World Applied Sciences Journal 18 (10): 1443-1448, 2012
ISSN 1818-4952

DOI: 10.5829/idosi.wasj.2012.18.10.1898

CFD Modelling of Increase Heat Transfer in Tubes by Wire Coil Inserts

[1]Arezuo Ghadi, 1Roja Parvizi Moghaddam and [2]Majid Taghizadeh Mazandarani

[1]Department of Chemical Engineering, Azad Eslamic University Ayatola Amoli Unit, Amol Mazandaran, Iran
[2]Department of chemical engineering, Industrial Nooshirvani Babol University, Babol, Mazandaran, Iran

Abstract: In this study has been studied the effect of improving heat transfer coils in heat exchanger in a laboratory by the method of computational fluid dynamics. A shell – tube heat exchanger is used in the laboratory. Difference in temperature and pressure are measured and compared in three different steps of coil, between input and output of each heat exchanger tubes, in the absence and presence coil.In this work the k–ε and RNG model has been used for representing the effects of the turbulence in tubes by CFD. Experimental results and CFD calculations show that increases heat transfer efficiency, by coil. Also, the heat transfer coefficient and friction coefficient increase with increasing the winding step.

Key words: Computational fluid dynamics・Coil・Shell – tube heat exchanger・Improved heat transfer

INTRODUCTION

In recent years, have been developed the application of enhanced heat transfer technology in many industrial, such as process industrial, car, etc [1, 2]. The industrials need, high efficiency heat exchangers according to economic interests and saving.for this target, reduce energy consumption (second low thermodynamic improvement) is important. The purpose of improving heat transfer is to achieve high thermal rate and reduce the size of heat exchanger or cost of them. Another benefit of this technology is decreasing the temperature driving force, which reduces the excess entropy and efficiency increases based on second law thermodynamic [3]. Furthermore, enhancement heat transfer causes with low velocity of fluid, achieve high heat transfer coefficient. Actually, create the lower pressure drop and reduce operating costs. The overall methods of improving heat transfer divided into two categories active and inactive. Active methods require outer energy sources, but in passive methods is not necessary directly outer energy sources. Examples of active methods include: mechanical aids, surface vibration, vibration fluid, induced flow devices and some inactive methods are: Use a variety of shiny surfaces, rough surfaces, increased levels, displaced enhancement devices, swirl flow devices, surface tension devices, strictures porous, additives. Application of these methods to improve heat transfer coefficient, depending on the type of thermal process which can be in range of free flow phase to film boiling of dispersed flow [4]. One of most general methods in improvement of heat transfer especially in shell and tube heat exchanger is inserts parts in tubes.

The effects of the V-nozzle turbulators on heat transfer friction, pressure drop and enhancement efficiency, have been investigated in a circular tube and uniform heat flux by S. Eiamsa-ard and P. Promvonge (2006). This experiments showed the enhancement efficiency decreases with increasing Reynolds number and pitch ratios of V-nozzle arrangement. The heat transfer in the circular tube could be increment by fitting with V-nozzles. The friction factor increase much higher than Nusselt number at the same Reynolds number [5].

Effects of star-shape fins inserts on the heat transfer and pressure drop was investigated in a concentric-tube heat exchanger, by Leonard Tijing (2006). The result showed that the straight fin is good to effectively enhance the heat transfer and not required using twisted fins in the counterflow heat exchanger. The overall heat transfer coefficient in a concentric-tube heat exchanger was enhanced as much as 51% at a constant pumping power compared with concentric-tube heat exchanger without star-shape fins inserts. a better heat transfer enhancement and lesser pressure drop could be obtained by making the fin thickness smaller so as to decrease flow restriction while maintaining a large surface region [6].

Corresponding Author: Arezuo Ghadi, Department of Chemical Engineering,
Azad Eslamic University Ayatola Amoli Unit, Amol Mazandaran, Iran.

P.K. Sarma and his assistants (2005) proposed equations for predicting heat transfer coefficient and friction coefficient in a tube with twisted tape inserts. Good agreement observed between experimental data and equations presented [7].

