to HE in which the pressure drops of process fluids are equal to the prescribed pressure drops [23], i.e.:

$$\Delta p_u = \Delta p_{u,pr} \le \Delta p_{u,all} \quad \Delta p_o = \Delta p_{o,pr} \le \Delta p_{o,all} \tag{1.2}$$

or that at least one of the pressure drops is equal to prescribed one.

Of course, it should be taken care about recommendations for flow velocities, pipe length and recommendations for geometrical characteristics of the space between pipes.

In this way, the whole range of dependence k=f(A) is practically investigated by this iterative procedure, for prescribed pressure drops.

Considering previous it is desirable to design HE in a way of complete usage of available pressure drop. This algorithm is based on modified both Polley's and Kern's procedure.

Relation between tubeside pressure drop and tubeside heat transfer coefficient as well as heat transfer area in the folowing form:

$$\Delta p_u = C_{pu1}\cdot A\cdot \alpha_u^{3,5} + C_{pu2}\cdot \alpha_u^{2,5} \tag{1.3}$$

and the average tubeside fluid velocity:

$$w_u = C_{wu}\cdot {\alpha_u}^{1,25} \tag{1.4}$$

where

$$C_{pu1} = \frac{12464,7}{\dot{m}_u}\cdot\frac{d_u^{1,5}}{d_s}\cdot\frac{\mu_u^{1,833}}{\rho_u\cdot\lambda_u^{2,333}\cdot c_{pu}^{1,167}}\cdot\left(\frac{\mu_u}{\mu_{uz}}\right)^{-0,63} \tag{1.5}$$

$$C_{pu2} = \frac{1}{2}\cdot\rho_u\cdot C_{lok}\cdot C_{wu}^2 \tag{1.6}$$

$$C_{wu} = 111,6\cdot d_u^{0,25}\cdot\frac{\mu_u^{0,583}}{\rho_u\cdot\lambda_u^{0,833}\cdot c_{pu}^{0,417}}\left(\frac{\mu_u}{\mu_{uz}}\right)^{-0,175} \tag{1.7}$$

where

$C_{lok} = 4,5\cdot n$ for HE with straight tubes, $C_{lok} = 2,5\cdot n$ for HE with U-tubes.

Relation between the shellside pressure drop and the shellside heat transfer coefficient as well as the heat transfer area:

$$\Delta p_o = C_{po}\cdot\frac{D_u^2}{N_c}\cdot A\cdot\alpha_o^{5,11} \tag{1.8}$$

and where the average shellside flow velocity is given by:

$$w_o = C_{wo} \cdot \alpha_o^{\ 1,818} \tag{1.9}$$

where

$$C_{po} = \frac{52,67 \cdot m}{\dot{m}_o} \cdot \frac{d_e^{1,11} \cdot (t - d_s)}{t \cdot d_s} \cdot \frac{\mu_o^{1,297}}{\rho_o \cdot \lambda_o^{3,406} \cdot c_{po}^{1,703}} \cdot \left(\frac{\mu_o}{\mu_{oz}} \right)^{-0,857} \tag{1.10}$$

$$C_{wo} = 6,41 \cdot d_e^{0,818} \cdot \frac{\mu_o^{0,394}}{\rho_o \cdot \lambda_o^{1,212} \cdot c_{p0}^{0,606}} \left(\frac{\mu_o}{\mu_{oz}} \right)^{-0,255} \tag{1.11}$$

The coefficients C_{pu1}, C_{pu2}, C_{wu}, C_{wo} and C_{p0} depend on tube diameter, tube pitch, number of passes of fluid through tubes and shell, thermo-physical properties and mass flow rates of fluids.

By presented procedure one can get the set of geometries which satisfy the thermo-hydraulic conditions of the project

Modeling of fluid flow and heat transfer phenomena in heat exchangers of arbitrary geometry is a complex process. Successful modeling of this process relies on quantifying the heat, mass and momentum transport phenomena. Generally speaking, Computational-Fluid- Dynamics (CFD) has been successfully used for predicting flow and heat transfer processes within complex geometry such as tube banks within heat exchangers. The advantages of this practice include diagnosis of flow, rapid evaluation of novel process route, and energy efficient and low cost design.

Heat exchangers have been extensively researched both experimentally and numerically [1-16]. However, most of the CFD simulation on heat exchangers was aimed at model validation. Also, very few applications can be found on using CFD technique as a tool for heat exchangers design optimization. Equally there have been very few reports on the effect of different turbulence models [18-20]. It is generally believed that heat exchanger modeling does not justify the use of turbulence models higher order of two, such as Reynolds stress model. But, it is still worth investigating the difference in the flow field and heat transfer coefficients resulting from the use of different turbulence models. This paper addresses these problems having two objectives. The first one is investigation the effect of different turbulence models on the velocity field and heat transfer coefficients (this is the objective of this paper). The second one is carrying out design optimization calculations, i.e. investigation the effects of baffle length and position on flow and heat transfer and choosing an optimal baffle size and position for the shell and tube heat exchanger considered (this will by the objective of a future paper).

2. Objective of work

By using the previously presented iterative procedure, based on prescribed pressure drop, the starting heat exchanger geometry (Fig.1) was determined, that satisfies thermo-hydraulic conditions. The heat exchanger contains 120 staggered arranged tubes of 1.91cm in pitch, 76.2cm in length and 1.27cm in outside diameter. The shell side fluid (combustion products) flows in cross flow direction. With a baffle in the middle along the tube length, the shell side fluid changes its flow direction by 180 degrees and then flows out. The cooling water flows inside the tubes. Computations were performed for Reynolds numbers up to 10000. Three different turbulence models were used to predict the turbulence velocity field. Three-dimensional simulation of the fluid flow and heat transfer processes was carried out.

