

# A Numerical Study of Single-phase Forced Convective Heat Transfer in Tube in Tube Heat Exchangers

P. Mohajeri Khameneh, I. Mirzaie, N. Pourmahmoud, M. Rahimi, S. Majidyfar

**Abstract**—Three dimensional simulations in tube in tube heat exchangers are investigated numerically in this study. In these simulations forced convective heat transfer and laminar flow of single-phase water are considered. In order to measure heat transfer parameters in these heat exchangers, FLUENT CFD Solver is used in this numerical method. For the purpose of creating geometry and exert boundary and initial conditions in the present model, finite volume method in Computational Fluid Dynamics is used in this study. In the present study, at each Z-location, variation of local temperatures, heat flux and Nusselt number at the whole tube is investigated in detail. Thereafter, averaged computational Nusselt number in this model is calculated. In addition, conceivable pressure drops have been obtained at each Z-location in this model. Then, pressure drop values in the present model are explored. Finally, all the numerical results for this kind of heat exchanger will be discussed precisely.

**Keywords**—Heat exchanger, Laminar flow, CFD, Nusselt number, Tube in tube, pressure drop.

## I. INTRODUCTION

HEAT exchangers serve a straightforward purpose: controlling a system's or substance's temperature by adding or removing thermal energy. According to Butterworth and Mascone [1], the area that might represent one of the more important aspects of a heat transfer engineer's job is compact heat exchangers. The design flexibility for selecting geometric configurations is a major advantage of compact heat exchangers. On the other hand, it may represent a major hurdle, because no comprehensive guideline can be easily developed for different kinds of compact heat exchangers. They have advantageous design features for selected applications [2], [3]; however, they are not the first choice for the design engineer in the process industries.

P. Mohajeri Khameneh is M.Sc student in Mechanical Engineering Department, Urmia university, Urmia, Iran.(corresponding author to provide phone:+98-9360835029, fax:+98-21-88977985, email: pedram.mohajeri@gmail.com)

I. Mirzaie is associate professor in Mechanical Engineering Department, Urmia university, Urmia, Iran. (e-mail: enpim@yahoo.com)

N. Pourmahmoud is assistant professor in Mechanical Engineering Department, Urmia University, Urmia, Iran, (e-mail: n.pormahmoud@mail.urmia.ac.ir)

M. Rahimi is M.Sc student in Mechanical Engineering Department, Urmia university, Urmia, Iran. (e-mail: m.rahimi259@yahoo.com).

S. Majidyfar is M.Sc student in Mechanical Engineering Department, Urmia university, Urmia, Iran. (e-mail: samad\_majidyfar@yahoo.com).

The major technical and non-technical barriers must be removed in order to make compact heat exchangers the first choice. Although there are many different sizes, levels of sophistication, and types of heat exchangers, they all use a thermally conducting element— usually in the form of a tube or plate—to separate two fluids, such that one can transfer thermal energy to the other. Home heating systems use a heat exchanger to transfer combustion gas heat to water or air, which is circulated through the house. Power plants use locally available water or ambient air in quite large heat exchangers to condense steam from the turbines. Many industrial applications use small heat exchangers to establish or maintain a required temperature. In industry, heat exchangers perform many tasks, ranging from cooling lasers to establishing a controlled sample temperature prior to chromatography. Anyone who wants to use a heat exchanger faces a fundamental challenge: fully defining the problem to be solved, which requires an understanding of the thermodynamic and transport properties of fluids. Such knowledge can be combined with some simple calculations to define a specific heat transfer problem and select an appropriate heat exchanger. Heat exchangers come in a wide variety of types and sizes. Here are a few of the most common ones.

Coil heat exchangers have a long, small diameter tube placed concentrically within a larger tube, the combined tubes being wound or bent in a helix. One fluid passes through the inner tube, and the other fluid passes through the outer tube. This type of heat exchanger is robust—capable of handling high pressures and wide temperature differences. Although these exchangers tend to be inexpensive, they provide rather poor thermal performance because of a small heat-transfer area. Nevertheless, a coil heat exchanger may be the best choice for low-flow situations, because the single tube passage creates higher flow velocity and a higher Reynolds number. These exchangers are commonly used to establish a fixed temperature for a process-stream sample prior to taking measurements. These exchangers can also be used to condense high-temperature stream samples.

