[Lioua Kolsi, 5(1): January, 2016]

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

3D NUMERICAL ANALYSIS OF A SHELL-AND-TUBE HEAT EXCHANGER

Lioua Kolsi [1,2,*]

College of Engineering , Mechanical Engineering Department, Haïl University, Haïl City , Saudi Arabia

Unité de Recherche de Métrologie et des Systèmes Énergétiques, Ecole Nationale d'Ingénieurs, 5000 Monastir, University of Monastir, Tunisia

ABSTRACT

In this paper, a three-dimensional mathematical model is developed and implemented. The turbulent numerical model solves a conjugated heat transfer problem in a shell-and-tube heat exchanger for different inlet conditions. Results indicate the existence of a zigzag flow pattern and that the increase of the air velocity (sell-side) increases the heat transfer coefficient and the presseur drop and decreases the air outlet temperature.

KEYWORDS: shell-and-tube heat exchanger, numerical simulation, 3D

NOMENCLATURE:

A_o	heat exchange area based on the outer diameter (m²)
C_p	Specific heat of Copper (J/kg.K)
C_u, C_2, C_3	turbulent model constant
d	Diameter (m)
h	Heat transfer coefficient ($W.m^{-2}.K$)
K	Turbulent kinetic energy
N_t	number of tube rows
T	Temperature (K)
u	Velocity (m/s)
u_i	velocity component in i-direction, m/s
Latin letters	
Φ	heat transfer rate, (W)
ε	turbulent energy dissipation
λ	thermal conductivity, W $(m\ K)^{-1}$
μ	dynamic viscosity of fluid, Pa s
ρ	density of fluid, kg m^{-3}
Subscripts	
t	tube
s	shell
in	inlet
out	outlet
w	wall

INTRODUCTION

Heat exchangers are devices in which heat is transfer from one fluid to another. The most commonly used type of heat exchanger is a shell-and-tube heat exchanger. Shell-and-tube heat exchangers are used extensively in engineering applications like power generations, refrigeration and air-conditioning, petrochemical industries etc. These heat exchangers can be designed for almost any capacity. The main purpose in the heat exchanger design is given task for heat transfer measurement to govern the overall cost of the heat exchanger.

The heat exchanger was introduced in the early 1900s to execute the needs in power plants for large heat exchanger surfaces as condensers and feed water heaters capable of operating under relatively high pressures. Both of these original applications of shell-and-tube heat exchangers continued to be used; but the design have become highly sophisticated and specialized, subject to various specific codes and practices. The broad industrial use of shell-and-tube heat exchangers known today also started in the 1900s to accommodate the demands of emerging oil industry. [1]

There are three principle means of achieving heat transfer, conduction, convection, and radiation. Heat exchangers run on the principles of convective and conductive heat transfer. Radiation does occur in any process. However, in most heat exchangers the amount of contribution from radiation is miniscule in comparison to that of convection and conduction. Conduction occurs as the heat from the hot fluid passes through the inner pipe wall. To maximize the heat transfer, the inner-pipe wall should be thin and very conductive. However, the biggest contribution to heat transfer is made through convection [1].

There are two forms of convection; these are natural and forced convection. Natural convection is based on the driving force of density, which is a slight function of temperature. As the temperature of most fluids is increased, the density decreases slightly. Hot fluids therefore have a tendency to rise, displacing the colder fluid surrounding it. This creates the natural "convection currents" which drive everything from the weather to boiling water on the stove. Forced convection uses a driving force based on an outside source such as gravity, pumps, or fans. Forced convection is much more efficient, as forced convection flows are often turbulent. Turbulent flows undergo a great deal of mixing which allow the heat to be transferred more quickly [1].

The steadily increasing use of shell-and-tube heat exchangers and greater demands on accuracy of performance prediction for a growing variety of process conditions resulted in the explosion of research activities. These included not only shell side flow but also, equally important, calculations of true mean temperature difference and strength calculations of construction elements, in particular tube sheets [2].

There are various experimental research works available in the literature which analyses the performance of heat exchangers. However, it is observed that the experimental analysis of heat exchangers is difficult and costly. Also, in experimental analysis, it is difficult to provide different conditions for testing or to prove the hypothesis as it is not feasible economically. In this sense, CFD technique play an important role to overcome all difficulties faced in experimental work.

Computational Fluid Dynamics (CFD) is a computational method used in recent years to obtain results for various parameters by numerical computation using software. CFD modeling and analysis allows to impose different constraints and boundary conditions for testing purposes without any involvement of actual cost experimentation. Also, results can be analyzed for different conditions. This part presents a brief review on CFD work available in the literature on heat transfer in shell and tube heat exchangers.
Shell-and-Tube heat exchanger is the basic type of straight tube heat exchanger. Mostly, tube used is of circular cross-section.

