ANÁLISE NUMÉRICA DE UM TROCADOR DE CALOR DE CASCO E TUBOS: EFEITO DAS FOLGAS ENTRE OS TUBOS E OS ORIFÍCIOS DAS CHICANAS NA TRANSFERÊNCIA DE CALOR

Numerical analysis of a shell and tube heat exchanger: effect of gaps between tubes and baffle holes on heat transfer

Cristiano Vitorino da Silva[1,*]; Charles Poletto[2]; Marco Antônio Sampaio Ferraz de Sousa[3]; Alexandre Vagtinski de Paula[4]; Maria Luiza Sperb Indrusiak[5]

[1] Universidade Regional Integrada do Alto Uruguai e das Missões. Departamento de Engenharias e Ciência da Computação, Erechim, Brasil. *E-mail*: cristiano@uricer.edu.br

[2] Universidade Regional Integrada do Alto Uruguai e das Missões. Departamento de Engenharias e Ciência da Computação, Erechim, Brasil. *E-mail*: charles.poletto@gmail.com

[3] Universidade Regional Integrada do Alto Uruguai e das Missões. Departamento de Engenharias e Ciência da Computação, Erechim, Brasil. *E-mail*: marco@uricer.edu.br

[4] Universidade Federal do Rio Grande do Sul. Departamento de Engenharia Mecânica, Porto Alegre, Brasil. *E-mail:* depaula@ufrgs.br

[5] Independent Researcher. *E-mail*: msperbindrusiak@gmail.com

*Corresponding author: cristiano@uricer.edu.br

Data do recebimento: 11/01/2022 - Data do aceite: 26/05/2022

RESUMO: O presente estudo avalia, usando simulações numéricas em CFD, os efeitos da lacuna entre os orifícios das chicanas e o feixe de tubos no processo de transferência de calor em trocadores de calor de casco e tubos, especialmente em relação ao coeficiente convectivo médio no lado do casco. O fluxo de água no lado do casco foi simulado numericamente aplicando-se dois modelos de turbulência, ou seja, $k-\varepsilon$ *standard* e $k-\varepsilon$ EARSM (*Explicit Algebraic Reynolds Stress Models*). Os resultados obtidos em CFD para os valores dos coeficientes convectivos médios foram comparados com resultados da literatura e com resultados analíticos obtidos pelos métodos de Kern e Bell-Delaware, apresentando boa concordância quando as folgas entre os orifícios das chicanas e os tubos são nulas, principalmente para o caso simulado com o modelo de turbulência $k - \varepsilon$ EARSM, validando o modelo numérico implementado para avaliações térmicas de trocadores de calor casco e tubos. O estudo mostra que os resultados para os coeficientes médios de convecção no lado do casco obtidos pelos métodos analíticos mencionados apresentam boa concordância com resultados obtidos em CFD, com erros inferiores a 3 %, apenas quando as folgas entre os tubos e os orifícios das chicanas são nulas ou muito pequenas, menores que 0,2 mm. Contudo, para folgas relativamente maiores, acima de 1 mm, acabam superestimando o coeficnete médio de convecção em mais de 14 %, mesmo para o caso do método Bell-Delaware, que prevê fatores de correção para ajustar influência de tais folgas.

Palavras-chave: Transferência de calor. $k - \varepsilon$ *standard*. $k - \varepsilon$ EARSM. CFD. Metodos Kern e Bell-Delaware.

ABSTRACT: This study evaluates the effects of the gap between baffle holes and bundle tubes on the heat transfer process in shell and tube heat exchangers, especially regarding the shell mean convective coefficient, through CFD numerical simulations. The water flow on the side of the shell was numerically simulated applying two turbulence models, namely k–ε standard and k–ε EARSM (Explicit Algebraic Reynolds Stress Models). The results obtained in CFD for the values of the mean convective coefficients were compared with the results from the literature and analytical results obtained by the Kern and Bell-Delaware methods, showing good agreement when the clearances between the baffle holes and the tubes are zero, mainly for the simulated case with the turbulence model $k - \varepsilon$ EARSM, validating the numerical model implemented for thermal evaluations of shell and tubes heat exchangers. This study shows that the results for the mean convection coefficients on the shell side obtained by the aforementioned analytical methods show good agreement with CFD results, with errors of less than 3 %, only when the gaps between the tubes and baffle holes are zero or very small, smaller than 0.2 mm. However, for relatively larger gaps, above 1 mm, these analytical methods end up overestimating the mean convection coefficient by more than 14 %, even in the case of the Bell-Delaware method, which provides correction factors to adjust the influence of such gaps.

Keywords: Heat Transfer. $k - \varepsilon$ standard. $k - \varepsilon$ EARSM. CFD. Kern and Bell-Delaware Method.

Introduction

Heat transfer is a major process in industries such as energy, food, chemical, space applications, electronics, refrigeration, and air conditioning. The usual device for heat transfer processes is the heat exchanger and, due to its main role in several important applications, efforts on research and technology development have being done.