Plate heat exchangers consist of a stack of parallel thin plates that lie between heavy end plates. Each fluid stream passes alternately between adjoining plates in the stack, exchanging heat through the plates. The plates are corrugated for strength and to enhance heat transfer by directing the flow and increasing turbulence. These exchangers have high heat transfer coefficients and area, the pressure drop is also typically low, and they often provide very high effectiveness. However, they have relatively low pressure capability.

Shell-and-tube heat exchangers consist of a bundle of parallel tubes that provide the heat-transfer surface separating the two fluid streams. The tube side fluid passes axially through the inside of the tubes; the shell-side fluid passes over the outside of the tubes. Baffles external and perpendicular to the tubes direct the flow across the tubes and provide tube support. Tube sheets seal the ends of the tubes, ensuring separation of the two streams. The process fluid is usually placed inside the tubes for ease of cleaning or to take advantage of the higher pressure capability inside the tubes. The thermal performance of such an exchanger usually surpasses a coil type but is less than a plate type. Pressure capability of shell-and-tube exchangers is generally higher than a plate type but lower than a coil type.

Laminar parallel-plate heat exchangers are a favorable design, since they provide high heat transfer for a given pressure drop [4]. They are favorable for miniature cryocooler designs [5] because of the large heat transfer coefficients and compactness. The problem with laminar plate heat exchangers is that if flow maldistribution occurs, it severely degrades the performance of high thermal effective heat exchangers [4,6,7].

In recent years, practical heat transfer enhancement techniques have been developed and many articles have been devoted to this area [8, 9]. The biggest benefit from heat transfer enhancement is reduction in the size of heat exchangers. For the same heat load requirement, much smaller heat transfer area is needed due to a higher overall heat transfer coefficient. Second, the heat transfer for the same load can be carried out for smaller driving forces, which implies higher thermodynamic efficiency. Third, a higher heat load can be exchanged for the same area and the same driving force.

On the other hand, enhancement techniques have been developed for the shell side as well. Especially in cases of a viscous fluid with high fouling tendencies, it is better to place this process stream into the shell side of a heat exchanger. In this case, exchangers with helical baffles can be considered as a very effective way for enhancement [10, 11].

In this article, heat transfer characteristics in these kind of heat exchangers are investigated numerically. In this study, variation of temperature, heat flux and Nusselt number in these models are examined in detail. In these present simulations, single-phase water in laminar flow is considered as "working fluid" in these investigations. FLUENT CFD Solver is selected as an appropriate software in order to calculate heat transfer parameters in this paper. Then, pressure drop values in the present model were explored. Finally, after observation the trend of temperature, heat flux and Nusselt number for single-phase water in this kind of heat exchangers, results will be discussed precisely.

## II. NUMERICAL ANALYSIS

### A. Description of Numerical simulation procedure

In the present paper, FLUENT CFD Solver is used to investigate heat transfer parameters in these heat exchangers and then obtained results will be investigated in detail. After several exact simulations, required results will be obtained in this model.

In fact, the major aim of this study is to investigate heat transfer characteristics in this kind of heat exchangers. In this numerical method, in order to have higher exactness in calculations, three dimensional computations and analysis is used.

In this article, in order to reach to maximum capacity of heat transfer, efficient options in FLUENT CFD Solver are selected that they will result maximum temperature difference between hot and cold water. For the purpose of reasonable results, initial and boundary conditions are selected close to experimental conditions as much as possible. Finally, it is attempted that the detailed and reasonable explanations are demonstrated for each results.

### B. Assumptions of physical model

In this present model, geometry and existing boundary and initial conditions based on actual conditions which they are exerted in laboratories are defined. Whereas, in this model the length of the tube is longer than the length of hydrodynamic entrance zone. So, the flow of this fluid is assumed "developed" in further simulations. Other assumptions which they are considered in these simulations are listed below:

1. In order to describe fluid flow and heat transfer in this physical model, three dimensional Navier-Stokes and energy equations were utilized.
2. The process is steady and the fluid is incompressible.
3. The flow is laminar.
4. The body forces are neglected.
5. Radiation heat transfer and natural convective heat transfer are neglected.
6. Aluminum is used for jacket (outer tube) in this present model.