Mercado and his colleagues (2001), a new computational method have been presented for solving fluids heat transfer equations in tubes. This technique allows flow to move in the radial direction and solves by using fourth-degree finite differential equations. In this study, is used computational fluid dynamics with appropriate turbulence model to obtain the flow pattern and calculate the heat transfer in exchangers [8].

Theory: CFD methods, includ numerical solution of mass, momentum and energy equations.also there are another equations related to problem such as chemical reactions. solving by CFD consists two main stages. At first the fluid area divide into smaller components or control volumes, then partial equations (Navier–Stokes equations) apply for all of them. Consequently, many non-linear equations is obtained that must be solved simultaneously, by numerical simulation. The conservation equations for compressibility flow with turbulence are [8]:

- The continuity equation

$$\partial\rho/\partial t + div(\rho U) = 0 \quad (1)$$

- The momentum equations

$$\rho\, Du/Dt = -\partial P/\partial x + div(\mu\, grad u) + S_{Mx}$$

$$\rho\, Dv/Dt = -\partial P/\partial y + div(\mu\, grad v) + S_{My} \quad (2)$$

$$\rho\, Dw/Dt = -\partial P/\partial z + div(\mu\, grad w) + S_{Mz}$$

- The energy equation

$$\partial(\rho h_{tot})/\partial t - DP/Dt + div(\rho V h_{tot}) = div(k\nabla T) + S_E \quad (3)$$

The turbulence model are the k-□ standard model. The equations for k (turbulence generation) and □ (turbulence dissipation) are given below:

$$\partial(\rho k)/\partial t + \partial(\rho k u_i)/\partial x_i = \partial[(\mu + \mu_T/\sigma_k)\partial k/\partial x_j] + G_k + G_b - \rho\varepsilon - Y_M \quad (4)$$

$$\partial(\rho\varepsilon)/\partial t + \partial(\rho\varepsilon u_i)/\partial x_i = \partial[(\mu + \mu_T/\sigma_k)\partial\varepsilon/\partial x_j] + (C_{\varepsilon 1}\varepsilon/K)(G_k + C_{\varepsilon 3}G_b) - (C_{\varepsilon 2}\rho\varepsilon^2/k) \quad (5)$$

where ui is the velocity component along *xi* direction, μT the turbulent viscosity, μ the viscosity, Gb the buoyancy related turbulent kinetic energy production, YM the compressibility related kinetic energy production, Gk the shear stress-related turbulent kinetic energy production.

$$G_k = -\rho\overline{u_i' u_j'}(\partial u_j/\partial x_i) \quad (6)$$

$$G_b = (\beta g_i \mu_t/\Pr_t)(\partial T/\partial x_i) \quad (7)$$

in this equations β is the coefficient of thermal expansion, T the temperature, Prt is the turbulent Prandtl number for energy, gi is the component of the gravitational vector along xi. *C*ε1, *C*ε2, *C*ε3, σ*k*, σε and *C*μ are empirical constants are given in table (1).

Device and Tests Description: A special set are designed for experiments. Schematic design of this collection is given in Figure (1). The main part of the device is shell and tube heat exchanger. The tubes made of copper with inner diameter 17 mm and thickness 1 mm. This collection are placed in bath water within 60×60×30 cm3.water with temperature 16o(c) and constant flow rate 2.9×10-4 m3/s enters to bath water and exit from opposite side. So that water flows parallel to tubes. Three kinds wire coils has been studied that are given in table (2). Temperature of entrance water increase to 41o(c), before entering the collection tubes by electrical heater. entrance flow rate to shells and tubes, is controlled by two rotameter.

Also, two pressure sensors to measurement and record of pressure is installed at first and last of tubes. Experiments are performed in eight different flow rates and in the range of 2065 to 11,800 Reynolds numbers.

Table 1: *k* –ε Model constants

*C*ε1	*C*ε2	*C*ε3	σ*k*	σε	*C*μ
1.44	1.92	1.0	1.0	1.03	0.09

RNG-K-ε model is a modified example of this model.