Nasr and Shafegat [3] developed a setup of shell and tube heat exchanger with segmented helical baffles for their research. Fluid flow pattern were studied in this paper. Comparison of numerical analysis had been carried out between different types of baffles in the heat exchanger. Segmented and helical baffles with 40° angle were compared. Also helical baffles of different angles were compared in this paper. The comparison was done on FLUENT 6.0 on various heat exchanger parameters of the CAD model. The results of this numerical analysis were shown graphically and various parameters were studied by the observation of the inferences drawn from the graph. By the results, it was concluded that the helical baffles heat exchanger was found to be perfect replacement for the shell and tube heat exchanger. Relationship between area, heat exchanger coefficient and pressure drop were derived for shell side developed in the helical baffles. All the parameters were better for helical baffles. However, when a pressure drop is observed, thermal duty has to be filled with a bigger cross sectional area. Usually the 40° whereas results for optimum results were seen in small cross sectional area only. Results were seen to be optimum at 20° were very close to the results seen in 40°. Results for other angles were found to be varying.

Ozden and Tari [4] studied the effect of baffles on straight tube heat exchangers and their performance. The shell side design of a shell-and-tube heat exchanger; in particular the baffle spacing, baffle cut and shell diameter dependencies of the heat transfer coefficient and the pressure drop were investigated by numerically modeling a small heat exchanger. The flow and temperature fields inside the *k*-ε model for analyzing the turbulence effect. Particle velocity path line was observed for baffle cut 25% and 36% by changing the number of baffles i.e. 6,8,10, & 12. Recirculation was observed for lower number of baffles. Thus, it was reported that in order to improve effectiveness, number of baffles must be increased.

Sarma and Das [5] predicted the performance of a shell and tube finned heat exchanger for waste heat recovery. The performance parameters such as effectiveness, overall heat transfer coefficient, energy extraction rate etc., were considered. The CFD analysis for given parameters was performed in FLUENT 6.3.16. The model for analysis was developed in GAMBIT 2.4.14. *k*-ε model was used for analysis of turbulence in fluid. Analysis was performed for different varying velocity range of fluid from 0.85 m/s to 145 m/s with temperature of 25°C, keeping the shell side fluid constant at the speed of 0.0709 m/s and has a temperature of 120°C. Pressure drop along the tube was found to have greater value. This implies that the pumping power of fluid should be high to nullify the pressure drop in order to attain desired pressure. It was observed that length of the pipe can be shortened to overcome the pressure drop. No back pressure flow was observed in the CFD results. Outlet temperature of fluid seems to decrease with increase in tube diameter. This indicates better heat transfer between the fluids. The CFD results were compared with the experimental results for effectiveness. The results were found to be in good agreement, having an error range of 6.4%.

Thundil and Ganne [6] studied various heat transfer parameters in the numerical analysis of the baffles of the shell and tube heat exchanger. The inclinations used were 0°, 10° and 20° and their results were compared. The numerical analysis was done on ANSYS CFX 12.1. It was observed that temperature gradually increases from 300K to about 340K. The pressure drop was found to be less for 20° baffles as compared to the other two. A smoother flow was observed in baffles other than 0°. The flow and temperature fields were resolved in the shell side of a small shell and tube heat exchanger. The mass flow rate must be below 2 kg/s, if it was more than 2 kg/s, the pressure drop was found to increase at a faster rate with little change in the outlet temperature. The baffles could be used effectively if the angle was beyond 20°, since the centre row of tubes are not supported. All the parameters were found to be better for the baffle with 20° inclination angle and hence it was the optimum angle for the given numerical problem.

Patel and Mavani [7] designed shell and tube heat exchanger and conducted numerical analysis. HTRI software was used for the numerical analysis of the problem. Heat transfer area and pressure drop were varied in this paper and various heat transfer parameters were observed in order to design the best shell and tube heat exchanger for the given conditions. It was observed that pressure drop was directly proportional to the fluid flow rate. The pitch ratio, length and the layout of the tube and the baffle spacing were found to have a major effect with the change in pressure ratio.

You *et.al.*[8] performed analysis by developing a numerical model and meshed using GAMBIT and the numerical analysis was done using FLUENT. Concepts of porosity and permeability were studied for the shell-side flow and heat transfer of shell-and-tube heat exchangers (STHXs). The distributed resistance, heat source, the distributed turbulence kinetic energy and its dissipation rate were studied. The numerical analysis was done at shell-side Reynolds numbers over a wide range from 6813 to 22,326 for a shell tube heat exchanger with flower baffles. The contours of the velocity and temperature fields were studied for the heat exchangers with and without flower baffles.