There are several design methodologies for heat exchangers, most of them standard oriented, e.g., TEMA – Tubular Exchanger Manufactures Association (TEMA, 1988) or ASME – American Society of Mechanical Engineers (ASME, 2015), which use generalized empirical correlations from experimental data to predict a bulk convection coefficient. However, improving the design and performance of heat exchangers is still a necessity, mainly due to the concern with the depletion of availability reserves, since these devices are responsible for thermal conversions in industrial processes. The main goal in these research is the enhancement of heat transfer. The main parameter in thermal analysis is the so-called global heat transfer coefficient, U, that summarizes the thermal resistances of the heat exchanger, mainly the convective ones. To obtain the global heat transfer coefficient, it is necessary to know the geometric parameters of equipment and, among other thermophysical parameters, the convective coefficients related to hot and cold streams. Various methodologies are available to compute these coefficients and computational fluid dynamics is nowadays the most promising one.

Numerical simulation results can be used to fit curves and analytical correlations to predict the behavior under operational conditions other than the simulated ones. Analytical correlations are also very useful in optimization processes. There are various papers about the subject. Among them, Zhang et al. (2009) presented a 3D simulation of a heat exchanger with superimposed helical baffles, using Fluent 6.3 code. The computational model and numerical method were detailed, and model validation was done by comparison with experimental data of total pressure drop and mean Nusselt number at shell side.

Ozden and Ilker (2010) analyzed numerically a small size shell and tube heat exchanger. The study evaluates the influence of baffle spacing, baffle cut and shell diameter over global heat transfer coefficient and pressure drop. Flow and temperature fields were obtained using ANSYS Fluent 6.33. The results were sensitive to the turbulence model selected for closure of the RANS (Reynolds Average Navier-Stokes) transport equations. To select the most appropriate turbulence model, a comparison was done among numerical results for convective heat transfer coefficient and the analytical results obtained through the Kern and Bell-Delaware methodologies (KAKAÇ; LIU, 2002). Simulation results showed good agreement with analytical ones.

You et al. (2012) developed a numerical model based on porosity and permeability concepts to study thermal and hydraulic performances at a heat exchanger shell side. To obtain results for comparison with the experimental data, a fixed Reynolds number was prescribed for the shell side flow. The work also shows that the applied model is economical and efficient in evaluating thermo-hydraulic performance of heat exchangers.

Chen et al. (2012) studied a novel concept of helical baffle shell-and-tube heat exchanger. A numerical model, using Fluent 6.3 and Gambit software, is applied to obtain the fluid flow profile. Results clearly show the secondary flow, a valuable mechanism to improve heat transfer. Results show also that flow leakages due to shortcuts at the triangles between adjacent baffles are small and that the flow pattern is almost a direct connection between primary and secondary flows.

In another work, Yang et al. (2014) applied four different numerical models to simulate a rod-baffle shell-and-tube heat exchanger. In two of the four methods, only a small subsection of the heat exchanger is modeled, in one a porous medium was used to model the heat exchanger and in the last one the heat exchanger is completely modeled with CFD. Periodic model, porous model and the whole model predicted accurately the heat transfer, while the unit model presented lower accuracy.

Pal et al. (2016) realized a numerical investigation in a short shell and tube type heat exchanger, with and without baffles in the shell side, using CFD code Open-FOAM-2.2.0 for different mass flow rates. They observed that the cross flow near the nozzle region has a significant influence on the heat transfer and, consequently, the conventional heat transfer correlations do not apply to short heat exchangers. They concluded also that the standard k–ε model gives best results for the velocity profile and heat transfer.

Maakoul et al. (2016) made three-dimensional CFD simulations, using the commercial software ANSYS Fluent, to study and compare shell-side flow distribution, heat transfer coefficient and pressure drop among heat exchangers with two different baffle types: the trefoil-hole helical baffles and the conventional segmental baffles. Good results were obtained in the prediction of thermo-hydraulic performance, showing that the trefoil-hole helical baffles present higher thermo-hydraulic and heat transfer performance but also large pressure drop compared to segmental baffles.

A study evaluating numerically baffle clearances was done by Leoni, Klein and Medronho (2017). They concluded that tube-baffle hole-gaps may reduce the heat transfer efficiency, but they also reduce recirculation and therefore, reduce the high temperature spots on stagnant zones. The study (LEONI; KLEIN; MEDRONHO, 2017) was done using the same turbulence models used in the present study and a similar approach. Nevertheless, the focus of the present article is the discussion about analytical design methods accuracy.

In Yang et al. (2017) the effect of flow maldistribution in a multi-channel heat exchanger was evaluated. A CFD model was used to obtain the fluid distribution in the core of heat exchanger, proving to be a good tool to solve problems of flow maldistribution.