Table 2: Geometry of designed coils

	p/d	e/d	p/e
W01, short pitch	1.25	0.076	16.4
W02, intermediate pitch	1.72	0.076	22.6
W03, long pitch	3.37	0.076	44.3

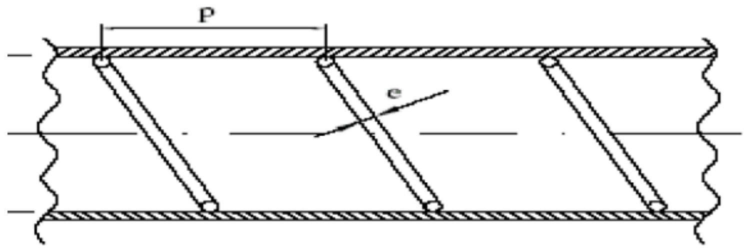

Fig. 1: View of the tubes and wire coils inserts and their dimensions

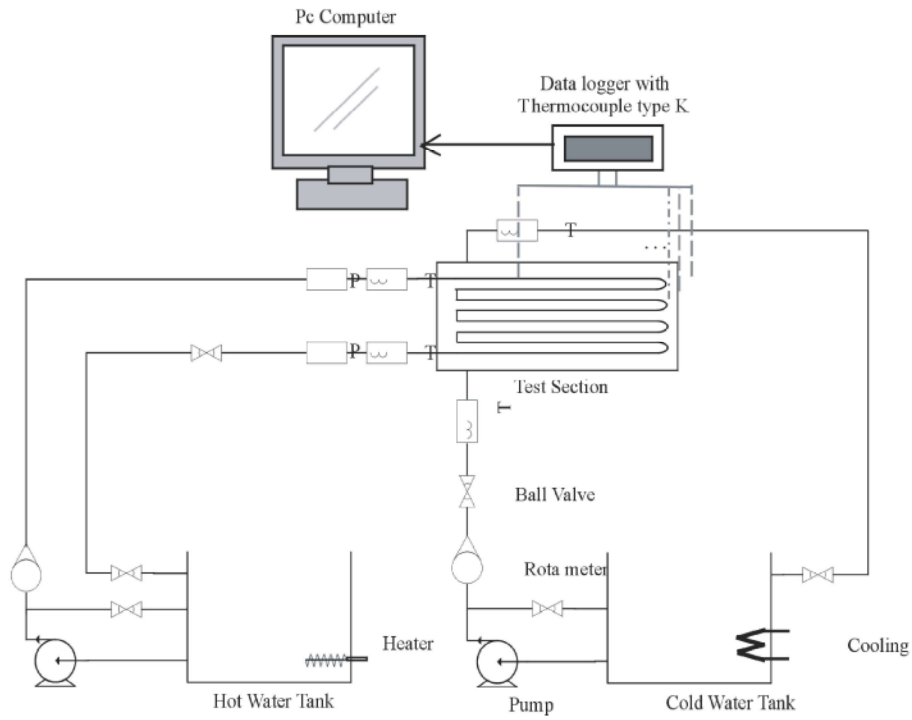

Fig. 2: View of designed collection

Temperature of inlet and outlet fluid and 20 points in the outer surface of tubes are measured by k type thermocouple. In this work, experiments was performed for all types of coils that mentioned in Table (2). In this tests, division of Reynolds number was equivalent to 2060, 2900, 4400, 5900, 7300, 8800, 10300, 11800.

Experiment Results and Discussion: For easier analysis of results, experimental data are displayed in form dimensionless numbers. Therefore, at first, heat transfer rate of hot fluid is calculated from Eq.(8):

$$Q = mC_p(T_o - T_i) \quad (8)$$

That Q is equal to the energy transferred from hot fluid to wall of tubes.

$$Q = hA(\tilde{T}_w - T_b) \quad (9)$$

In these equations $T_b = (T_o - T_i)/2$, $\tilde{T}_w = (\sum T_w)/20$, T_i is the inlet temperature, T_o the outlet temperature and T_w is the wall temperarure.