Parmar and Chopra [9] presented mathematical modelling of cross-counter flow heat exchanger. An attempt was made to analyze the performance of shell and tube type cross counter flow heat exchanger by changing the various parameters like both hot and cold fluid flow rate, direction of fluid flow. After changing the various parameters, the maximum performance obtained. For that the mathematical model of counter flow heat exchanger was adopted and also the analysis of the heat exchanger was carried out. ε-NTU Method A new numerical methodology for thermal performance calculation in cross-flow heat exchangers was developed. Effectiveness-number of transfer units (ε-NTU) data for several standard and complex flow arrangements was obtained using this methodology. The maximum performance for cross counter flow heat exchanger by varying the various parameters was determined. A method based on ε-NTU approach was utilized for the analysis of heat exchanger performance.

Jadhav and Koli [10] analyzed shell and tube heat exchanger for heat transfer properties and effect of baffles on pressure drop. Numerical modeling was performed on shell side of heat exchanger to observe change in heat transfer co-efficient and pressure drop. Analysis was performed in ANSYS FLUENT. A small heat exchanger model was developed with 6 baffles, as larger model would consume more computational time. For the given model with two types of baffles, it was found that the CFD analysis showed a lower pressure drop value at baffle cut of 30% than that of value observed for baffle cut at 25%. Thus it was noticed that, less pumping power was required and greater heat transfer was achieved in the model.

Yang *et.al.* [11] performed 3-D numerical simulations of a rod-baffle shell-and-tube heat exchanger with four different modeling approaches were developed and validated with experimental results. In two models there were subsection of the heat exchanger, in one heat exchanger was a porous medium and in one it was a whole medium. Meshing was done using GAMBIT. The surfaces of the rod baffles were set as adiabatic because the impact caused by thermal conduction of the rod-baffle could be neglected, and taking the rod-baffle surface as adiabatic allows for a coarser grid. The inner wall of shell side was also set as adiabatic because the heat exchanger is thermally isolated during the experiment. It was concluded that the periodic model, porous model and whole model have a high accuracy on predicting heat transfer performance, while the unit model had a relatively low accuracy; it was concluded that the porous model and whole model has high accuracy on predicting pressure drop, while unit model and periodic model are unable to directly predict hydraulic performance.

The objective of this project titled is to investigate numerically the effect of the inlet conditions on the flow, heat transfer and pressure drop of a shell tube heat exchanger.

MATHEMATICAL MODEL AND NUMERICAL SIMULATION

Governing equations

The renormalization group (RNG) k–ε model [12–13] is adopted because it can provide improved predictions of near-wall flows and flows with high streamline curvature.

Figure 1, presents the considered geometry, only the half of the heat exchanger is treated for symmetry reasons. The Number of tubes is Nt = 37 with a 0.5 m as length.

