Proceedings of the 3rd IASME/WSEAS Int. Conf. on HEAT TRANSFER, THERMAL ENGINEERING AND ENVIRONMENT, Corfu, Greece, August 20-22, 2005 (pp79-84)

The Simulation and Optimal Design of Key Structure on Shellside of Large Heat Exchanger with Longitudinal Flow of Shellside Fluid

QI-WU DONG, MIN-SHAN LIU, XIAO-DONG ZHAO
Thermal Energy Engineering Research Center
Technology Institute of Zhengzhou University
97 Wenhua Road, Zhengzhou, Henan Province, 450002
P.R.CHINA

Abstract: In the paper, the periodic unit duct model is built for large heat exchanger with longitudinal flow of shellside fluid according to its structural characteristic. By employing CFD software FLUENT, the numerical simulation of models under different media, different Re number and different rod-baffle pitch are carried out. And the detailed characteristics of fluid flow and heat transfer in the duct and the optimal value range of rod-baffle pitch are obtained by analyzing and comparing the simulation results, which provides some reference for the research and application of heat exchanger with longitudinal flow of shellside fluid.

Key words: heat exchanger; longitudinal flow; numerical simulation; rod-baffle pitch

1 Introduction

Heat exchanger is a kind of universal process equipments widely used in chemical industry, chemical fertilizer industry, oil refinery, power plant, light industry and so on. The heat exchanger with longitudinal flow of shellside fluid (HELFSF)[1,2], which has been developed in the past few decades, has obtained wide usage due to its extraordinary advantages such as high heat transfer coefficient, low pressure drop and good antivibration performance.

In the research field of heat exchanger, experimental method is a traditional method while in recent years numerical method has gained wide use. With the progress of CFD and CFD software, numerical method has shown great advantages by contrast with experimental method. Therefore, numerical method is used by employing CFD software FLUENT in this paper to obtain detailed condition of fluid flow and heat transfer in HELFSF. And based on the simulation, optimal design method of rod-baffle pitch is put forward.

2 Building of simplified model[3,4,5]

2.1 Physical model

As we all know, the size of heat exchanger in practical industry is very large and they commonly have complex internal structure. Therefore, it is difficult to simulate a whole heat exchanger. In view of the symmetry of structure and the periodicity of fluid flow on shellside, a simplified periodic unit duct model is built, which is shown in Fig. 1.

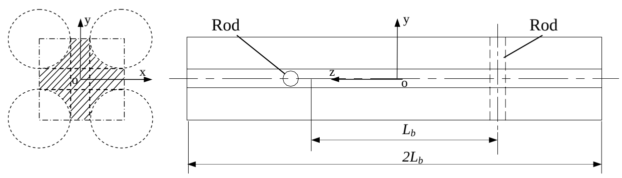
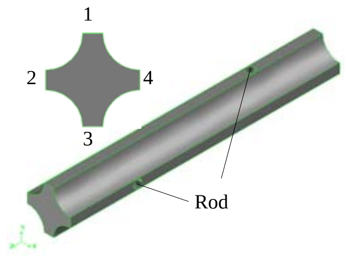


Fig. 1 Simplified periodic unit duct model

In Fig.1, the dashed area represents the flow section, L_b is the rod-baffle pitch and one periodic section has a length of $2L_b$. There are two reasons why this simplified model can be effective: firstly, the diameter of real heat exchanger is so big that the shell has little influence on the flow condition of major fluid. The major fluid flows longitudinally so the radial velocity of fluid is very small, thus the symmetry simplification can be effective; secondly, the length of real heat exchanger is so big that except the inlet and outlet area, the fluid flow in fully-developed area is periodical due to the periodicity of structure, thus the periodic simplification can be effective.

Based on the above two points, the geometric model is built in Gambit as is shown in Fig.2. Faces 1~4 are symmetry faces and the two end faces are set as periodic face. The fluid flows along positive z and passes two perpendicular rods.



Parameters:

Tube: ϕ25×2.5mm

Tube pitch: 32mm

Lb: 100,150,200,250,……

D of rod: ϕ6mm

Fig. 2 Geometric model of periodic unit duct

2.2 Governing equations and periodic flow

The governing equations of fluid flow in shellside of heat exchanger are expressed as follows:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{1}$$

Momentum equation:

$$\rho \frac{d\mathbf{u}}{dt} = \rho f + \nabla \cdot \boldsymbol{\sigma} \tag{2}$$

Energy equation:

$$\rho C_p \frac{dT}{dt} + p\nabla \mathbf{u} = (\nabla \lambda \cdot \nabla)T + \rho q - \nabla q_r + \Phi \tag{3}$$

For periodic fully-developed fluid flow, if temperature change is limited and physical property is constant, periodic flow has the following characteristics:

$$u(x, y, z) = u(x, y, z+L) = u(x, y, z+2L) = \cdots\cdots$$

$$p(x, y, z) - p(x, y, z+L) = p(x, y, z+L) - p(x, y, z+2L) = \cdots\cdots \tag{4}$$

where, $u(u, v, w)$ is the velocity vector of fluid, p is the pressure of fluid.