There are many other recent experimental or analytical work related to the evaluation of shell and tube type heat exchangers, where the effect of different baffles or cut baffles can be observed (WEN et al., 2015; AFSAR et al., 2018; CHAUHAN; BAROT, 2018; PÉTRIK; SZEPESI, 2018; MOHANTY, 2020; AYDIN et al., 2020; GUPTA; VERNA, 2022), or related to cost-energy optimization (WEN et al., 2016), or analyses of the performance optimizations of parameters using CFD (HANAN et al., 2021). There are also several mathematical models for heat exchangers; some are simple, e.g., Kern method, some are more comprehensive and consider details of the flow, e.g., Bell Delaware (KAKAÇ; LIU, 2002). Nevertheless, in all methods, the mean convective coefficient is a considerably empirical factor, which depends on fluid properties, geometry, tube and baffle arrangement, surfaces roughness, recirculation, secondary flows and turbulence intensity. The gaps between tubes and baffles and shell and baffles are responsible for secondary flows that disturbs main flow and probably reduce convective heat transfer.

As mentioned, there are few data available in the literature concerning the effect of the baffle-tube gap on the convective heat transfer coefficient. This article presents a numerical analysis of the influence on heat transfer of the gap between baffle and tubes of a heat exchanger presented in the literature. The basis article did not address the gap issue. Fitness of Kern and Bell Delaware methods were evaluated for design situations where a gap is assigned.

Methods

Practical realizations of a shell and tube heat exchanger require a gap between baffle holes and tubes to allow tubes assembly and disassembly and to compensate thermal dilatations while in operation. Nevertheless, the gap is also a drawback because the flow induced vibrations may cause tube failure in the region of repeated contact with the baffles. Additionally, the flow bypass through the gaps affects the heat exchanger thermal performance. Considering the huge impact of thermal design of heat exchangers on plant performances, a numerical study was done to evaluate the role of flow by-pass through the gaps on the thermal performance of a shell and tube heat exchanger. The simulations were done considering three cases: zero gap, 0.2 mm and 1 mm gap.

The commercial code CFX v.14.5, Ansys@Inc was used for simulations. The results were compared with analytical ones obtained through two methods, Kern and Bell-Delaware (KAKAÇ; LIU, 2002), and with numerical data from Ozden and Ilker (2010). Two turbulence models were evaluated: $k-\varepsilon$ standard (LAUNDER; SPALDING, 1974) and $k-\varepsilon$ EARSM - Explicit Algebraic Reynolds Stress Models (WALLIN, 2000).

Physical Model

The physical reference for this work was a small shell and tube heat exchanger used in the work (OZDEN; ILKER, 2010), which was the reference for validation of the results. Figure 1 shows the physical domain set for the problem. Table I presents the parameters of the reference heat exchanger. The thermophysical properties of the working fluid, water, were obtained from the software libraries and from Incropera and DeWitt (2003).

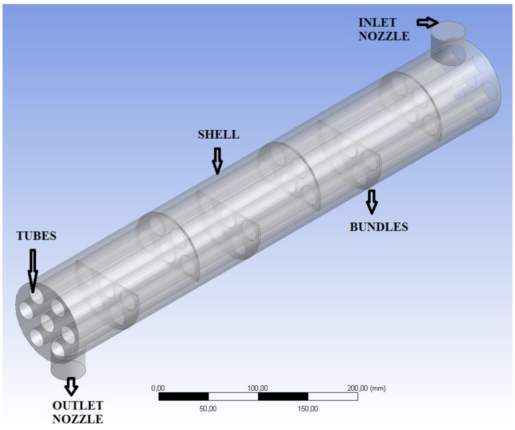
Mathematical model

Water, modeled as viscous and incompressible, was the working fluid. The continuum hypothesis was adopted. The steady state flow was modeled through RANS conservation equations. Ensemble averages were adopted for fluid properties (FREIRE; MENUT; SU, 2002). Two turbulence models were applied: $k-\varepsilon$ standard and $k-\varepsilon$ EARSM. The second one is an adaptation of the first and is more suitable to predict flow recirculation and boundary layer detachment (HELLSTEN, 2005). The wall-function approach adopted by Ansys CFX is an extension of the method of (LAUNDER; SPALDING, 1974). In the log-law region, the near wall tangential velocity is modeled by a logarithmic function of the wall shear stress, τ_ω.

Table I. Project parameters

Shell diameter	90 mm
Tube outer diameter	20 mm
Tube layout and pitch	Triangle, 30 mm
Number of tubes	7
Shell length	600 mm
Number of baffles and cut	6; 36 %
Distance between baffles	86 mm
Inlet and outlet tube diameter	38.1 mm

Figure 1. Physical domain



Mass conservation equation is done by:

$$\frac{\partial(\tilde{U}_j)}{\partial x_j} = 0 \qquad (1)$$

where x_j is the space coordinate and $\tilde{U}_j$ is theRANS mean velocity in the j-direction.