For calculation the heat transfer coefficient and Nuselt number, used below equations:

$$h = mC_p(T_o - T_i)/A(\tilde{T}_w - T_b) \quad (10)$$

$$Nu_m = hD_h/K \quad (11)$$

Also, to abtain friction coefficient, from pressure data, used this equation:

$$f = \Delta P \Big/ (L/D)(\rho U^2/2) \quad (12)$$

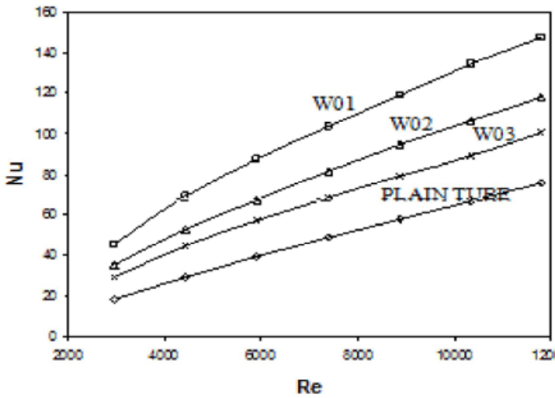

Fig. 3: Nusselt number versus Reynolds number in different cases of tubes with wire coils tube wire less

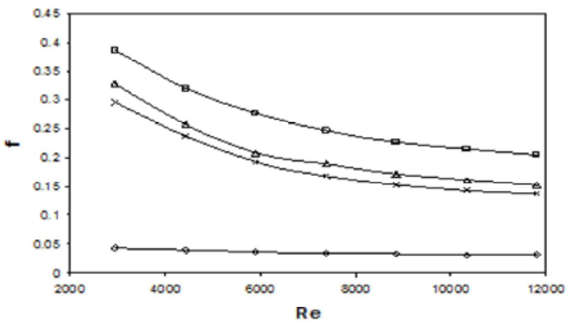

Fig. 4: Friction coefficient versus Reynolds number in different cases of tubes with wire coils tube wire less

In diference case of wire coils inserts and without it, Nusselt namber versus Reynolds number is depicted for water in Figure (3). In all cases, Nusselt number increase with increment of Reynolds. It's reasons are increase of fluid velocity and thermal convection coefficient. Also, for all cases of wire coils inserts, Nusselt number is higher than without it. So that, In most case, at Reynolds number 11800, This increase comes to about 100%. The reason can be expressed the development of tangential component of fluid velocity and rotation of the fluid with high spead. effect of wire coils and mixing of fluids near the wall and tube center couse to this increscent. The Nusselt number with wire coil W_{01}, is higher than W_{02} and W_{03}. In maximum Reynolds number, increscent of Nusselt namber with W_{01} than W_{03} is about 50%.

Figure (4) shows the variation of friction coefficient versus Reynolds number in diference case of wire coils inserts and without it. According to the diagram can be found that friction coefficient with wire coil inserts is very high than wireless. Also with increase pitch of wire, the friction coefficient decrease. So that, for a Reynolds

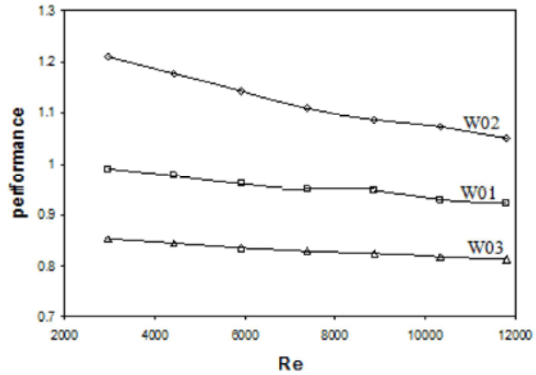

Fig. 5: Performance coefficient versus Reynolds number in different cases of tubes with wire coils tube wire less

number of 2900, friction coefficient in tubes with W_{01} is approximately 31% more than W_{03}. The reasons are more fluid contact with the tube's wall and wire coils, also, The reduce of cross section of fluid and creation of more rotational flow with W_{01}.