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

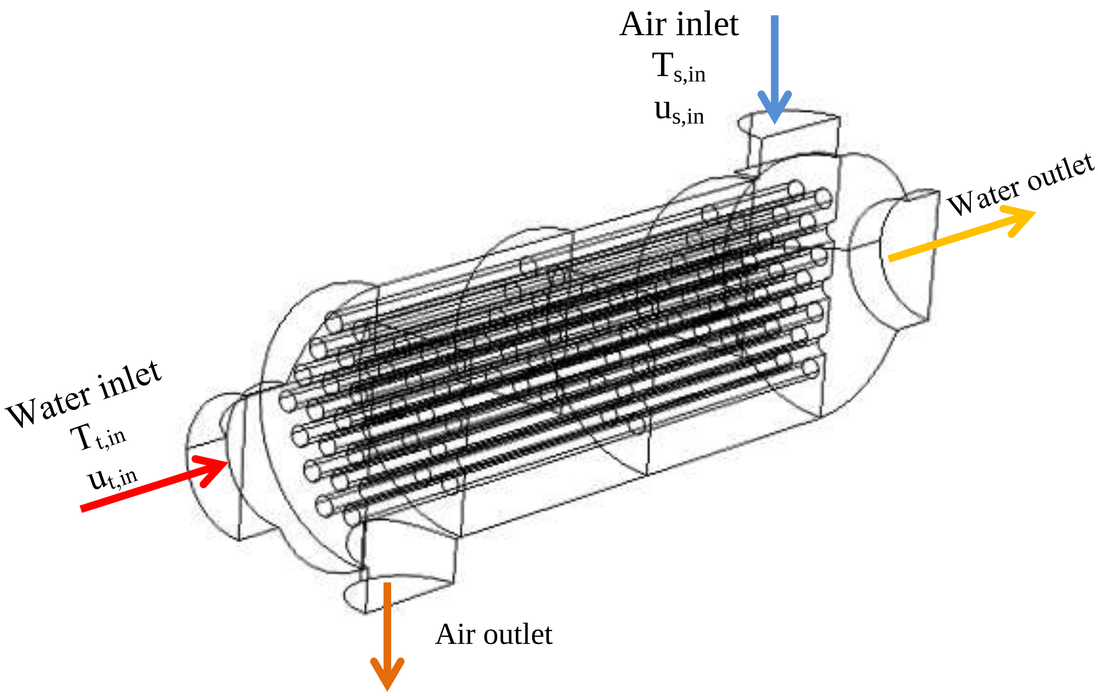


Figure 1. Geometry of the studied shell and tube heat exchanger

The governing equations for the Continuity, momentum, and energy conservations, and for k and ε can be expressed as follows:

Continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[(\mu+\mu_t)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) - \delta_{ij}\tfrac{2}{3}\frac{\partial u_k}{\partial x_k}\right] - \frac{\partial(\delta_{ij} p)}{\partial x_j} \tag{2}$$

Energy equation:

$$\frac{\partial(\rho u_j C_p T)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\lambda + \frac{\mu_t C_p}{\mathrm{Pr}_T}\right)\frac{\partial T}{\partial x_j}\right] + \mu\frac{\partial u_j}{\partial x_i}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i} - \delta_{ij}\tfrac{2}{3}\frac{\partial u_k}{\partial x_k}\right) + \rho\varepsilon \tag{3}$$

Turbulent kinetic energy equation:

$$\frac{\partial(\rho u_j K)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\mathrm{Pr}_K}\right)\frac{\partial K}{\partial x_j}\right] + \mu_t\frac{\partial u_j}{\partial x_i}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) - \rho\varepsilon \tag{4}$$

Dissipation rate of turbulent kinetic energy:

$$\frac{\partial(\rho u_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\mathrm{Pr}_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{C_1\varepsilon\mu_t}{K}\frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) - \frac{C_2\rho\varepsilon^2}{K} \tag{5}$$

The turbulent viscosity is defined as:

$$\mu_t = \frac{C_\mu \rho \varepsilon^2}{K} \tag{6}$$

With:

$C_\mu = 0.09$, $C_1 = 1.47$, $C_2 = 1.92$, $\Pr_K = \Pr_T = 1$, $\Pr_\varepsilon = 1.3$ [12]

Boundary conditions

Non-slip boundary condition is applied on the inner wall of the shell and all solid surfaces within the computational domain. The standard wall function method is used to simulate the flow in the near-wall region. The mass-flow-inlet and out flow boundary condition are applied on the inlet and outlet sections, respectively. To the authors' knowledge, such treatments of the inlet and out let conditions are corresponding to the average velocity distribution at the inlet and the fully developed condition [13] at the outlet. The shell wall of heat exchanger is set as adiabatic. Heat conduction of baffles in heat exchanger is considered by using shell conduction in thin-walls model in Comsol Multiphysics. The baffles are made from stainless steel. The air is taken as working fluid for shell Side and water for tube side of heat exchanger and thermophysical properties of the fluids are taken as function of temperature.

Shell-side heat transfer coefficient

The heat exchange rate of shell –side fluid:

$$\phi_s = \dot{m}_s Cp_s \left(T_{s,in} - T_{s,out}\right) \tag{7}$$

The shell-side heat transfer coefficient hs is equal to [20]:

$$h_s = \frac{\phi_s}{A_o.LMTD} \tag{8}$$

$$A_o = N_t.\pi.d_o.l \tag{9}$$

$$LMTD = \frac{\Delta T_{\max} - \Delta T_{\min}}{\ln\left(\Delta T_{\max} / \Delta T_{\min}\right)} \tag{10}$$

$$\Delta T_{\max} = T_{s,in} - T_w \tag{11}$$

$$\Delta T_{\min} = T_{s,out} - T_w \tag{12}$$

Grid generation and numerical method

The 3D computational domain is discretized with unstructured tetrahedral elements and the region adjacent to the tubes is meshed much finer to meet the requirement of wall function method. The total number of tetrahedral element is 104839. The meshes of the computational model are shown in Fig. 2.