Momentum conservation equation is done by:

$$\frac{\partial}{\partial x_j}(\tilde{U}_i\tilde{U}_j) = -\frac{\partial p'}{\partial x_j}\delta_{ij} + \frac{\partial}{\partial x_j}\left(\mu_e\frac{\partial \tilde{U}_i}{\partial x_j}\right) + \frac{\partial^2 \tilde{U}}{\partial x_j \partial x_i} + S_u \qquad (2)$$

where effective viscosity μ_e is the sum of dynamic viscosity of the fluid, μ, and turbulent viscosity of the flow, $\mu_t = C_\mu \rho k^2/\varepsilon$. Terms k and ε are respectively the turbulent kinetic energy and its dissipation of $k-\varepsilon$ turbulence model. The term $p' = \overline{p} - (2/3)k$ is the modified pressure, being $\overline{p}$ the Reynolds average pressure of the fluid. The term δ_{ij} is the Krönecker delta function and S_u is a generic source/sink used to deal with the additional mathematical turbulent terms.

Energy conservation equation considers the energy transport due to advection and diffusion:

$$\frac{\partial}{\partial x_j}(\tilde{U}_j\tilde{h}) = -\frac{\partial}{\partial x_j}\left(\left(\frac{\lambda}{c_p} + \frac{\mu_t}{Pr_t}\right)\frac{\partial \tilde{h}}{\partial x_j}\right) + S_h \qquad (3)$$

where $\tilde{h}$, c_p and λ are respectively average enthalpy, specific heat at constant pressure and thermal conductivity of the fluid, Pr_t is the turbulent Prandtl number and S_h is a generic source/sink therm.

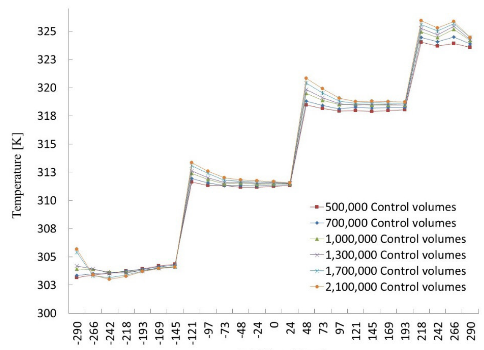
Boundary conditions

At heat exchanger inlet, mass flux and temperature were prescribed at respectively 1 kg/s and 300 K and the average turbulence intensity considered was 5 %. The no-slip assumption was assigned to all solid surfaces. Adiabatic boundary conditions were used at shell walls. The prescribed temperature at tube surfaces was 450 K, considering an isobaric condensation process inside the tubes.

Computational Mesh

An unstructured mesh grid with approximately 1.7×10^6 volumes was selected after a mesh independence test, which evaluated seven meshes with respectively 0.5, 0.7, 1.0, 1.3, 1.7 and 2.1 million volumes. Figure 2 presents the temperature profiles for each mesh, obtained at a reference axial line. The discontinuities in the profiles correspond to baffle crossing points.

Figure 2. Comparison of temperature profiles for mesh grids evaluated

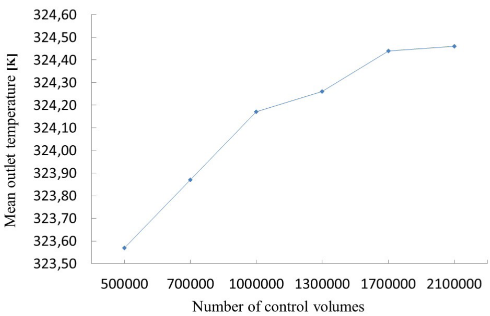


According to the results, as the number of volumes is increased, there is a decrease in temperature values. Figure 3 shows the evolution of temperature values with the number of mesh volumes for the point at 290 mm on the reference axial line. There is a clear asymptotic temperature tendency as the number of mesh volumes increases. The unstructured mesh comprises tetrahedral and prismatic volumes, the latter applied near to

PERSPECTIVA, Erechim. v. 46, n. 174, p. 43-55, junho/2022 / DOI:10.31512/persp.v.46.n.174.2022.229.p.43-55

solid surfaces to better predict the behavior of flow and heat transfer in boundary layer. Refinements are applied to the domain regions with smaller dimensions, the maximum volume size is 2 mm.

Figure 3. Evolution of temperature values with the number of mesh volumes



Numerical Model

The commercial code Ansys CFX 14.5, based on Finite Volumes Method of Patankar (PATANKAR, 1980) was used for simulations. The up-wind interpolation scheme was adopted to compute the fluxes on volume surfaces. Pressure-velocity coupling was solved using SIMPLE algorithm (PATANKAR, 1980). A sub-relaxation was adopted to deal with the strong coupling and non-linear behavior of conservation equations. Root Mean Square – RMS was the convergence criterion adopted with residual values lower than 1×10^{-5}. A workstation with eight parallel processing units was used for this work.