### C. Governing equations and boundary conditions

According to the above assumptions, the Navier-Stokes and energy equations are utilized to describe the fluid flow and heat transfer in the whole region of this model. The governing equations are:

Continuity

$$\nabla \cdot (\rho_f \vec{V}) = 0 \quad (2)$$

Momentum

$$\nabla \cdot (\rho_f \vec{V} \vec{V}) = -\nabla P + \nabla \cdot (\mu_f \nabla \vec{V}) \quad (3)$$

Energy

$$\nabla \cdot (\vec{V} (\rho_f c_{p,f} T)) = \nabla \cdot (k_f \nabla T) \quad (4)$$

Where  $k_f$ ,  $\mu_f$ ,  $\rho_f$ ,  $c_{p,f}$  represent the thermal conductivity of water, molecular viscosity, density and specific heat at constant pressure of water flow, respectively.

#### D. Computational modeling and Boundary conditions

After review of previous studies in this research area, it is concluded that in channel surfaces in comparison to outer tube finer meshing should be used. In these simulations in order to decrease the volume of calculations and the number of iterations in FLUENT CFD Solver a section of model is used for these simulations. With using of "SYMMETRY" options for number of walls in this simulation- instead of full simulation- a section of this model could be simulated with the same accuracy. In figure 1, the geometry of this model with its geometric characteristics is shown. Meanwhile, in this figure, Z-direction and X-Y plane are the indicators for the direction of fluid flow and cross section plane, respectively.

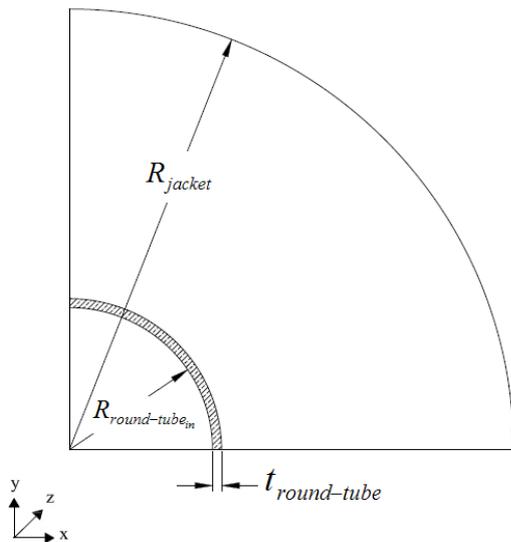


Fig. 1 Cross section of tube in tube heat exchanger

In the meanwhile, after several investigations among available and reputable experimental results, in table I geometric characteristics of this indicated model are tabulated.

TABLE I  
GEOMETRIC CHARACTERISTICS FOR TUBE IN TUBE HEAT EXCHANGER

Parameter	Length (mm)
$R_{\text{round-tube in}}$	5.53
$t_{\text{round-tube}}$	0.4
$R_{\text{jacket}}$	13
$L_{\text{tube}}$	1000

Besides, after examination and comparison those several kind of meshing finally "Tri-Pave" meshing is selected in Gambit Software. This kind of meshing has high accuracy and it can save the iteration time extremely. In figure 2, this kind of selected meshing is presented.

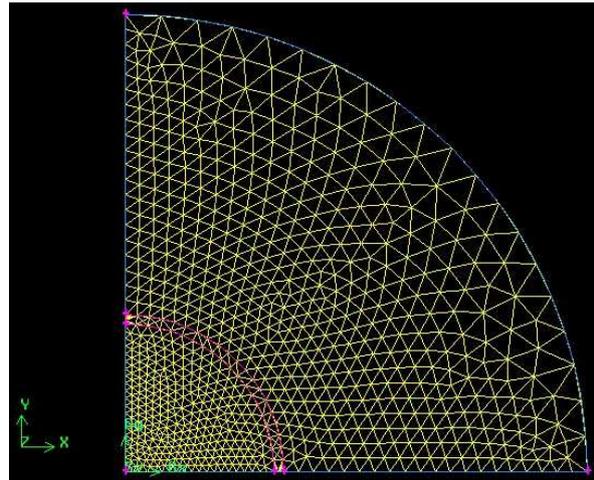


Fig. 2 Partial geometry and grid generated of tube in tube heat exchanger

Moreover, in order to gain confidence that all these numerical results aren't affiliated to different grid dimensions after exertion of meshing process, a grid independence examination is performed in this three dimensional model. In this study, based on number of grids which are available during the meshing procedure, three kind of grid qualities (fine, mediocre and harsh) were generated. After comparing iteration results and convergence speed, between these three kind of grids "mediocre meshing" is selected as a base model in order to continue further investigations and iterations in this numerical simulation.