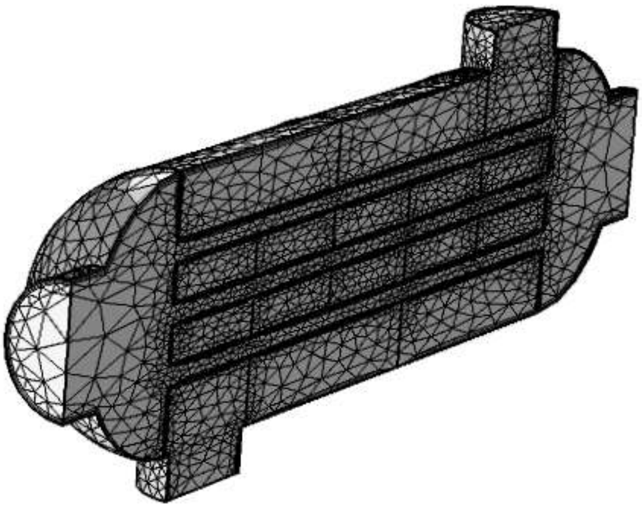


Figure 2. Geometry meshing

Non-Isothermal flow problem

The conjugated problem as described in the mathematical model which contained a cascade of different concerns that needs to be dealt with in order to ready the formulation for analysis. First of all we require that the temperature and heat fluxes be continuous everywhere. In other words we need the boundary conditions of equations governing the problem to be appropriately imposed in such a way that the temperature fields and fluxes between the solid matrix, water stream and air stream are continuous.

The commercial code Comsol Multiphysics is adopted to simulate the flow and heat transfer in the computational model. The governing equations are discretized by the finite element method. The convergence criterion is that the mass residual should be less than 10^{-6} for the flow field and the energy residual less than 10^{-8} for the energy equation. The computation is performed on a HP-Desktop Dual-Core CPUs and 8 GB memory every simulation case takes approximately 72 h to get converged solutions.

RESULTS AND DISCUSSION

The flow path lines in the shell side of the heat exchanger are shown in Figure 3. It can be clearly observed that except the inlet end region the fluid passes though the tube bundles basically in a zigzag flow pattern. Because of the zigzag flow pattern there are large dead spaces and significant back mixing at the back of the baffles where fluid recirculates with low velocity, which is confirmed by figure 4 presenting the shell-side velocity profile in some cross sections. Those dead zones results also in inefficient use of the heat transfer area and in pressure drop.

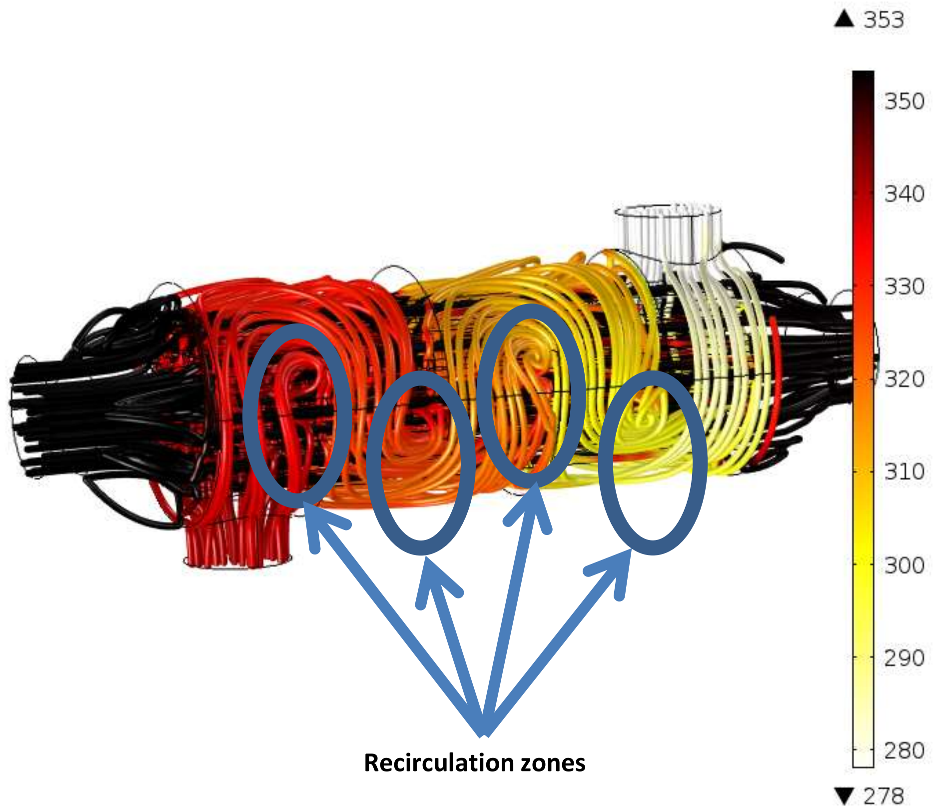


Figure 3. Path lines for $T_{t,in}$=353 K, $u_{t,in}$=0.1 m/s, $T_{s,in}$=278 K, and $u_{s,in}$=1 m/s