Results

Verification

For verification, present work considered the same heat exchanger working conditions and geometry of Ozden and Ilker (2010), with zero gap between baffle-holes and tubes. The baffle thickness was not analyzed. Boundary conditions were the same, but a different mesh was used, with 1.7×10^6 volumes. In both works turbulence model, $k - \varepsilon$ standard, and tetrahedral meshes with prismatic volumes at solid boundaries were adopted. Nevertheless, details were omitted in Ozden and Ilker (2010) and there is no mention of mesh refinements.

Table II shows the results from Ozden and Ilker (2010) and of present work for water outlet temperature and average shell side convective coefficient. Comparison shows coherence among results of the two works. The 16.62 % discrepancy in simulation results for the convective coefficient is imputed to the probable use of different mesh refinements at walls and small regions. Ozden and Ilker (4) omitted those details. The discrepancies with the analytical results according to Kern and Bell-Delaware of respectively 0.36 % and 7.11 % are possibly due to different sources for thermophysical properties adopted in this work and that of Ozden and Ilker (2010). Nevertheless, the results of present work are closer to analytical ones, leading to the conclusion that mesh and thermophysical properties applied at the present work are more adequate than the ones applied by Ozden and Ilker (2010).

Turbulence models evaluation

Results for zero gap from Ozden and Ilker (2010), for outlet temperature and mean convective coefficient, as well as relative errors between analytical (Kern and Bell-Delaware) and numerical results are presented in Table III. The relative errors are much larger for the $k - \varepsilon$ standard and $k - \varepsilon$ standard 2nd order turbulence models than for $k - \varepsilon$ realizable, evidencing that turbulence models play an important role in numerical simulation of thermal processes like the present one. Analytical results were obtained with fluid properties taken at mean temperature. Because outlet temperature is a result of the simulation, the analytical results also differ for each model.

These results, also shown in Table III, present a small relative error between $k - \varepsilon$ standard and $k - \varepsilon$ EARSM turbulence models. In addition, relative errors between numerical results using $k - \varepsilon$ standard and analytical ones, are smaller than that of Ozden and Ilker (2010). Numerical results show better correspondence with results obtained through Bell-Delaware method, and the present results show the lowest relative errors.

Evaluation of gap influence

To evaluate the gap influence on heat exchanger flow and thermal performance, two different gap values, of 0.2 and 1.0 mm, were assigned and the simulations were realized with all other conditions unchanged. Considering that $k - \varepsilon$ EARSM turbulence model showed best results in the previous simulations, it was adopted from now on. Table IV shows outlet temperatures and convective coefficients obtained for zero gap and for the two gap values simulated. As expected, the

Table II. Results by Ozden and Ilker (2010), present work and Kern and Bell-Delaware methods

	CFD results		Analytical results	
			Kern	Bell-Delaware
Reference works (k - ε standard)	Outlet temperature (K)	Convective coefficient (W/(m² K))	Convective coefficient (W/(m² K))	Convective coefficient (W/(m² K))
Ozden and Ilker (2010)	326	3561	3063	3276
Present work	324	2969	3074	3043
Relative error	**4.26 %**	**16.62 %**	**0.36 %**	**7.11 %**

Table III. Results by Ozden and Ilker (2010) and results from present work for zero gap

		CFD results		Analytical results			
				Kern		Bell-Delaware	
	Turbulence model	**Outlet temperature (K)**	**Convective coefficient (W/(m² K))**	**Convective coefficient (W/(m² K))**	**Relative error**	**Convective coefficient (W/(m² K))**	**Relative error**
Ozden and Ilker (2010)	$k-\varepsilon$ standard	326	3561	3063	**13.98 %**	3276	**8 %**
	$k-\varepsilon$ standard 2nd order	325	3547	3058	**13.79 %**	3267	**7.89 %**
	$k-\varepsilon$ realizable	327	3348	3072	**8.24 %**	3290	**1.73 %**
Present work	$k-\varepsilon$ standard	324	2969	3074	**3.41 %**	3043	**2.43 %**
	$k-\varepsilon$ EARSM	325	3088	3078	**0.35 %**	3052	**1.19 %**
	Relative error	**0.27 %**	**3.87 %**	---	---	---	---

Table IV. Results of present work: zero gap, 0.2 and 1.0 mm gaps

Turbulence model and gap	CFD results		Analytical results			
			Kern		Bell-Delaware	
	Outlet temperature (K)	Convective coefficient (W/(m² K))	Convective coefficient (W/(m² K))	Relative error	Convective coefficient (W/(m² K))	Relative error
$k-\varepsilon$ EARSM zero gap	325	3088	3078	**-0.35 %**	3052	**-1.19 %**
$k-\varepsilon$ EARSM gap 0.2 mm	325	2999	3077	**+2.52 %**	3049	**+1.63 %**
$k-\varepsilon$ EARSM gap 1.0 mm	322	2586	3060	**+15.46 %**	3017	**+14.25 %**

outlet temperature reduces, although slightly, when a gap is assigned, being smaller for the larger gap. The convective coefficient experienced a reduction in relation to zero gap case in 3 % and 19 % for, respectively for 0.2 and 1.0 mm gaps, demonstrating that the gap influence is important for the evaluation of the thermal performance of a heat exchanger.