For periodic fully-developed heat transfer, if fluid flows along z axis, then

$$\frac{\partial}{\partial z}\left(\frac{T_{w,m} - T}{T_{w,m} - T_b}\right) = 0 \tag{5}$$

where, $T_{w,m}$ is the mean tubewall temperature; T_b is the mean temperature of fluid; T is the fluid temperature.

If tubewall temperature (T_w) is constant, defining dimensionless temperature $\Theta = \dfrac{T - T_w}{T_b - T_w}$, then

$$\Theta(x, y, z) = \Theta(x, y, z+L) = \Theta(x, y, z+2L) = \ldots$$

2.3 Mesh and boundary conditions

In view of the characteristics of structure and fluid flow, in order to make the simulation more accurate, splitting meshing method is used and the area near rod is separated to refine the mesh. 3-D boundary layer is also set on the rod face to reflect the effect of boundary layer better. The final mesh is shown as Fig. 3.

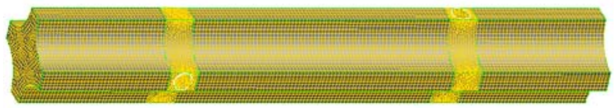


Fig. 3 Refined mesh of periodic unit duct model

Then, the mesh file is imported into FLUENT and such suppositions are made:

1) Physical properties of fluid as density, viscosity, specific heat and so on in duct is constant.
2) Fluid is incompressible, isotropic and continuous.
3) Fluid is Newton fluid.
4) The unit ducts don't influence each other along radial direction.
5) The fluid flow in periodic unit duct is periodically fully-developed and is not influenced by inlet and outlet.
6) No net mass addition and other source terms exist in the duct during the fluid flows through the duct.

After mesh is examined, segregated solver and k-ε model are selected and energy equation is included. In boundary conditions, such setups are made: material is selected; temperature of tube wall is set as

400k (equivalent to water vapor condensing in tube); upstream bulk temperature is set as 300k; operating pressure is one atm; the wall is set as no-slipping wall; the flux in duct is input; SIMPLE algorithm is selected and second order upwind is selected to discretize momentum equation and energy equations.

3 Detailed analysis of fluid flow and heat transfer in periodic unit duct model

The result of simulation when working medium is water, L_b=150mm and Re=4000 is provided to carry out the detailed analysis of fluid flow and heat transfer in the fully-developed section on shellside of HELFSF, which is shown as follows.

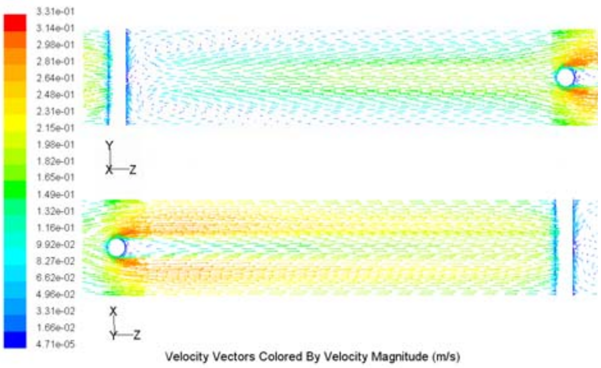


Fig.4 Velocity vector distribution on plane x=0 and y=0

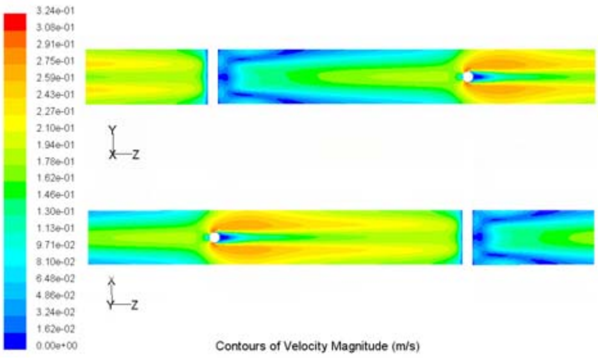


Fig.5 Velocity contour on plane x=0 and y=0 (filled)

Fig. 4 is the velocity vector distribution on plane x=0 and y=0, Fig.5 is the velocity contour on plane x=0 and y=0. From the two figures, it can be seen that when fluid passes rod, due to the decrease of flow area, high velocity fluid comes into being on both sides of the rod and rush into the space between adjacent tubes, where fluid flows fast near tube wall and thus heat transfer is enhanced. And because of the shedding of boundary layer, low pressure area comes into being behind the rod, which is shown in Fig.6. And high pressure fluid between tubes on both sides of rod flows toward low pressure area, so the turbulence and heat transfer between central duct and near tube wall are enhanced.