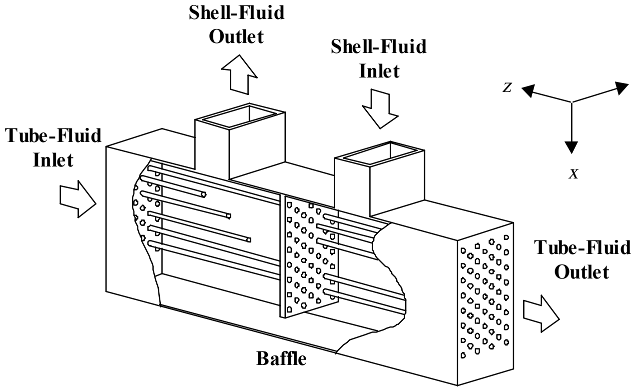


Fig. 1. Heat exchanger model scheme (not scaled)

3. Mathematical Model

3.1. Governing Differential Equations

The equations, which are supposed to govern the distributions of the shell-fluid averaged velocity components U, V and W in the three Cartesian coordinate directions x, y and z are:

$$\rho\frac{\partial U}{\partial t}+\rho U\frac{\partial U}{\partial x}+\rho V\frac{\partial U}{\partial y}+\rho W\frac{\partial U}{\partial z}=-\frac{\partial P}{\partial x}+\mu_{eff}\left(\frac{\partial^2 U}{\partial x^2}+\frac{\partial^2 U}{\partial y^2}+\frac{\partial^2 U}{\partial z^2}\right)-f_x \quad (3.1)$$

$$\rho\frac{\partial V}{\partial t}+\rho U\frac{\partial V}{\partial x}+\rho V\frac{\partial V}{\partial y}+\rho W\frac{\partial V}{\partial z}=-\frac{\partial P}{\partial y}+\mu_{eff}\left(\frac{\partial^2 V}{\partial x^2}+\frac{\partial^2 V}{\partial y^2}+\frac{\partial^2 V}{\partial z^2}\right)-f_y \quad (3.2)$$

$$\rho\frac{\partial W}{\partial t}+\rho U\frac{\partial W}{\partial x}+\rho V\frac{\partial W}{\partial y}+\rho W\frac{\partial W}{\partial z}=-\frac{\partial P}{\partial z}+\mu_{eff}\left(\frac{\partial^2 W}{\partial x^2}+\frac{\partial^2 W}{\partial y^2}+\frac{\partial^2 W}{\partial z^2}\right)-f_z \quad (3.3)$$

Here t stands for time, P for pressure, ρ for density and f_x , f_y , f_z for distributed-resistance coefficients. The latter may vary in an arbitrary manner with position, time and the magnitudes and direction of the velocities; but they are always greater then or equal to zero.

The mass-conservation principle for the shell-fluid provides another differential equation involving velocities, namely:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho U)}{\partial x}+\frac{\partial(\rho V)}{\partial y}+\frac{\partial(\rho W)}{\partial z}=0 \quad (3.4)$$

The temperature T^S of the shell-fluid obeys a differential equation that is similar form to equations (3.1) to (3.3), namely:

$$\rho C_P^S\left(\frac{\partial T^S}{\partial t}+U\frac{\partial T^S}{\partial x}+V\frac{\partial T^S}{\partial y}+W\frac{\partial T^S}{\partial z}\right)-\left(\frac{\partial P}{\partial t}+U\frac{\partial P}{\partial x}+V\frac{\partial P}{\partial y}+W\frac{\partial P}{\partial z}\right)=$$
$$=\lambda_S\left(\frac{\partial^2 T^S}{\partial x^2}+\frac{\partial^2 T^S}{\partial y^2}+\frac{\partial^2 T^S}{\partial z^2}\right)+\alpha_S(T^W-T^S)+ \tag{3.5}$$
$$+\mu_{eff}\left\{\begin{array}{l}2\left[\left(\frac{\partial U}{\partial x}\right)^2+\left(\frac{\partial V}{\partial y}\right)^2+\left(\frac{\partial W}{\partial z}\right)^2\right]+\\+\left[\frac{\partial W}{\partial y}+\frac{\partial V}{\partial z}\right]^2+\left[\frac{\partial W}{\partial x}+\frac{\partial U}{\partial z}\right]^2+\left[\frac{\partial U}{\partial y}+\frac{\partial V}{\partial x}\right]^2\end{array}\right\}$$

where, C_P^S is the specific heat et constant pressure; α_S is a 'volumetric heat transfer coefficient' (W/m^3K), representing the intensity with which heat can pass from the shell-fluid to the tube wall at temperature T^W; λ_S is the thermal conductivity. The tube-fluid temperature, T^T, obeys a similar equation; however, since tube-fluid flows only in one direction, with mass velocity g^T (kg/m^2s), say, the equation has only one convection term. It is:

$$\rho^T C_V^T\frac{\partial T^T}{\partial t}+C_P^T g^T\frac{\partial T^T}{\partial z}=\alpha_T(T^W-T^T) \tag{3.6}$$

Here ρ^T, C_V^T, C_P^T are the relevant properties of the tube-fluid, and the tubes are supposed to be aligned with the z direction. Finally, there is a differential equation for the variation with time of tube-wall temperature T^W. It is:

$$\rho^W C_V^W\frac{\partial T^W}{\partial t}=\alpha_S(T^S-T^W)+\alpha_T(T^T-T^W) \tag{3.7}$$

where ρ^W and C_V^W are the relevant properties of the tube-wall material.