Comparison of results from simulations and analytical ones showed that the larger the gap, the greater the relative error. The results for the mean convection coefficients on the shell side obtained by the analytical methods show good agreement, with errors of less than 3 %, only when the gaps between the tubes and baffle holes are zero or very small, smaller than 0.2 mm. However, for relatively larger gaps, above 1 mm, these analytical methods end up overestimating the mean convection coefficient by more than 14 %, even in the case of the Bell-Delaware method, which provides correction factors to adjust the influence of such gaps. This important result shows that the analytical methods do not properly consider the gap and, unless for very thin gaps, the values obtained for thermal parameters will be overestimated. It is worthy to note that TEMA and ASME standards do not consider or even mention the gap influence.

Figures 4 to 6 depicts the streamlines of the shell side flow along with local temperatures under the streamlines for the three cases simulated with the $k-\varepsilon$ EARSM turbulence model and presented at Table IV. The figures show that flow and temperature evolutions are affected by the gap value. The larger the gaps, the more fluid deviates through them and do not follow main flow. Therefore, the velocity field changes, and other flow characteristics change as well. The partial flow by-pass through the gaps, as showed in detail in Figure 6, changes Reynolds and Nusselt numbers, mean convective coefficient and, therefore, heat transfer rate. The numerical simulation can consider detailed features of the flow that depend on the geometry characteristics and, consequently, to compute more actual parameters of the phenomena.

Figure 7 presents the flow field at a normal plane to tube bundle axis at the middle point, for the two turbulence models used in present work. The $k-\varepsilon$ EARSM model show higher temperatures on the upper side than the $k-\varepsilon$ standard. This is due to the better capability of EARSM model in capture vorticity and secondary flows. As consequence, the flow is adequately represented at the upper side with a lower mass flux and higher vorticity, favoring higher temperatures.

Figure 4. Temperature in streamlines for zero gap. Turbulence model $k - \varepsilon$ EARSM

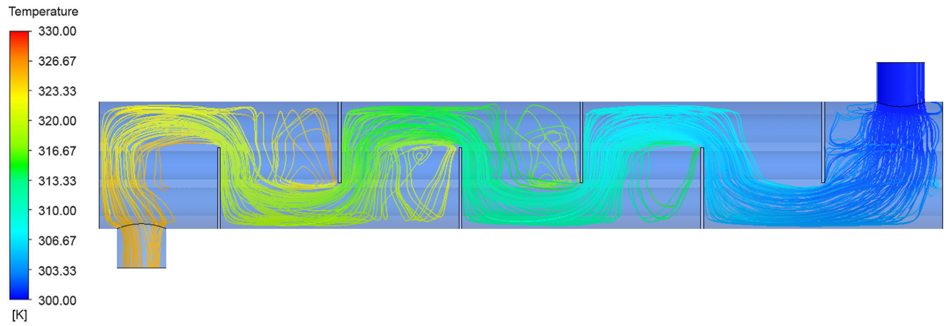


Figure 5. Temperature in streamlines for 0.2 mm gap. Turbulence model $k - \varepsilon$ EARSM

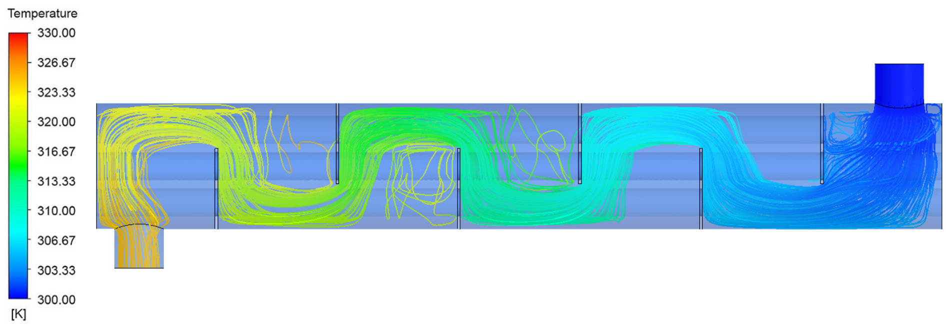


Figure 6. Temperature in streamlines for 1.0 mm gap. Turbulence model $k - \varepsilon$ EARSM