In order to prevent simulation of whole geometry, "SYMMETRY" boundary condition is exerted to some distinct walls in this model. Besides, "Mass flow inlet" boundary condition is used for definition of entrance zone of this model. In order to obtain better convergence and to prevent in forming of reverse flow in outflow sections "pressure outlet" boundary condition is used. Finally, after modelling of this geometry in Gambit Software and exertion of all initial and boundary conditions, the present model is ready for numerical simulation.

#### E. Numerical simulation

By using reputable and available experimental results from previous articles, initial and boundary conditions are defined exactly for this present model. At first, in order to determining the water mass flow rate ( $\dot{m}$ ), equation 5 is exerted.

$$\dot{m} = \frac{ReA_{\text{port}}\mu}{D_h} \quad (5)$$

In order to locate  $D_h$  in equation 5, the value of hydraulic diameter is calculated with using of equation 6, as follows:

$$D_h = \frac{4(\text{cross-section-area})}{\text{Wetted-perimeter}} \quad (6)$$

The Darcy–Weisbach formulation, Equation 7, which it is presented in “Introduction to Heat Transfer” [12], will be utilized to determine pressure drop in laminar region of fluid flow. By placing Darcy friction factor ( $f$ ) in equation (7), pressure outlets are calculated for each simulation.

$$\Delta p = f \cdot \frac{L}{D_h} \cdot \frac{\rho V^2}{2} \quad (7)$$

Above procedure for calculating pressure drops will be presented admissible results. Consequently, initial and boundary conditions, pressure drop and friction factor ( $f$ ) which they were applied in simulation of this heat exchanger model are tabulated in table II.

TABLE II  
INITIAL CONDITIONS OF TUBE IN TUBE HEAT EXCHANGER

Parameter	Tube	Jacket
$T_{in}$ (K)	283.6	333
$\dot{m}$ (kg/s)	0.03	0.03
Re	1447.5	1165.3
$f$	0.05	0.078
$P_{gauge}$ (Pa)	325	6.68

Finally after the process of modelling and meshing, grid dependency process and also exact assigning of initial and boundary conditions, iteration process will be started.

### III. RESULTS AND DISCUSSION

The aim of this study is to calculate and discuss about the trend of variation for heat transfer parameters such as: temperature, heat flux and Nusselt number in this kind of heat exchanger. After the process of grid dependency and selecting the best kind of meshing, with using this kind of model, changes of heat transfer characteristics between outer wall and cold water is investigated precisely.

#### A. Wall and fluid local temperature

One of the most important aims in this simulation is to calculate wall and fluid temperature at each Z-location in the depth of this model. In this three dimensional numerical simulation with utilizing “X-Y plot” option in Fluent CFD Solver, the trend of variations for wall and fluid temperature are calculated. Finally, after averaging the results in Microsoft Excel at each Z location, temperature profiles for wall and water fluid are acquired. Wall and water fluid temperatures at each Z- locations are shown in Figure 3.

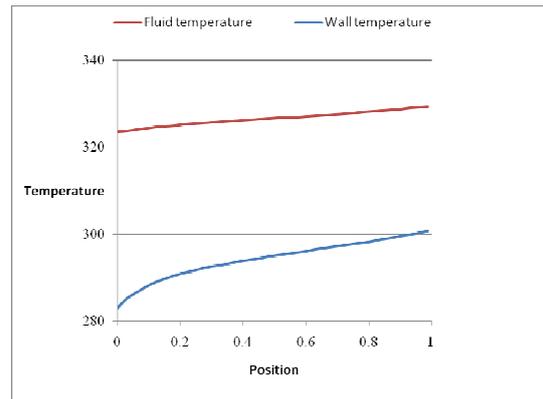


Fig. 3 Local fluid and wall temperature of tube in tube heat exchanger

As it can be seen in figure 3, with moving in the +Z direction, a decreasing temperature difference is obtained. As it is cleared, the maximum temperature difference will be occurred in the cold water inlet region or in the other words it will be happened in the outlet zone of the hot water.

As it is shown in figure 3 and with moving in the +Z direction, fluid and wall temperature could not reach together in any sections of this model.

#### B. Local heat flux variation

The rate of local heat transfer from hot water to channel wall at the +Z direction for tube in tube heat exchangers is plotted in Figure 4.