There is an important remark to be made about all these equations: it is that the densities ρ, ρ^T, ρ^W are to be interpreted as the mass of shell-fluid, tube-fluid and solid wall-material, respectively, per unit heat-exchanger volume. Thus ρ is the density of the shell-fluid multiplied by the proportion of the shell volume that is not occupied by the tubes and their contents. The $\rho^{'s}$ can of course vary from place to place within the heat exchanger, to correspond with a non-uniform distribution of tubes. This practice is known as the 'porous media' concept.

3.2. Auxiliary Relations

It is clear how the densities are to be calculated, but what about $f^{'s}$ and $\alpha^{'s}$? For each of these, we need algebraic relationships connecting them with the prescribed geometry, the fluid properties, and the local velocity components. Dimensional analysis reveals that these relations must have the forms:

$$\frac{f_x L}{\rho U} = F_x\left(\rho \frac{UL}{\mu}, \frac{V}{U}, \frac{W}{U}\right) \tag{3.8}$$

with similar expressions involving f_y and f_z; and:

$$\frac{\alpha_S L}{\lambda} = F_S\left(\rho \frac{UL}{\mu}, \frac{V}{U}, \frac{W}{U}, \frac{C}{\lambda}\right) \tag{3.9}$$

with corresponding expression involving α_T. Here L stands for a local length dimension, for example the tube diameter, λ(W/mK) represents the thermal conductivity of the shell-side fluid, and μ its viscosity. The functions may well posses more arguments than are shown above; thus the tube-wall thickness and thermal conductivity will normally enter the function F_S in dimensionless form; and it will be the Reynolds and Prandtl numbers of the tube-side fluid which mainly influence the function F_T, from which α_T is obtained.

The functions F_x, F_S, etc. can be obtained from various fairly obvious sources. For example, if y is the direction normal to the tube axis, F_y can be deduced from experimental data on the pressure-drop performance of tube banks in cross-flow. Of course, it may be that data cannot be found for the particular tube arrangement in question, or for the influence of the velocity ratios U/V and W/V: then the F_y function must be supplied in part by way of a more detailed theoretical study, new experiments, or guesswork. The F_T and F_S functions are obtainable from published or newly developed correlation for heat transfer in tube banks and rod bundles. Ordinarily allowance must also be made for the heat-transfer resistance of the wall material and of layers of semi-insulating deposits.

In the present calculations, the concept of distributed hydraulic resistance of the tube bundle was used. It was first suggested by Patankar and Spalding [1] and then developed and used by several other users [3-16]. The local hydraulic flow resistance, f_x , f_y an f_z in the momentum equations (3.1-3.3) caused by the tube bundle, are related to the pressure loss coefficients ξ_x , ξ_y and ξ_z, by

$$f_x = \xi_x \rho U U_R \tag{3.10}$$

$$f_y = \xi_y \rho V U_R \tag{3.11}$$

$$f_z = \xi_z \rho W^2 \tag{3.12}$$

where $U_R = \sqrt{U^2 + V^2}$ stands for velocity vector magnitude, and the pressure loss coefficients are defined by:

$$\xi_x = 2\left(\frac{C_x}{p}\right)\left(\frac{p\beta}{p - D_h}\right)^2\left(\frac{1-\beta}{1-\beta_0}\right) \tag{3.13}$$

$$\xi_y = 2\left(\frac{C_y}{p}\right)\left(\frac{p\beta}{p - D_h}\right)^2\left(\frac{1-\beta}{1-\beta_0}\right) \tag{3.14}$$

$$\xi_z = \left(\frac{2C_y}{p}\right)\left(\frac{1-\beta}{1-\beta_0}\right) \tag{3.15}$$

where D_h is the hydraulic diameter for shell-side fluid; p stands for tube pitch, β and β_0 are local and overall volume porosity factor for shell-side fluid, respectively. Friction factors, C_x, C_y and C_z, are given as:

1) for cross flow:

$$\begin{aligned} C_x &= 0.619 \cdot Re_x^{-0.198}, & Re_x &< 8000 \\ &= 1.156 \cdot Re_x^{-0.2647}, & 8000 \le Re_x &< 2\times 10^5 \end{aligned} \tag{3.16}$$

$$\begin{aligned} C_y &= 0.619 \cdot Re_y^{-0.198}, & Re_y &< 8000 \\ &= 1.156 \cdot Re_y^{-0.2647}, & 8000 \le Re_y &< 2\times 10^5 \end{aligned} \tag{3.17}$$

2) for parallel flow:

$$\begin{aligned} C_z &= 31/Re_z, & Re_z &< 2250 \\ &= 0.131 \cdot Re_z^{-0.294}, & 2250 \le Re_z &< 25000 \\ &= 0.066 \cdot Re_z^{-0.227}, & Re_x &\ge 25000 \end{aligned} \tag{3.18}$$

where Reynolds number is related to the velocity components U, V, W and outer tube diameter.