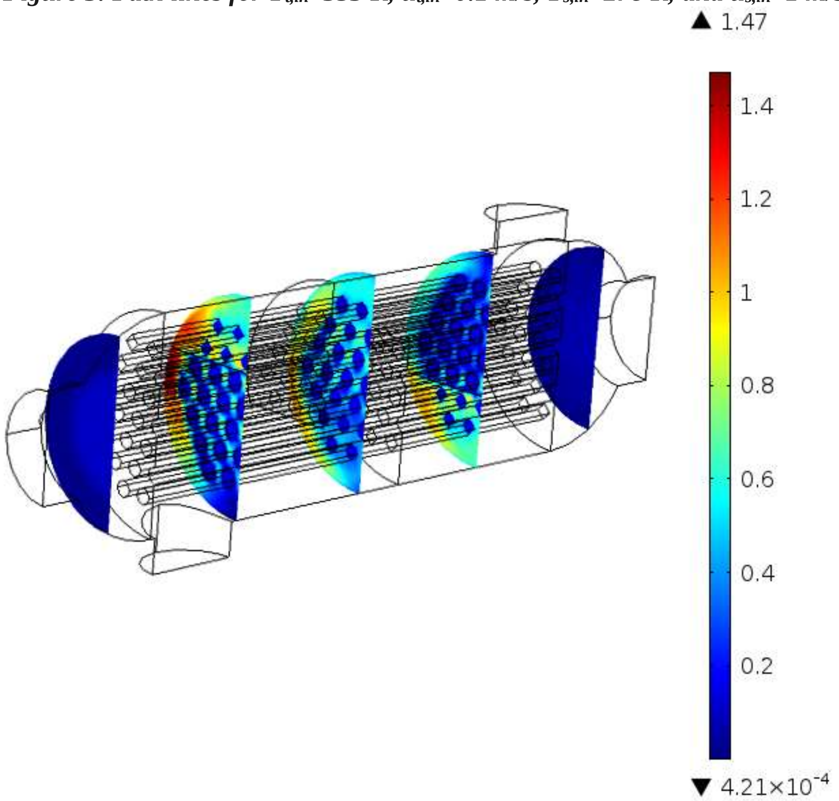
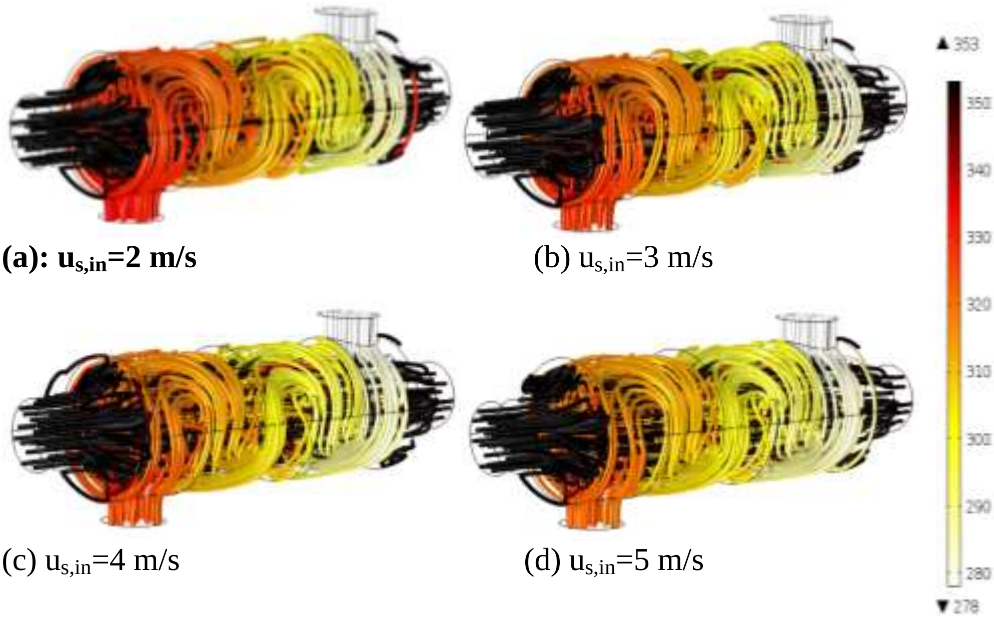


Figure 41. velocity profile for for $T_{t,in}$=353 K, $u_{t,in}$=0.1 m/s, $T_{s,in}$=278 K, and $u_{s,in}$=1 m/s

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

From figure 5 it's clear that the increase of the air inlet velocity causes a decrease of the outlet air temperature due to the decrease of the residence time. This result is confirmed by figure 20 showing the variation of water and air outlet temperatures as function of inlet air velocity. From figure 6, it is also noted that water outlet temperature decrease also by increasing $u_{s,in}$



(a): $u_{s,in}$=2 m/s (b) $u_{s,in}$=3 m/s

(c) $u_{s,in}$=4 m/s (d) $u_{s,in}$=5 m/s

Figure 5. Path lines for $T_{t,in}$=353 K, $u_{t,in}$=0.1 m/s and $T_{s,in}$=278 K.

