CHEMICAL ENGINEERING TRANSACTIONS

VOL. 39, 2014

A publication of AIDIC

The Italian Association of Chemical Engineering
www.aidic.it/cet

Guest Editors: Petar Sabev Varbanov, Jiří Jaromír Klemeš, Peng Yen Liew, Jun Yow Yong

ISBN 978-88-95608-30-3; **ISSN** 2283-9216 DOI:10.3303/CET1439054

Parametric Study on Flow and Heat Transfer Performance of Multi-Flow Spiral-Wound Heat Exchanger

Gaopeng Zhang, Xing Lu, Xueping Du, Min Zeng, Qiuwang Wang*

Key Laboratory of Thermo-Fluid Science and Engineering, Ministry of Education, Xi'an Jiaotong University, Xi'an, 710049, China
wangqw@mail.xjtu.edu.cn

The multi-flow spiral-wound heat exchangers are widely used in industrial applications, but the mechanism of flow and heat transfer has not been clarified yet. A three-dimensional model is developed based on the FLUENT software in this study. The effects of different geometrical factors, such as the space bar thickness and the tube pitch in the first layer, on flow and heat transfer performance are numerical studied . It is found that as the space bar thickness increases, the Nusselt number and the pressure loss per unit length both decreases and the similar trend is obtained as the tube pitch increases. Meanwhile, once the geometrical factors and the Reynolds number in the tube-side are determined, the Nusselt number and pressure loss per unit length in the tube-side keep constant generally, regardless of the change of Reynolds number in the shell-side.

1. Introduction

Due to its high compactness and heat transfer coefficient under multi-stream condition, spiral-wound heat exchangers are widely used in industrial applications, such as air separation, process plants, nuclear industry, and refrigeration. Although this kind of heat exchanger has been put into use, the mechanism of flow and heat transfer on shell side is still not clarified yet and the method how to calculate the heat transfer coefficient and friction accurately is still being studied. Based on the heat transfer coefficient and pressure drop formulas when the fluid flows across the straight tube bank, Gilli (1989) deduced the heat transfer coefficient and pressure drop formulas for fluid flows outside the coiled tube bundles. Neeraas et al. (2004) performed an experimental investigation to study the heat-transfer and pressure drop characteristics on the shell-side of spiral-wound heat exchanger, using different working fluid (methane, ethane and propane, etc.) in different phase states. Ghorbani et al. (2010) implemented an experimental study on mixed convective heat transfer under various Reynolds number, curvature ratio and pitch. Moawed (2011) investigated the influence of various curvature and torsion rate on outer-side heat transfer coefficient. The experiment model was a vertical heat exchanger whose coil was wound by hollow pipe inserted with electric heating wire. Picón-Núñez et al. (2012) developed the graphical design method originally applied for shell and tube heat exchangers and extended to the case of spiral heat exchangers which have similar and simpler structure than the multilayer SWHEs.

The study on spiral-wound heat exchanger mainly consists of experimental method and two-dimension numerical simulation. In addition, almost all of the research subjects are single-stream spiral-wound heat exchanger and few papers report the flow and heat transfer performance of multi-stream spiral-wound heat exchanger. Meanwhile, due to its complex geometric structure and flow and heat transfer mechanism, there is no study report the three-dimension numerical study on multi-stream spiral-wound heat exchanger. In this study, a three-dimensional model is developed based on the FLUENT software. The effects of different geometrical factors, such as the space bar thickness, the tube pitch in the first layer and the tube external diameter, on flow and heat transfer performance are numerical investigated .

Please cite this article as: Zhang G., Lu X., Du X., Zeng M., Wang Q., 2014, Parametric study on flow and heat transfer performance of multi-flow spiral-wound heat exchanger, Chemical Engineering Transactions, 39, 319-324 DOI:10.3303/CET1439054