Volumetric heat transfer coefficient α_S can be obtained from well-known correlation of Nusselt, Reynolds and Prandtl numbers, having general form:

$$Nu = C_1 C_2 Re^n Pr^m \tag{3.19}$$

were C_1, n and m are geometrical factors which vary according to the correlation used, and C_2 is the row correction factor. According to reference [21], for Re > 6.0 E + 3 and $Pr \ge 1.0$, recommended values are: $C_1 = 0.3363$, $C_2 = 0.96$, n = 0.6 and m = 0.4. Nusselt ($Nu = h_S D_0/\lambda_S$), Reynolds ($Re = \rho U_R D_0/\mu_1$) and Prandtl ($Pr = C_P^S \mu_1/\lambda_S$) numbers are related to the shell-fluid properties and tube outer diameter, D_0. Connection between volumetric heat transfer coefficient α_S (W/m^{3}K), and convection heat transfer coefficient h_S (W/m^{2}K) has to be deduced choosing a characteristic length of shell-side fluid. Obviously, it is tube outer diameter, so $\alpha_S = h_S/D_0$. Tube-side volumetric heat transfer coefficient α_T (W/m^{3}K) can be deduced according to Pethukov [17]:

$$Nu_T = \frac{(\zeta/8)\,Re\,Pr}{K + 12.7\cdot(\zeta/8)^{0.5}(Pr^{2/3}-1)} \tag{3.20}$$

where: $K = 1.07 + \frac{900}{Re} - \frac{0.63}{1+10Pr}$, and $\varsigma = (1.82\cdot\log Re - 1.64)^{-2}$ is the friction coefficient in turbulent flow of liquid in a smooth, round pipe. Similarly, Nusselt, Reynolds and Prandtl numbers in correlation (3.20) are related to the tube-side fluid properties. Connection between volumetric heat transfer coefficient α_T and convection heat transfer coefficient h_T is based on tube outer diameter, so $\alpha_T = h_T/D_0$. Equation (3.20) agrees with the most reliable experimental data on heat and mass transfer to an accuracy of ±5% [17]. It is true in the Reynolds number range $4\times10^3 < Re < 5\times10^6$ and in the Prandtl (or Schmidt) number range $0.5 < Pr < 10^6$.

3.3. Turbulence Models

Fluid flow is one of the important characteristics of a heat exchanger. It strongly affects the heat transfer process of a heat exchanger and its overall performance. However, modeling turbulent flow is complicated and time consuming. So, it is important to choose an economic and suitable turbulence model. In this investigation, three different turbulence models were used for the process in the heat exchanger. The turbulence models are as follows.

1) *Constant Turbulent Viscosity Model*

This means that the turbulent viscosity μ_t is a constant all over the heat exchanger. Previous researches used 20 to 40 times the value of the laminar viscosity μ_l. It is considered that a several-fold variation in μ_t has no significant effect on the results. Thus, μ_t was chosen to be 30 times the value of μ_l in the current calculations.

2) *Standard $k-\varepsilon$ Turbulence Model*

The $k-\varepsilon$ turbulence model proposed by Harlow and Nakayama (1968) is by far the most widely used two-equations eddy-viscosity turbulence model, mainly because the $\boldsymbol{\varepsilon}$ was long believed the require no extra terms near walls. The popularity of the model, and its wide use and testing, has thrown light on both its capabilities and its shortcomings, which are well documented in the practice. For high turbulent Reynolds numbers, the standard form of the $k-\varepsilon$ model may be summarized as follows:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho Uk)}{\partial x}+\frac{\partial(\rho Vk)}{\partial y}+\frac{\partial(\rho Wk)}{\partial z}-\left(\mu_l+\frac{\mu_t}{\sigma_k}\right)\left(\frac{\partial^2 k}{\partial x^2}+\frac{\partial^2 k}{\partial y^2}+\frac{\partial^2 k}{\partial z^2}\right)= \\ =\rho(\nu_t P_k-\varepsilon) \tag{3.21}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho U\varepsilon)}{\partial x}+\frac{\partial(\rho V\varepsilon)}{\partial y}+\frac{\partial(\rho W\varepsilon)}{\partial z}-\left(\mu_l+\frac{\mu_t}{\sigma_\varepsilon}\right)\left(\frac{\partial^2 \varepsilon}{\partial x^2}+\frac{\partial^2 \varepsilon}{\partial y^2}+\frac{\partial^2 \varepsilon}{\partial z^2}\right)= \\ =\rho\frac{\varepsilon}{k}(C_{\varepsilon1}\nu_t P_k-C_{\varepsilon2}\varepsilon) \tag{3.22}$$

where P_k is the volumetric production rate of turbulence kinetic energy k, by shear forces. It is calculated from:

$$P_k=2\left[\left(\frac{\partial U}{\partial x}\right)^2+\left(\frac{\partial V}{\partial y}\right)^2+\left(\frac{\partial W}{\partial z}\right)^2\right]+ \\ +\left(\frac{\partial U}{\partial y}+\frac{\partial V}{\partial x}\right)^2+\left(\frac{\partial U}{\partial z}+\frac{\partial W}{\partial x}\right)^2+\left(\frac{\partial W}{\partial y}+\frac{\partial V}{\partial z}\right)^2 \tag{3.23}$$

Turbulent viscosity is deduced by eddy-viscosity concept, from locally distributed turbulent kinetic energy k, and its dissipation rate **ε**:

$$\mu_t = \rho C_\mu \frac{k^{3/2}}{\varepsilon} \tag{3.24}$$

The following model constant are normally used:

$$\sigma_k = 1.0\,,\ \sigma_\varepsilon = 1.314\,,\ C_\mu = 0.09\,,\ C_{\varepsilon 1} = 1.44\,,\ C_{\varepsilon 2} = 1.92 \tag{3.25}$$