To obtain the overall efficiency of coils inserts in tubes and detection of heat transfer rates increase and Reduced pressure drop in various coils is used the thermal-hydraulic performance criteria as this equation:

$$\text{Thermal-hydraulic Performance} = (Nu / Nu_0)/(f / f_0) \quad (13)$$

That Nu and f are the parameters of tube with coils inserts and Nu_0 ana f_0 are related to empty tube. Therefore, from the results of heat transfer coefficient and friction coefficient is used to obtain a performance ratio. The result shows in Figure (5).

With increasing Reynolds number, decrease General trend of all diagrams. In other words, at low Reynolds numbers all tapes of coils are shown better performance. W_{02} have maximum performance coefficient that is justified with considering high rotational flow and heat transfer coefficient reduction of friction coefficient. After W_{02}, the maximum performance coefficient is for W_{01} and W_{03}, respectively. Because the heat transfer coefficient for W_{01} is more than W_{03}.

Simulation Results and Discussion: First, the system geometry made in Gambit software and performed the proper mesh. Control volume is divided to 1250000 tetrahedral elements. View of the geometry is shown in Figure (6).

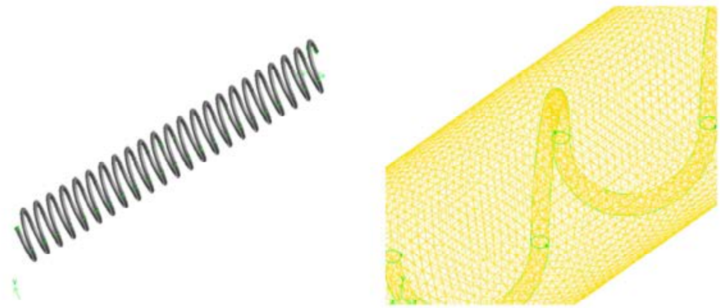

Fig. 6: A view of coil inserts and it's mesh

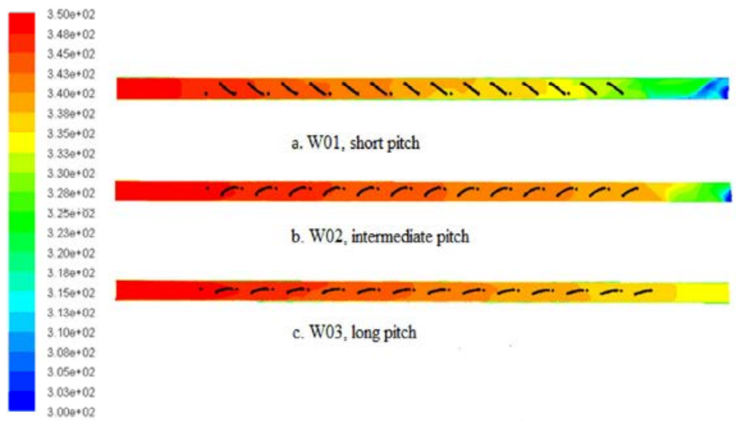

Fig. 7: Water temperature inside tube for three different type of coils

Table 3: Comparison between measured and predicted Nusselt number in tubes and error of them

Re	plain			W01			W02			W03		
	exp	CFD	e%	exp	CFD	e%	exp	CFD	e%	exp	CFD	e%
2950	18.23	18.42	1.0	45.41	45.86	1.0	35.19	35.72	1.5	29.26	29.53	0.9
5900	39.33	40.11	2.0	87.65	89.58	2.2	67.09	68.43	2.0	56.87	58.57	3.0
8850	57.97	59.65	2.9	118.7	122.3	3.0	94.36	96.91	2.7	79.03	82.35	4.2
11800	75.33	78.65	4.4	147.76	155.15	5.0	117.71	123.48	4.9	100.27	105.29	5.0

Table 4: Comparison between measured and predicted friction coefficient in tubes and error of them