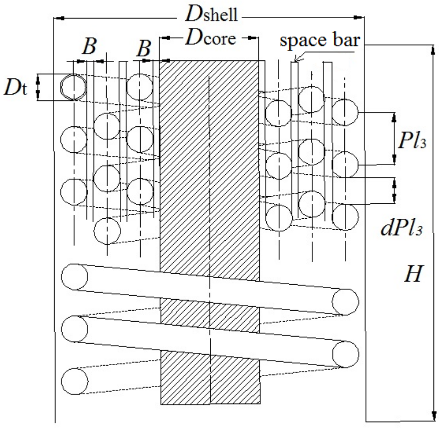

Figure 1: Element of a spiral-wound heat exchanger

2. Characteristics of multi-flow spiral-wound heat exchanger

Figure 1 shows the typical element of a spiral-wound heat exchanger. It can be seen that the structure of multi-flow spiral-wound heat exchanger is quite complicated. In the cylinder with space bar around, the central coil wrapped around the central cylinder using the space bar fixed. The coils' winding direction is opposite between adjacent layers. The study subject in this paper is a three-flow spiral-wound heat exchanger, there are two streams in the tube side and the flow directions of the two streams are opposite.
It consists of five main geometric factors which affect the flow and heat transfer. They are the central cylinder diameter D_{core} , the tube external diameter D_{t} , the tube pitch in the first layer Pl_1, the space bar thickness *B* and the helical layer number *C*.

3. Computation model

3.1 Physical model

To simplify the simulation, the following assumptions are made in this study:
1) The thermal properties of fluid in shell-side fluid are kept constant.
2) The flow and heat transfer are steady state and laminar.
3) The heat loss between heat exchanger and the environment is neglected.
4) The natural convection caused by the variation of fluid density is neglected.
In order to avoid the backflow in the end, prolongations are used at the inlet and outlet. The physical model for numerical simulation of spiral-wound tube heat exchanger is showed in Figure 2.

3.2 Type font and type size

The governing equations for continuity, momentum and energy are shown as follows.

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

$$\frac{\partial u_i u_j}{\partial x_i} = -\frac{\partial p}{\rho \partial x_i} + \frac{\partial}{\partial x_j}\left(\left(\nu + \nu_t\right)\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right)\right) \tag{2}$$

$$\frac{\partial u_{\mathrm{i}} T}{\partial x_{\mathrm{i}}} = \frac{\partial}{\partial x_{\mathrm{i}}}\left(\left(\frac{\nu}{\mathrm{Pr}} + \frac{\nu_{\mathrm{t}}}{\mathrm{Pr}_{\mathrm{t}}}\right)\frac{\partial T}{\partial x_{\mathrm{i}}}\right) \tag{3}$$

The velocity at inlet of shell side is assumed as uniform. Pressure-outlet condition is prescribed at the outlet. The tube wall temperature is constant. The walls of center tube and shell tube are adiabatic. The above equations along with the specified boundary conditions are solved numerically using a finite-volume method based on the ANSYS FLUENT software platform. The first order upwind scheme is used to treat the convection terms. The SIMPLEC method is adopted for pressure and velocity coupling. Grids in the computational region are unstructured generated by ANSYS ICEM. In order to obtain more precise

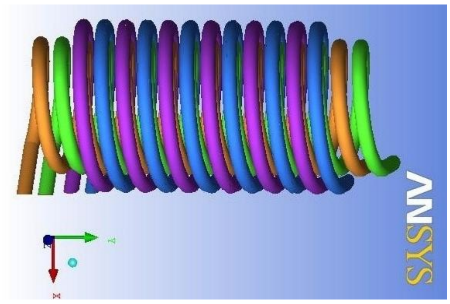

Figure 2: Physical model

simulation results and make the detailed analysis, the grids of specific zones including walls of the tubes, the central cylinder, the inlet and outlet are increased appropriately and sufficiently.

3.3 Numerical verification and model validation

Three kinds of grid systems have been established to test the effect of grid number. Figure 3 shows the variation of the average Nusselt number and pressure drop with different numbers of grid points. When the grid mesh number changes from 1,397,200 to 1,260,000, the average Nusselt number varies from 235.07 to 229.73 (i.e., 2 %), pressure drop per unit length from 1,338 to 1,241 (i.e., 7.3 %). The average Nusselt number and pressure drop per unit length varies slightly when the number of grid points reaches 1,137,771, which is used for taking both the accuracy and convergence rate into account.
In order to validate the numerical model employed in the paper, the numerical results are compared with the experimental results in (Neeraas et al., 2004) where the structures were exactly the same as the model size in the present study. Table 1 (Neeraas et al., 2004) shows the parameters of the inlet.

Table 1: Model operating conditions

Case	P_{in}/bar	T_{in}/°C	v_{in}/m·s^{-1}
1	13.664	-11.87	2.38
2	13.682	-11.67	4.65
3	13.501	-12.20	7.06
4	13.659	-13.82	8.07

Based on the experimental working conditions in (Neeraas et al., 2004), three kinds of turbulent models (Standard k-ε, RNG k-ε and Realizable k-ε) are adopted. As shown in Figure 4, the heat transfer coefficient predicted by RNG k-ε model agrees the best with the experimental. Therefore, the RNG k-ε model is adopted in the following simulation work.