3) *Chen-Kim k-ε Turbulence Model*

The standard k-ε model employs a single time scale τ = k/ε to characterize the various dynamic processes occurring in turbulent flows. Accordingly, the source, sink and transport terms contained in the closed set of model equations (3.21 and 3.22) are held to proceed at rates proportional to k/ε. Turbulence, however, comprises fluctuating motions with a spectrum of time scales, and a single-scale approach is unlikely to be adequate under all circumstances because different turbulence interactions are associated with different parts of the spectrum. In order to ameliorate this deficiency in the standard model, Chen and Kim (1987) proposed a modification, which improves the dynamic response of the ε equation (3.22) by introducing an additional time scale $\tau_{CK} = k/P_k$. In addition, several of the standard model coefficients (3.25) are adjusted so that the model maintains good agreement with experimental data on classical turbulent shear layers. In view of its success for a number of high recirculated-flow calculations, this modification of **k – ε** model was used.

The modification involves dividing the **ε** production term (3.22) into two parts, the first of which is the same as the standard model but with a smaller multiplying coefficient, and the second of which allows the turbulence distortion ratio P_k/ε to exert an influence on the production rate of **ε**. According to the authors, the extra source term represents the energy transfer rate from large scale to small-scale turbulence controlled by the production-range time scale and the dissipation-range time scale. The net effect is to increase **ε**, and thereby decrease k, when the mean strain is strong $P_k/\varepsilon > 1$, and to decrease **ε** when the mean strain is weak $P_k/\varepsilon < 1$. The feature may be expected to offer advantages in recirculated flows and also in other flows where the turbulence is removed from local equilibrium. The modified k-**ε** model differs from the standard high-Reynolds-form of the k-**ε** model in that:

a) the following model constants take different values:

$$\sigma_k = 0.75\,,\ \sigma_\varepsilon = 1.15\,,\ C_\mu = 0.09\,,\ C_{\varepsilon 1} = 1.15\,,\ C_{\varepsilon 2} = 1.9 \tag{3.26}$$

b) an extra time scale is introduced in the **ε** equation (3.22) via the following additional source term per unit volume:

$$S_{CK} = \rho f_1 C_{\varepsilon 3} \frac{P_k^2}{k} \tag{3.27}$$

where $C_{\varepsilon 3} = 0.25$ and f_1 is the damping function, introduced for low-Reynolds number, which tends to unity at high turbulence Reynolds numbers.

4. COMPUTATIONAL DETAILS

A 32×10×70 non-uniform numerical grid (x, y, z - directions) was used. Previous researchers [2, 21] have shown that this grid-resolution can give grid independent results. Computations were carried out for a shell-fluid Reynolds number of $Re = 1.0\times10^4$ (based on the velocity and hydraulic diameter at entry of the heat exchanger), at 120°C inlet temperature. An inlet water temperature was 20°C and water mass velocity was $g_{in}^{T} = 5.0\ \text{kg/m}^2\text{s}$. Regarding the tube out diameter and cell dimensions, the local volume porosity factor $\beta = 0.653$ in the tube bundle region and $\beta = 0.588$ near the walls, were used. The overall volume porosity factor $\beta_0 = 0.653$ was used. The special Neighbor-Technique was used which produce sources along-the-tube convection fluid-to-metal heat transfer. Also, additional subroutines have developed in order to calculate distributed hydraulic resistances of the tube bundle and local volumetric heat transfer coefficients. The steady-state flow was considered, with constant densities.

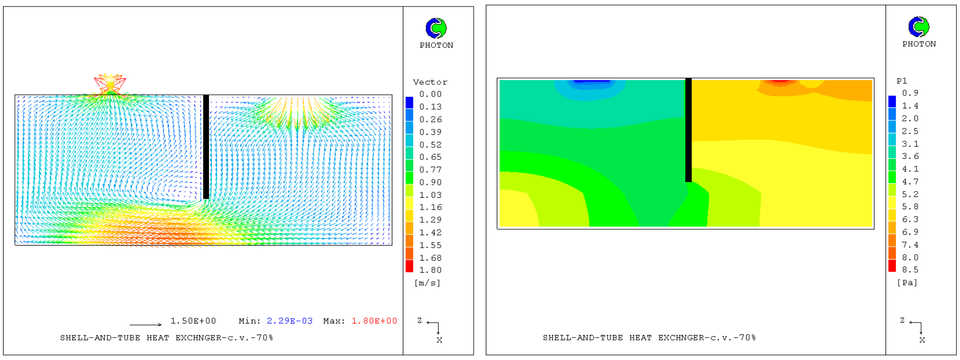
5. DISCUSSION OF RESULTS

Obviously, influence of different turbulence models to the governing equations 3.1-3.3, is introduced via effective viscosity, that is sum of laminar and turbulent part ($\nu_{eff} = \nu_l + \nu_t$). The eddy-viscosity ($\nu_t$) distribution is shown in Fig. 6, that is comparable to the constant value, introduced in the first turbulent model, $\nu_t = 4.355E-04[\text{m}^2/\text{s}]$.