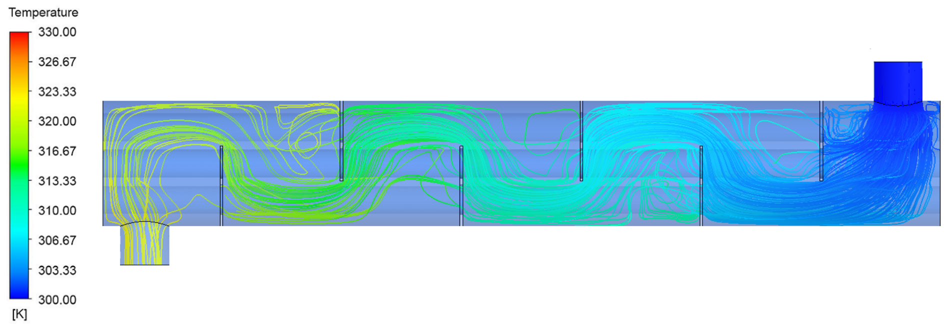
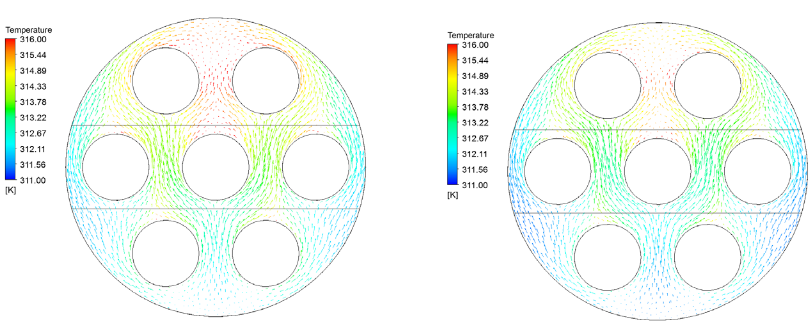


Figure 7. Flow vectors and temperature field at a normal plane to tube bundle axis at the middle point: (a) $k - \varepsilon$ standard; (b) $k - \varepsilon$ EARSM



(a) (b)

Conclusions

This work performed an evaluation of the influence of the gap between baffle holes and tubes on the thermal performance of a shell and tube heat exchanger. The main control parameter was the mean convective coefficient. A numerical analysis was conducted concerning the water flow at shell side of a small heat exchanger, assigning values of zero, 0.2 and 1.0 mm for the gaps between baffle holes and bundle tubes. The results obtained were compared with the analytical ones obtained through Kern and Bell-Delaware empirical correlations prescribed by TEMA and ASME standards. Two turbulence models were applied: $k-\varepsilon$ standard and $k-\varepsilon$ EARSM. Numerical results obtained in this work presented good agreement with those of literature and with analytical results for zero gap, mainly when used the $k-\varepsilon$ EARSM turbulence model. The results shows that the mean convection coefficients on the shell side obtained by the analytical methods of Kern and Bell Delaware agree with CFD results, with errors of less than 3 %, only when the gaps between the tubes and baffle holes are zero or very small, smaller than 0.2 mm. However, for relatively larger gaps, above 1 mm, these analytical methods end up overestimating the mean convection coefficient by more than 14 %, even in the case of the Bell-Delaware method, which provides correction factors to adjust the influence of such gaps.

Nomenclature

Symbol	Description	Units	Symbol	Description	Units
k	turbulent kinetics energy	[m²/s²]	$\overline{p}$	Mean pressure	[N/m²]
ε	turbulent kinetics energy dissipation	[m²/s³]	p'	Modified pressure	[N/m²]
τ_ω	wall shear stress	[N/m²]	δ	Krönecker delta function	[-]
$\tilde{U}$	mean velocity	[m/s]	S_u	generic source/sink therm	[(kg/m²s²)/m³]
x	space coordinate	[m]	$\tilde{h}$	Mean enthalpy	[kJ/kg]
μ_e	effective viscosity	[N×s/m²]	c_p	specific heat	[kJ/(kg×K)]
μ	fluid viscosity	[N×s/m²]	λ	thermal conductivity	[W/(m×K)]
μ_t	Turbulent viscosity	[N×s/m²]	S_h	generic source/sink therm	[W/m³]
C_μ	Constant	[-]	ρ	Density	[kg/m³]
Pr_t	turbulent Prandtl number	[-]			

REFERENCES

AFSAR, A.; INAM, M.I. CFD analysis of shell and tube heat exchanger with different baffle orientation and baffle cut. *In*: AIP Conference Proceedings 1980, **Proceedings** […], 2018. Disponível em: https://aip.scitation.org/doi/abs/10.1063/1.5044342, 2018. Acesso em: 30 abr. 2022.

AYDIN, A.; YASAR, H.; ENGIN, T.; BÜYÜKKAYA, E. Optimization and CFD analysis of a shell-and-tube heat exchanger with a multi segmental baffle. **Thermal Science**, v. 26, Issue 1, Part A, p. 1-12, 2022.

ASME - **Boiler and Pressure Vessel Code**. 2015.

CHAUHAN, J.V.; BAROT, J.M. CFD Analysis of Shell and Tube Heat Exchanger by Changing Baffle Arrangement. **IJRTI - International Journal for Research Trends and Innovation**, v. 3, n .4, p. 145-151, 2018.

CHEN, Y.P.; SHENG, Y.J.; DONG, C.; WU, J. F. Numerical Simulation on Flow Field in Circumferential Overlap Trisection Helical Baffle Heat Exchanger. **Appl. Therm. Eng**., v. 50, p. 1035-1043, 2013.