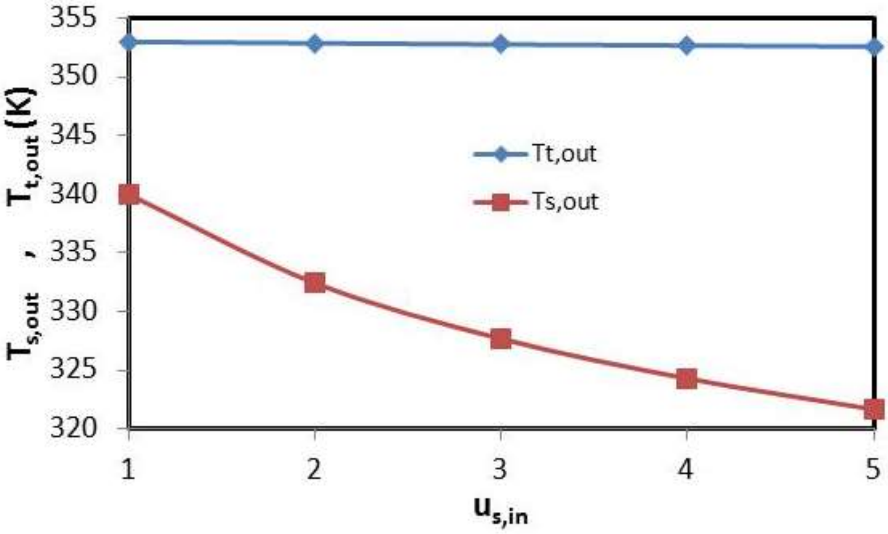


Figure 6. Variation of average $T_{t,out}$ and $T_{s,out}$ as function of $u_{s,in}$

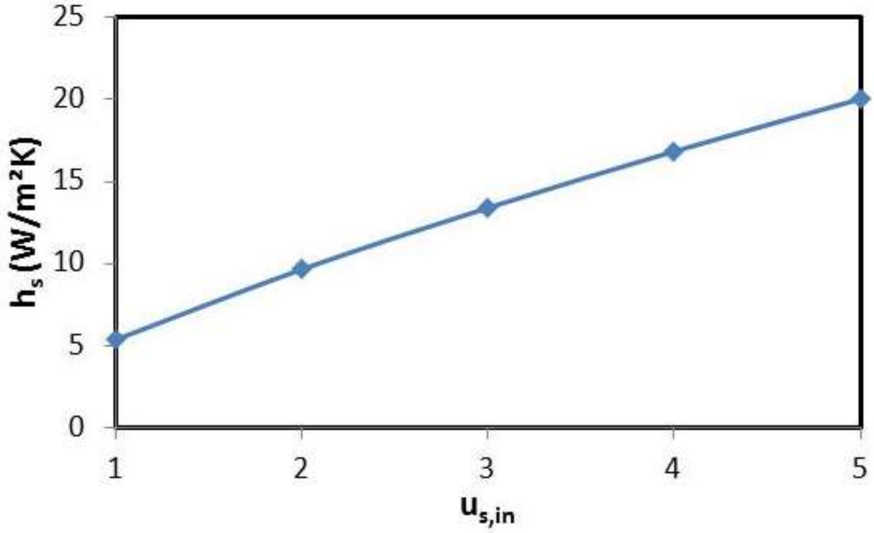


Figure 7. Heat transfer coefficient versus air inlet velocity

Figure 7, reports the heat transfer coefficient verses air inlet velocity, from this figure, it can be found out that heat transfer coefficient is higher for higher velocity. This difference is due to the flow pattern which becomes more turbulent for higher velocity.
The pressure drop is an important parameter in the design of heat exchanger. Pumping costs are depended on the pressure drop therefore lower pressure drop leads to lower operation costs.
Figure 8 shows the variation of the pressure drop verses air velocity. The flow separation at the edge of each baffles causes abrupt momentum change and serves pressure loss and also it is obvious from mentioned figure that the pressure drop increases with the increase of velocity.