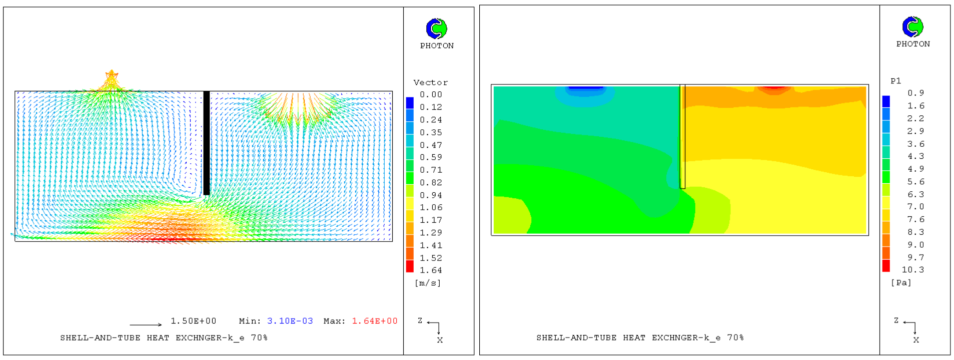
It can be seen from Fig. 6, that eddy-viscosity distribution in both turbulence models is similar, but with different magnitude, approximately by two orders. Furthermore, these values are much greater then constant value in the first turbulence model $\nu_t = 4.355E-04[\text{m}^2/\text{s}]$.

Because this parameter directly influences the viscous contribution of shear stresses in equations 3.1-3.3, it can be conclude that the third turbulence model is the most appropriate one. In the previous study [14], differences between the same turbulence models were not found. It should be expected because the viscous terms, in equations of motion 3.1-3.3, were omitted in these calculations. So in that case, the influence of eddy-viscosity was out of consideration. It is true that the influence of the wall-shear stresses are included via distributed resistance (3.10-3.12), but it is not reason to omitted the viscous terms in equations of motion, because they are influenced by turbulence via eddy-viscosity.

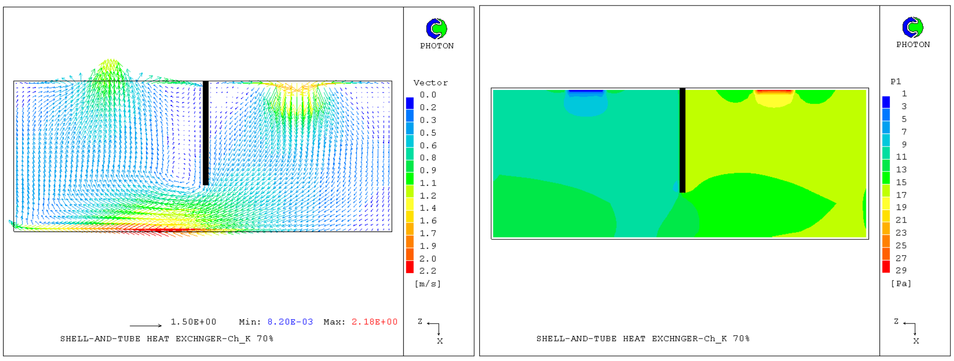
Additional parameter, that can help as to estimate the turbulence model validation is pressure distribution, shown in Fig. 3. As can be expected, the highest-pressure drop was obtained by the Chen-Kim modifications of $k-\varepsilon$ turbulence model. Certainly, it is the consequence of effective viscosity level. The consequences of different turbulence models to the shell and tube fluid temperature are shown in Fig. 4 and Fig.5, respectively. It can be seen that higher heat flow rate was obtained by calculations with the third turbulence model. Finally, it can be concluded that Chen-Kim modification of $k-\varepsilon$ turbulence model give the most accurate velocity field then turbulence model of constant eddy-viscosity or standard k – **ε** turbulence model



(a)



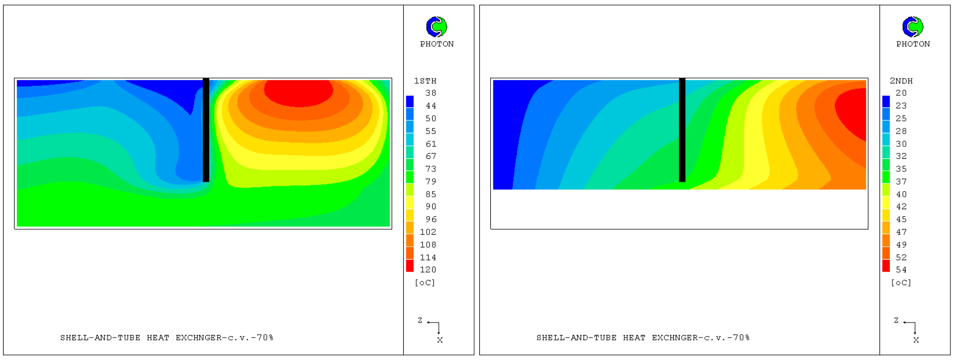
(b)



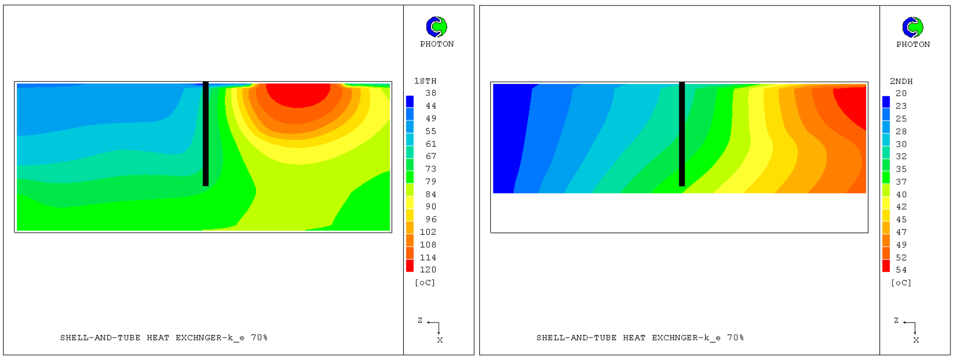
(c)