Re	plain			W01			W02			W03		
	exp	CFD	e%	exp	CFD	e%	exp	CFD	e%	exp	CFD	e%
2950	0.044	0.044	1.5	0.385	0.39	1.2	0.327	0.334	2.0	0.295	0.297	1.0
5900	0.037	0.038	4.0	0.277	0.286	3.5	0.209	0.217	4.0	0.193	0.199	3.0
8850	0.033	0.035	6.0	0.225	0.239	5.9	0.171	0.181	6.0	0.153	0.16	5.0
11800	0.031	0.033	7.0	0.204	0.219	7.7	0.152	0.164	8.0	0.138	0.147	6.9

For solving was used the Fluent software. After define of fluid and it's physical properties, calculations for shaping the flow pattern, inside and outside of tube in different flow rate and steady state is done by Fluent. Simpel algoritm firt order method is used for Coupling pressure and velocity and $RNG - K - \varepsilon$ turbulence model to predict the turbulance flow pattern. convergence criteria is defined less than 10^{-4}.

For example, a horizontally counter of temperature for tubes with velocity flow 0.125 ($m.s^{-1}$) is shown in Figure 7. Process of temperature changes of tube is shown in this figure. The result showes the efficiency of tube with coil inserts is higher than the empty tube and W_{01} is effectiveness than other. The reasons are the rotary flow in tube, Increase of turbulency, the effect of wire coil inserts in creation high temperature difference and increase of heat transfer coefficient.

The predicted Nusselt number in various Reynolds by CFD, are compared with experimental results in table 3. Also, the friction coefficient is investigated by CFD and it's result are given in table 4.

Predicted results are in good agreement with experimental results. In high velocity the error increase for weakness of turbulancy model in high Reynolds. The wire coils Shows better performance in low Reynolds because the coils increase turbulancy and this device has no effect in high Reynolds number.

CONCLUSION

- Wire coils in tubes, is modeld, as a good tools for enhance heat transfer by CFD.
- In low spead region, usage wire coils in tube is very effective for increase heat transfer.
- Friction coefficient and Nuselt number in tubes with coils reduce with increase wire coils step.
- Computational fluid dynamics predict the experimental results with good accuracy.

REFERENCES

1. Webb, R.L., 1994. Principles of Enhanced Heat Transfer. John Wiley & Sons.
2. Bergles, A.E., 1997. Heat transfer enhancement - the encouragement and accommodation of high heat fluxes. Journal of Heat Transfer, 119: 8-19.
3. Bejan, A., G. Tsatsaronis and M. Moran, 1996. Thermal Design and Optimization. John Wiley & Sons.
4. Jacobi, A.M. and R.K. Shah, 1995. Heat transfer surface enhancement through the use of longitudinal vottices. Experimental Thermal and Fluid Science, 11(3): 295-309.
5. Eiamsa-ard, S. and P. Promvonge, 2006. Experimental investigation of heat transfer and friction characteristics in a circular tube fitted with V-nozzle turbulators. Heat and Mass Transfer, 33(5): 591-600.
6. Leonard D. Tijing, Bock Choon Pak, Byung Joon Baek and Dong Hwan Lee, 2006. A study on heat transfer enhancement using straight and twisted internal fin inserts. Heat and Mass Transfer, 26(5-6): 613-772.
7. Sarma, P.K., P.S. Kishore, V. Dharma Rao and A. Subrahmanyam, 2005. A combined approach to predict friction coefficient and convective heat transfer characteristics in a tube with twisted tape inserts for a wide range of Re and Pr. International Journal of Thermal Science, 44(4): 393-398.
8. Mercado, E.R.L., V.C. Souza, R. Guirardello and J.R. Nunhez, 2001. Modeling flow and heat transfer in tubes using a fast CFD formulation. Computers and Chemical Engineering, 25(4-6): 713-722.
9. Mahesh, N. and A. Kannan, 2006. CFD studies on natural convective heating of canned food in conical and cylindrical containers. Journal of Food Engineering, 77(4): 1024-1036.