FREIRE, A. P. S.; MENUT, P. P. M.; SU, J. **Turbulência**, v. 1, ABCM. Rio de Janeiro, Brazil. 2002.

GUPTA, S; VERNA, K.S. Computational fluid dynamics analysis of shell and tube heat exchager with having different types of fins. **IJRTI - International Journal for Research Trends and Innovation**, v. 7, issue 2, p. 15-27, 2022.

HANAN, A.; ZAHIL, U.; FEROSE, T.; KHAN, S.Z. Analysis of the performance optimisation parameters of shell and tube heat exchanger using CFD. **Australian Journal of Mechanical Engineering**. Accepted 05 Apr 2021, Published online: 26 Apr. 2021.

HELLSTEN, A. New advanced k-ω turbulence model for high-lift aerodynamics. **AIAA Journal**. Vol. 43, p. 1857-1869, 2005.

INCROPERA, F.; Dewitt, D. **Fundamentos de transferência de calor e de massa**. LTC. Rio de Janeiro, Brazil. 2003.

KAKAÇ, S.; LIU, H. **Heat exchangers**: Selection, rating, and thermal design. Boca Raton, CRC 2002.

LAUNDER, B. E.; SPALDING, D. B. The numerical computations of turbulent flows. **Comp. Method. Appl. Mech. Eng**., v. 3, p. 269-289, 1974.

LEONI, G. B.; KLEIN, T. S.; MEDRONHO, R. A. Assessment with CFD of the effects of baffle clearance on the shell side flow in a shell and tube heat exchanger. **Appl. Therm. Eng**., v. 112, p. 497-506, 2017.

MAAKOUL, A. E.; LAKINIZI, A.; SAADEDDINE, S.; METOUI, M. E.; ZAITE, A.; MEZIANE, M.; ABDELLAH, A. B.. Numerical comparison of shell-side performance for shell and tube heat exchangers with trefoil-hole, helical and segmental baffles, **Appl. Therm. Eng**., v. 109, p. 175-185, 2016.

MOHANTY, S. CFD Analysis of a Shell and Tube Heat Exchanger with Single Segmental Baffles. **International Journal of Automotive and Mechanical Engineering**, v. 17, n. 2, p. 7890-7901, 2020.

OZDEN, E.; ILKER, T.. Shell Side CFD Analysis of a Small Shell-And-Tube Heat Exchanger. **Energy Conversion and Management**, v. 51, p. 1004-1014, 2010.

PAL, E.; KUMAR, I.; JOSHI, J. B.; MAHESHWARI, N. K. CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles. **Chem. Eng. Sci**., v. 143, p. 314-340, 2016.

PATANKAR, S. V. **Numerical Heat Transfer and Fluid Flow**. Washington, DC: Hemisphere, 1980.

PÉTRIK, M.; SZEPESI, G.L. Shell Side CFD Analysis of a Model Shell-and-Tube Heat Exchanger. **Chemical Engineering Transactions**, v. 70, 2018.

TEMA - **Standards of Tubular Exchanger Manufacturers Association**, 7.th Edition, New York, 1988.

WALLIN, S.; JOHANSSON, A. A complete explicit algebraic Reynolds stress model for incompressible and compressible flows. **J. Fluid Mech**, v. 403, p. 89-132, 2000.

WEN, J.; YANG, H.; WANG, S.; XUE, Y.; TONG, X. Experimental investigation on performance comparison for shell-and-tube heat exchangers with different baffles, **Int. J. of Heat and Mass Transfer**, v. 84, p. 990-997, 2015.

WEN, J.; YANG, H.; JIAN, G. P.; TONG, X.; LI, K.; WANG, S. M. Energy and cost optimization of shell and tube heat exchanger with helical baffles using Kriging metamodel based on MOGA, **Int. J. Heat and Mass Transfer**, v. 98, p. 29-39, 2016.

YANG, H.; WEN, J.; GU, X.; LIU, Y.; WANG, S.; CAI, W.; LI, Y. A Mathematical Model for Flow Maldistribution Study in a Parallel Plate-fin Heat Exchanger. **Appl. Therm. Eng**., v. 121, p. 462-472, 2017.

YANG, J.; MA, L.; BOCK, J.; JACOBI, A. M.; LIU, W. A comparison of four numerical modeling approaches for enhanced shell-and-tube, **Appl. Therm. Eng**., v. 66, p. 369-383, 2014.

YOU, Y.; FAN, A.; HUANG, S.; LIU, W. Numerical modeling and experimental validation of heat transfer and flow resistance on the shell side of a shell-and-tube heat exchanger with flower baffles. **Int. J. Heat and Mass Transfer**., v. 55, p. 7561-7569, 2012.

ZHANG, J.F.; HE, Y.L.; TAO, W.Q. 3D numerical simulation on shell-and-tube heat exchangers with middle-overlapped helical baffles and continuous baffles. **Int. J. Heat and Mass Transfer**., v. 52, p. 5381-5389, 2009.