Fig. 2. Velocity distribution Fig. 3. Pressure distribution

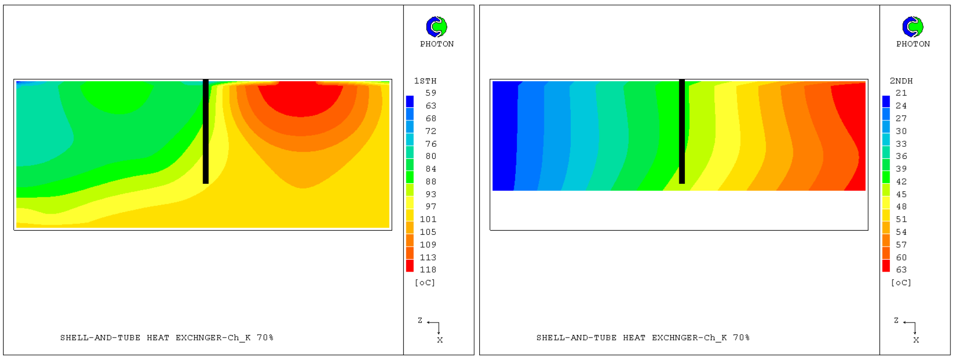
a) constant viscosity; b) standard k – **ε** model, c) k – **ε** - Chen-Kim model



(a)

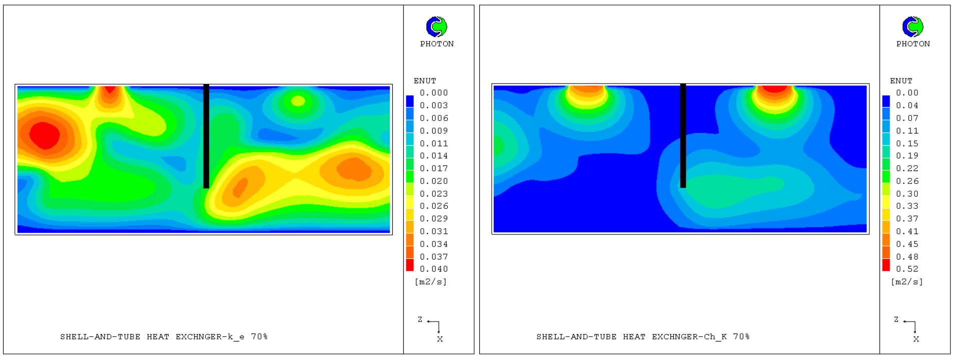


(b)



(c)

Fig 4. Shell-fluid temperature distribution Fig. 5. Tube-fluid temperature distribution
a) constant viscosity; b) standard $k-\varepsilon$ model, c) $k-\varepsilon$ - Chen-Kim model



(a) (b)

Fig. 6. Eddy-viscosity distribution
a) standard $k - \varepsilon$ model, b) $k - \varepsilon$ - Chen-Kim model

6. CONCLUSION

- A three-dimensional numerical simulation of fluid flow and heat transfer in a shell-and-tube heat exchanger has been carried out.
- The effect of different turbulence models on both flow and heat transfer is significant. This is due to the introduction effects of eddy-viscosity. It was concluded that Chen-Kim modification of the standard $k - \varepsilon$ turbulence model give the best agreement to the experimental data of velocity field.
- Flow process has an important effect to heat transfer. An optimal flow distribution can result in a higher heat transfer rate and lower pressure drop. Therefore, the optimization of flow distribution is an essential step in heat exchanger design optimization.
- CFD technique has demonstrated to have great potential in predicting the performance of both existing and newly developed that transfer equipment.
- Further improvements of heat transfer and fluid flow modeling are possible, specially in algebraic terms of local distributed resistance and volumetric heat transfer coefficients. But, it should be followed by more detailed experimental investigations and numerical simulations of locally flow field of tube bundle.

Acknowledgments. *This paper is a result of activities undertaken in the frame of a DAAD stability pact project under title: Development and Application of Numerical Methods for Calculation and Optimization of Pollutant Reduced Industrial Furnaces and Efficient Heat Exchangers. The authors gratefuly acknowledge the financial support.*