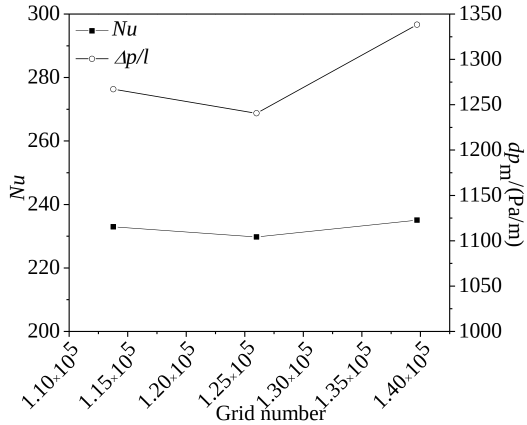

Figure 3: Grid independence test

3.4 Boundary conditions and physical property

The working fluid in this study is methanol. The velocity-inlet condition is prescribed at the inlet. The temperature of the fluid in the shell side inlet is 223 K and the temperature of the fluid in the tube side is 238 K and 253 K. Pressure-outlet condition is prescribed at the outlet. The coupling heat transfer is calculated using fluid area and solid area coupling solution. The thickness of the tube is 2 mm. The walls of both the central and external cylinders are adiabatic.

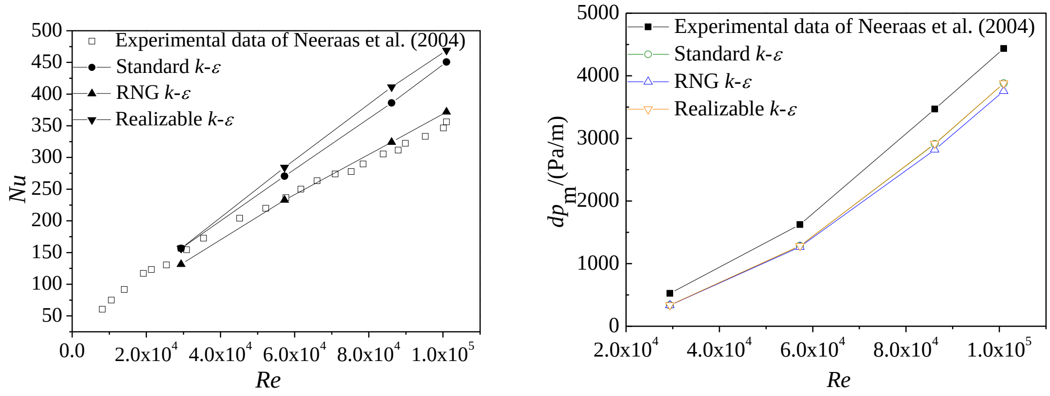

Figure 4: Model validation

4. Numerical simulation results and discussion

4.1 Effects of thickness of space bar

The thickness of space bar determines the shell-side area in which fluids pass through. Large space bar thickness means large flow area channel, as shown in Figure 1. In the case of central cylinder diameter is 80 mm, tube external diameter is 12 mm, the number of layer is 1, the tube pitch in the first layer is 17 mm, and we numerical study the effects of thickness of space bar on flow and heat transfer performance. Figure 5 shows effect of space bar on *Nu* number and pressure drop.

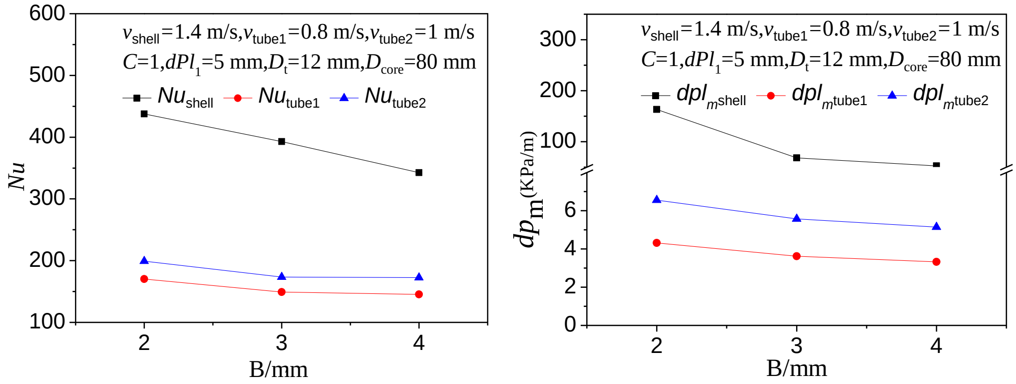

Figure 5: v_{shell}=1.4 m/s, effect of space bar on Nu number and pressure drop per length