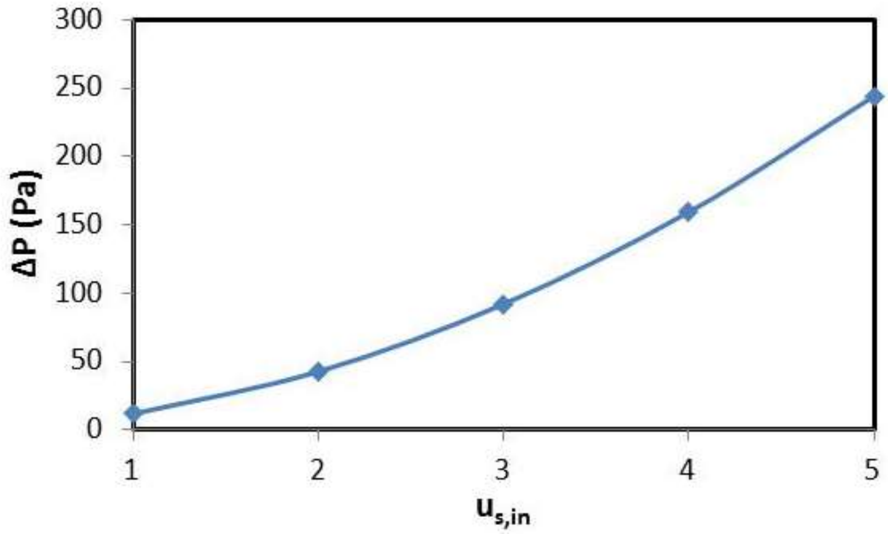


Figure 8. Pressure drop as function of inlet air velocity

CONCLUSION

A three-dimensional mathematical model for flow and heat transfer in a shell and tube heat exchanger has been developed from physical principles, with efforts aimed at a detailed description of the thermally and hydraulically developing flow problem. The mathematical model was implemented to analyze the phenomena occurring in shell and tube heat exchanger in full three-dimensional detail. Different inlet conditions are compared to evaluate the performances of the heat exchanger. It was found that the increase of the inlet air velocity (sell-side) increases the heat transfer coefficient and decreases the air outlet temperature.

REFERENCES

[1] Heat exchanger design handbook, T. KUPPRN MARCEI,DEKKER,INC (2000).
[2] Heat-Transfer Equipment, Perry section 11 (1999)
[3] M.R. Jafari Nasr, A. Shafeghat, 'Fluid flow analysis and extension of rapid design algorithm for helical baffle heat exchangers', (2008).
[4] Ender Ozden, Ilker Tari, 'Shell side CFD analysis of a small shell-and-tube heat exchanger', (2010).
[5] Swapnaneel Sarma, D. H. Das, 'CFD Analysis of Shell and Tube Heat Exchanger using triangular fins for waste heat recovery processes', (2012).
[6] Rajagapal Thundil Karuppa Raj, Srikanth Ganne, 'Shell Side Numerical Analysis Of A Shell And Tube Heat Exchanger Considering The Effects Of Baffle Inclination Angle On Fluid Flow', (2012)
[7] Sandeep K. Patel, Prof. Alkesh M. Mavani, 'Shell & Tube heat exchanger thermal design with optimization of mass flow rate and baffle spacing', (2012).
[8] Yonghua You, Aiwu Fan, Suyi Huang, Wei Liu, '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', (2012).
[9] Parmar Kalpesh D., Prof. Manoj Chopra, 'Performance Analysis Of Cross Counter Flow Shell And Tube Heat Exchanger By Experimental Investigation & Mathematical Modelling', (2013).
[10] Avinash D. Jadhav, Tushar A. Koli, 'CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop', (2014).
[11] Jie Yang, Lei Ma, Jessica Bock, Anthony M. Jacobi, Wei Liu, 'A comparison of four numerical modeling approaches for enhanced shell-and-tube heat exchangers with experimental validation', (2014).
[12] V. Yakhot, S.A. Orszag, Renormalization group analysis of turbulence 1: basic theory, J. Sci. Comput. 1 (1996) 3–11.
[13] W.Q. Tao, Numerical Heat Transfer, second ed., Xi'an Jiaotong University Press Xi'an, China, (2001).

AUTHOR BIBLIOGRAPHY

	**Lioua Kolsi** Lioua Kolsi, is an assistant professor of Energy engineering at the mechanical engineering department of college of engineer of Hail KSA. He got his Ph. D., in Energy Engineering in 2010, from the college of engineering of Monastir Tunisia. His main research interests are fluid flow and heat and mass transfer.