REFERENCES

1. Patankar, S.V. and Spalding, D.B., (1974), "A Calculation for the Transient and Steady-State Behavior of Shell-and-Tube Heat Exchangers", Heat Transfer Design and Theory Sourcebook, Editors: Afgan A.A. and Schluner E.U., Washington, pp. 155-176.
2. Theodossious, V.M., Sousa, A.C.M., (1988), "Flow Field Prediction in a Model Heat Exchanger", Computational Mechanics, Vol. 3, pp. 419-428.
3. Yang, S.R., Li, N., Zhao, H., Zhang, C.J., (1990), "On-Line fouling Monitoring Technique for Shell-Tube Heat Exchangers", Proc. 9th Heat Transfer Conf., Israel, pp. 223-227.
4. Diaper, A.D. and Heseler, L.E., (1990), "Crossflow Pressure Drop and Flow Distribution within a Tube Bundle Using Computational Fluid Dynamics", Proc. 9th Proc. 9th Heat Transfer Conf., Israel, pp. 235-240.
5. Zhang, C. and Sousa, A.C.M., (1990), "Numerical Simulation of Turbulent shear flow in an Isothermal Heat Exchanger Model", Transactions of the ASME Journal of Fluid Engineering, Vol. 112, pp. 48-55.
6. Haseler, L.E., Wadeker, V.V., Clarke, R.H., (1992), "Flow Distribution Effects in a Plate and Frame Heat Exchanger", IChemE Symposium Series, No. 129, pp. 361-367.
7. Revesz, Z., (1992), "Convergence Behavior of Three-Dimensional Two-Medium Finite Difference computation to Model Shell-and-Tube Heat Exchangers", Advance Computational Methods in Heat Transfer, Vol. 2., Nature/Forced Convection and Combustion Simulation, pp. 317-326.
8. Lorenzini, E., Orlandelli, C.M., Spiga, M., Tartarini, P., (1992), "Experimental and Theoretical Investigation of a Tube-Bundle Heat Exchanger", Institution Chemical Engineers Symposium Series, Vol. 1(129), pp. 459-465.
9. Prekdemir, T., Devies, T.W., Haseler, L.E., Diaper, A.D., (1993), "Flow Distribution on the Shell Side of a Cylindrical Shell-and-Tube Heat Exchanger", Int. Journal of Heat and Fluid Flow, Vol. 14, No. 1, pp. 76-85.
10. Zhang, C., Sousa, A.C.M., Venart, J.E.S., (1993), "The Numerical and Experimental Study of a Power Plant Condenser", Journal of Heat Transfer, Vol. 115, pp. 435-445.
11. Keene, L.W., Davies, T.W., Gibbsons, D.B., (1994), "Flow Distribution on the Shell-Side of a Shell-and-Tube Heat Exchanger", Transaction of IchemE, Vol. 71, part A, pp. 310-311.
12. Keene, L.W., Gibbsons, D.B., Evans, (1994), "Computer Modeling Validation for Shell-and-Tube Heat Exchangers", Transaction of IchemE, Vol. 72, part A, pp. 611-315.
13. Pekdemir, T., Davies, T.W., Heseler, L.E., Diaper, A.D., (1994), "Pressure Drop Measurements on the Shell-Side of a Cylindrical Shell-and-Tube Heat Exchanger", Heat Transfer Engineering, Vol. 15, No.3.
14. Huang, L.Y., Wen. J.X., Karayiannis, T.G., Mathewws, R.D., (1996), "CFD Modeling of Fluid Flow and Heat Transfer in a Shell and Tube Heat Exchanger", The PHOENICS Journal of CFD and its applications, Vol. 9. No. 2, pp. 181-209.
15. Nemoda, S., Stevanović, Ž., Turanjanin, V., Sladić, Z., (2000), "Ispitivanje sistema za hla|enje ulja na dizel lokomomotivi 641-300", Izveštaj NIV-ITE 168, Institut za nuklearne nauke - Vinča, (in Serbian).
16. Stevanović, Ž., Turanjanin, V., Popović, S., Nemoda, S., (2000) "Parametarska analiza uticaja geometrijskih karakteristika na toplotno strujne karakteristike dobošastog izmenjivača toplote ulje-voda za dizel lokomotivu 641", Izveštaj NIV-ITE 167, Institut za nuklearne nauke - Vinča, (in Serbian).
17. Petukhov, B.S., (1976), "Turbulent Heat Transfer in Tubes with Variable Fluid Properties", Heat Transfer Design and Theory Sourcebook, Editors: Afgan A.A. and Schluner E.U., Washington, pp. 239-263.
18. Harlow, F.H. and Nakayama, (1968), "Transport of Turbulence Energy Decay Rate", LA-3854, Los Alamos Science Lab., Univ. California, USA.
19. Launder, B.E. and Spalding, D.B., (1972), "Mathematical Models of Turbulence", Academic Press.
20. Chen, Y.S. and Kim, S.W., (1987), "Computation of Turbulent Flows Using an Extended k-ε Turbulence Closure Model", NASA CR-179204.
21. Rhodes, D.B. and Carlucci, L.N., (1983), "Predicted and Measured Velocity Distribution", International, Conference, on Numerical Methods in Nuclear Society, pp. 935-948.
22. Vukić, M., Ilić, G., Radojković, N., Stefanović, V.: "A New Approach to the Prediction and Design of Shell and Tube Heat Exchangers", Facta Universitatis, Niš, Vol. 1, No7, 2000, pp.775-787.

PROJEKTOVANJE DOBOŠASTIH IZMENJIVAČA TOPLOTE PRIMENOM CFD TEHNIKE - PRVI DEO: TERMO-HIDRAULIČKI PRORAČUN

Žarko Stevanović, Gradimir Ilić, Nenad Radojković, Mića Vukić, Velimir Stefanović, Goran Vučković

U ovom radu je prikazana jedna iterativna procedura za određivanje geometrije dobošastih izmenjivača toplote bazirana na pripisanom padu pritiska procesnih fluida, a zatim je primenom CFD tehnike izvršen termo-hidraulički proračun i optimizacija usvojene geometrije razmenjivača toplote. Dizajniranje razmenjivača toplote primenom CFD tehnike sve više se primenjuje u industriji. U ovom radu je razmatrano trodimenzionalno strujanje fluida i razmena toplote u dobošastom razmenjivaču toplote. Primenjen je koncept porozne sredine. Korišćena su tri turbulentna modela. Određena su brzinska i temperaturska polja procesnih fluida u omotaču i cevima razmenjivača toplote. Proračun je izvršen primenom PHOENICS-a 3.3.

Ključne reči: *dobošasti razmenjivač toplote, turbulentni model, CFD*