From Figure 5 we can see that the heat transfer coefficient and pressure drop per length decrease with the increase of space bar thickness. From left to right in Figures 6-7 are the temperature and pressure distribution in shell side with space bar thickness is 2 mm, 3 mm and 4 mm . It could be found clearly that smaller thickness of space bar means better heat transfer performance and higher pressure drop. As the thickness of the space bar increases, the flow channel of the fluid becomes wider and the spacing between adjacent layers becomes larger which lead to the decreasing of the average flow velocity at the same time. So the thin space bar means narrow fluid flow channel and adequate fluid mixing. As a result, the heat transfer performance becomes better as well as the pressure loss becomes larger at the same time. So it is suitable to use neither too thick nor too thin space bar in the reality of designing a spiral-wound heat exchanger and the space bar with the thickness of 2~3 mm is recommended.

4.2 Effects of tube pitch in the first layer

As shown in Figure 1, the tube pitch refers to the longitudinal distance between the centres of adjacent tubes in the same layer. When other factors keep constant, the geometric dimension of tube pitch determines the tube winding angle, and the big tube pitch means big winding angle. In the case of central cylinder diameter is 80 mm, the tube external diameter is 12 mm, the number of layer is 1, the thickness of space bar is 2 mm, and the effects of tube pitch in the first layer on flow and heat transfer performance are numerically studied.

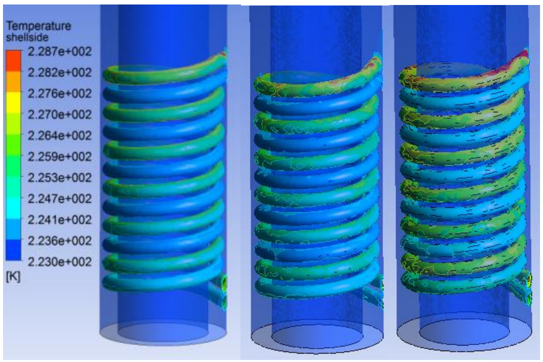

Figure 6: v_{shell}=1.4 m/s, temperature distribution in shell side with different space bar thickness

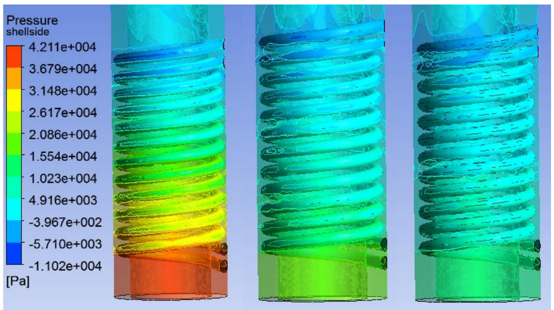

Figure 7: v_{shell}=1.4 m/s, pressure distribution in shell side with different space bar thickness

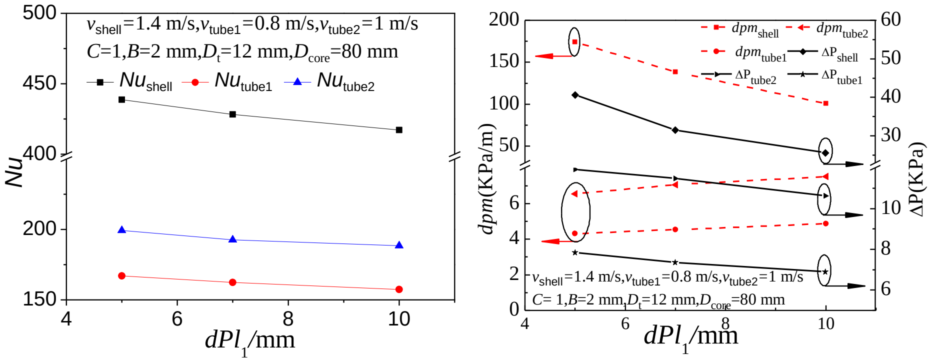

Figure 8: v_{shell}=1.4 m/s, effect of tube pitch in the first layer on Nu number and pressure drop