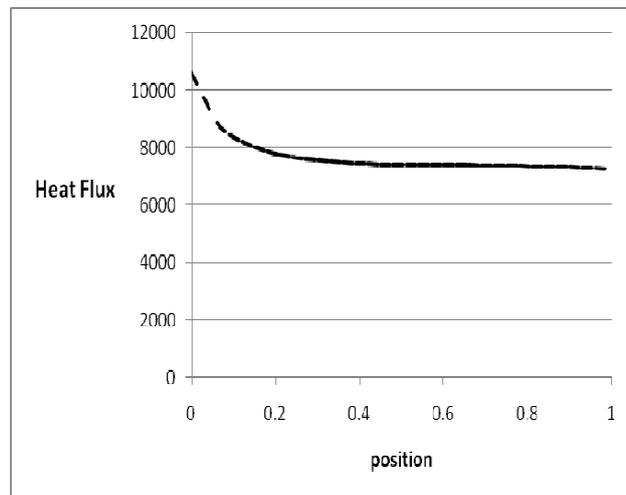


Fig. 4 Local heat flux variation for tube in tube heat exchanger

As it is cleared in figure 4, the maximum rate of heat flux is occurred in hot water outlet region. Based on temperature diagram in figure 3, it is obvious that maximum heat flux will be happened in that indicated region. With moving in +Z direction, the rate of heat flux is decreased gradually. Finally, in hot water inlet region because of minimum temperature difference between fluid and wall temperature, the minimum rate of heat flux is occurred.

### C. Local Nusselt number variation

The main goal of this section is to calculate local Nusselt number at each Z location along the tube length of this heat exchanger. With utilizing parameters which they were calculated later such as; wall and fluid temperatures, heat flux,...and with using equation (8), local Nusselt number can be obtained at each Z location.

$$Nu(z) = \left( \frac{q''(z)}{T_w(z) - T_f(z)} \right) \frac{D_h}{k_f} \quad (8)$$

Which  $k_f$ ,  $D_h$  and  $q''$  are indicated thermal conductivity of water, hydraulic diameter and heat flux, respectively. Also,  $w$  and  $f$  indices are introducers for wall and fluid, respectively. The trend of variation for local Nusselt number in this model is shown in figure 5.

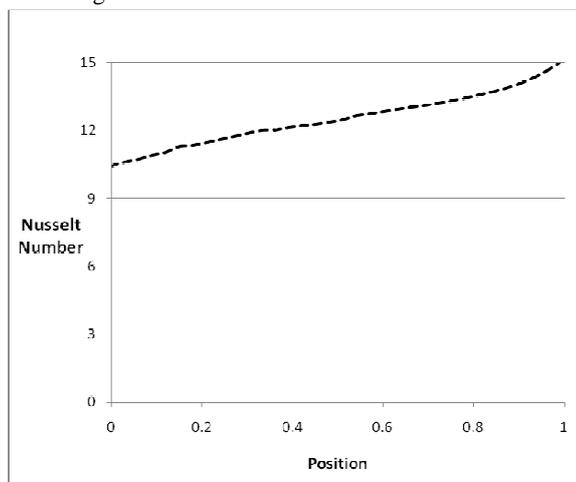


Fig. 5 Local Nusselt number distribution for tube in tube heat exchanger

With moving in +Z direction the local Nusselt number will be increased gradually because two efficient parameters in Nusselt number formula i.e.: heat flux and wall and fluid temperatures have a similar trend in this model. Finally, as it is cleared in figure 5, maximum local Nusselt number is occurred in the inlet region of hot water because in this region thermal boundary layer is beginning to develop.

### D. Computational average Nusselt number

In order to calculate the averaged Nusselt number in outer tube of this heat exchanger model, obtained values for local Nusselt number which they were listed in Microsoft Excel should be averaged. In this solution the distance for each grid is assumed 0.03. Consequently, averaged Nusselt number in the outer tube of this present model is presented, as:

$$Nu_{avg, Numerical} = 12.50$$

### E. Heat exchanger pressure drop

In this section, variation in pressure values along the Z direction of this model has been investigated numerically. The trend of pressure changes along the +Z direction of this tube in tube heat exchanger model can be observed in Figure 6.

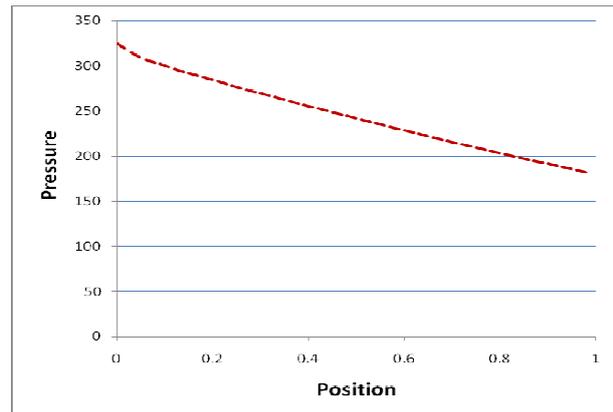


Fig. 6 Variation in pressure values along the Z direction in tube in tube heat exchanger model

Since a sudden flow contraction and expansion existed at the entrance and exit of this heat exchanger, the actual measured pressure drop included these contraction and expansion losses. The pressure drop, in figure 6, however, represents the pressure drop only along the channel, hence, the calculated pressure drop caused by the contraction and expansion were subtracted from the measured values.

## IV. CONCLUSION

Three dimensional simulations in this heat exchanger model were investigated numerically in this study. In these simulations forced convective heat transfer and laminar flow of single-phase water were considered. In order to measure heat transfer parameters in these models, FLUENT CFD Solver was used in this numerical method. For the purpose of creating geometry and exert boundary and initial conditions in the present model, finite volume method in Computational Fluid Dynamics was used in this study. In the present study, at each Z-location, variation of local temperatures, heat flux and Nusselt number at the whole tube was investigated in detail.

Thereafter, averaged computational Nusselt number in this model was calculated. In addition, conceivable pressure drops had been obtained at each Z-location in this model. Then, pressure drop values in the present model were explored for each Reynolds number. Finally, all the numerical results for this kind of heat exchanger were discussed precisely.

## REFERENCES

- [1] D. Butterworth and C. F. Mascone, "Heat Transfer Heads Into the 21st Century," *Chem. Eng. Prog.*, Sept., pp. 30-37 1991.
- [2] R. K. Shah, et al., *Compact Heat Exchangers*, Hemisphere Publishing Co., New York 1990.
- [3] J. M. Robertson, "The Development of Compact Heat Exchangers to Save Weight, Space, and Power Offshore," TEC88 Conference: Recent Advances in Heat Exchangers, Oct. 12-13, Grenoble, UK 1988.
- [4] Rawlins, Timmerhaus and Radebaugh, "Measurement of Performance of a Spiral Wound Polyimide Regenerator in Pulse Tube Refrigerator", *Adv. Cryogenic Eng.*, Vol.37, part B, p.947, 1991.

- [5] J. A. Crunkleton, J. L. Smith, Jr. and Y. Iwasa, " High Pressure Ratio Cryocooler with Integral Expander and Heat Exchanger", *Adv. Cryogenic Eng.*, vol.33, p.809, 1988.
- [6] E. B. Ratts, J. L. Smith, Jr. and Y. Iwasa, " Heat Transfer and Friction Factor Data for Gap Helical Screen Fin Counter Flow Heat Exchanger", *Adv. Cryogenic Eng.*, vol.39, part A, pp.1607-1614, 1993.
- [7] R. B. Fleming, " The Effect of Flow Distribution In Parallel Channels of Counter Flow Heat Exchangers ", *Adv. Cryogenic Eng.*, vol.12, pp. 352-362, 1967.
- [8] A. E. Bergles, "Heat Transfer Enhancement—The Encouragement and Accommodation of High Fluxes", *Trans. ASME J. Heat Transfer*, vol. 119, pp. 8–19, 1997.
- [9] R. L. Webb, *Principles of Enhancement Heat Transfer*, Wiley, 1994.
- [10] P. Stehlik, J. Nemcansky, D. Kral, and L. W. Swanson, "Comparison of Correction Factors for Shell-and-Tube Heat Exchangers with Segmental or Helical Baffles", *Heat Transfer Eng.*, vol. 15, pp. 55–65, 1994.
- [11] D. Kral, P. Stehlik, H. J. van der Ploeg and B. I. Master, "Helical Baffles in Shell-and-Tube Heat Exchangers, Part I: Experimental Verification", *Heat Transfer Eng.*, vol. 17, pp. 93–101, 1996.
- [12] F. P. Incropera, D. P. DeWitt, T. L. Bergman, A. S. Lavine, *"Introduction to Heat Transfer"*, Fifth edition, Wiley , Newyork, 2007.