From Figure 8 we can see that the heat transfer coefficient and the pressure drop decrease with the increase of tube pitch in the first layer. From left to right in Figures 9-10 are the temperature and pressure distribution in shell side with tube pitch in the first layer is 17 mm, 19 mm and 22 mm. We could see clearly that smaller tube pitch in the first layer means better heat transfer performance and more pressure drop. This phenomenon could be explained as follows: when the tube pitch in the first layer increases, the gap among tube bundles becomes large which make less fluid keep stagnant and the pressure decreases correspondingly. Meanwhile, the flow resistance that restricts the fluid flows towards adjacent layers increases, which lead to the tendency that the fluid flows across the tube bundles is weakened. As a result, the Nusselt number and the heat transfer performance decreases. So small tube pitch in the first layer brings enhanced heat transfer performance as well as more pressure loss. It is suitable to choose neither too large nor too small tube pitch in the first layer in designing a spiral-wound heat exchanger and the tube pitch in the first layer that making the winding angle between 5^{o}~15^{o} is recommended.

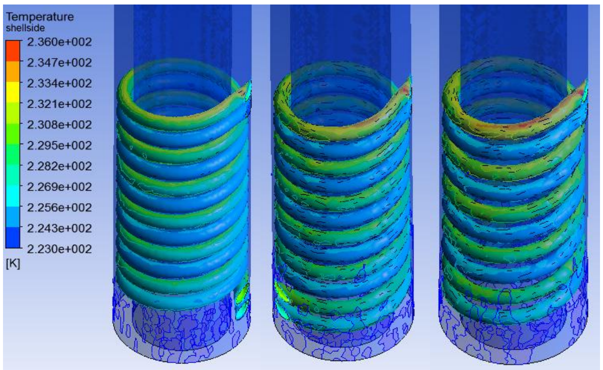

Figure 9: v_{shell}=1.4 m/s, temperature distribution in shell side with different tube pitch in the first layer

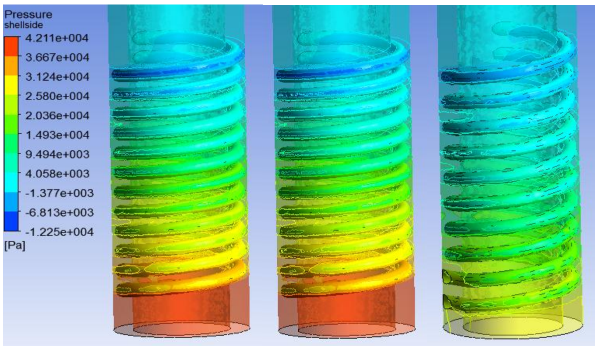

Figure 10: v_{shell}=1.4 m/s, temperature distribution in shell side with different tube pitch in the first layer

5. Conclusion

A three-dimensional numerical simulation model for multi-flow spiral-wound heat exchanger is developed and validated in the present study. The effects of some geometry parameters on the flow and heat transfer are investigated in detail. The main conclusions are drawn as follows. The Nusselt number and pressure loss per length both decrease with the increase of thickness of space bar. As the tube pitch in the first layer increases, the Nusselt number and the pressure loss both decreases.

Acknowledgments

This work was supported by the Natural Science Foundation of China (Grant No. 51276139).

References

Editorial Committee., 1989, The Latest Heat exchanger design calculation and heat transfer strengthening and quality inspection standards practical manual. The Northern Industrial Press, Beijing, China (in Chinese).

Neeraas B.O.,Fredheim A.O., Aunan B., 2004. Experimental shell-side heat transfer and pressure drop in gas flow for spiral-wound LNG heat exchanger, International Journal of Heat and Mass Transfer, 47,353-361, DOI: 10.1016/S0017-9310(03)00400-9.

Ghorbani N., Taherian H.,Gorji M., Mirgolbabaei H., 2010. Experimental study of mixed convection heat transfer in vertical helically coiled tube heat exchangers, Experimental Thermal and Fluid Science, 34,900-905, DOI: 10.1016/j.expthermflusci.2010.02.004.

Moawed M., 2011. Experimental study of forced convection from helical coiled tubes with different parameters, Energy Conversion and Management, 52, 1150-1156,DOI:1016/j.enconman.2010.09.009.

Picon-Nunez M., Polley G.T., Riesco-Avila J.M., 2012. Design space for the sizing and selection of heat exchangers of the compact type, Chemical Engineering Transactions, 29, 217-222, DOI:10.3303/CET1229037.