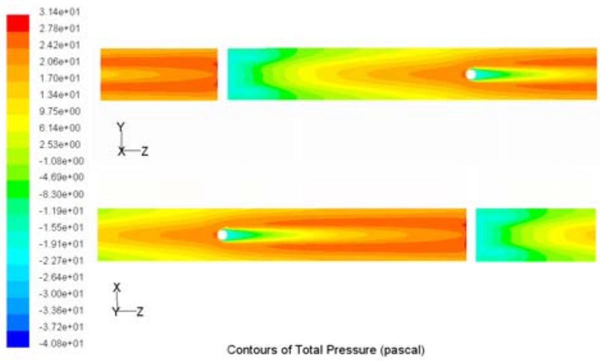


Fig.6 Pressure contour on plane x=0 and y=0

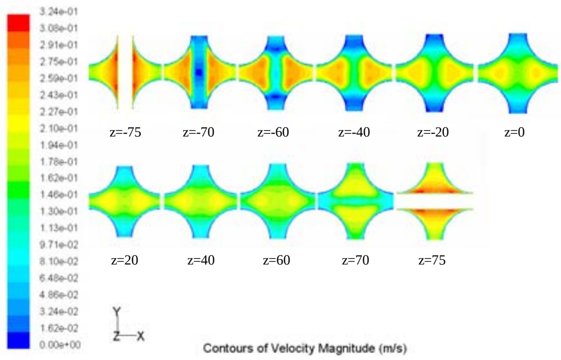


Fig.7 Velocity contour on xy planes along z(mm) axis

Fig. 7 is the velocity contour on xy planes along z axis. From this figure, the changing of flow condition in the duct can be seen clearly. At the position of z=-75mm, the rod makes the flow area decrease and high velocity fluid comes into being on both sides of the rod (shown as red area). In the space from z=-70mm to z=60mm, due to vortex and back flow, high velocity area extends to central area and the whole fluid in the duct become uniform gradually until the next rod appears. Such process repeats and the fluid flow and heat transfer in the whole shell is enhanced.

Fig. 8 and Fig.9 reflects the change of temperature field in the duct. According to the flow condition, the effect of flow condition on the heat transfer and temperature field can be seen. Where the velocity of fluid is higher, the heat transfer is faster and the temperature is higher. Right behind the rod, the effect of backflow accelerates the mixture of fluid thus the temperature here is high, which can be seen on x=0 plane in Fig.8 and z=70mm plane in Fig.9. After the fluid passes the rod and goes on, the temperature in the whole duct is heightened.

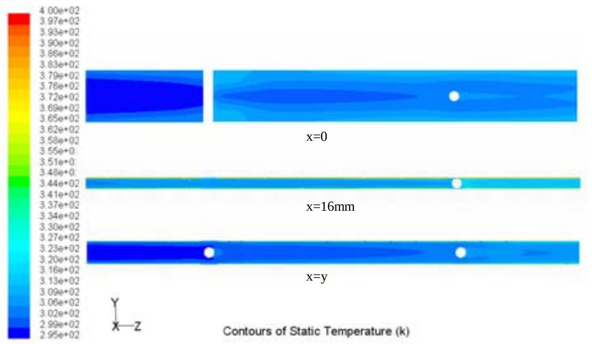


Fig. 8 Temperature contour on plane x=0, x=16mm and x=y

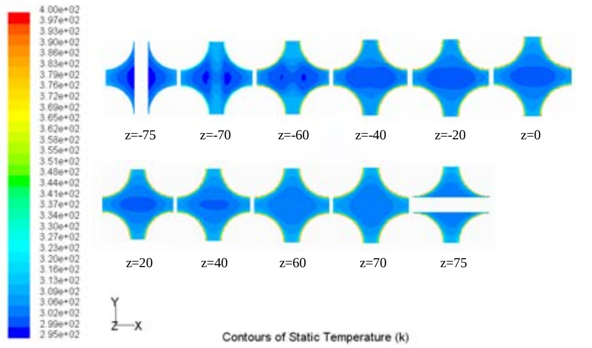


Fig. 9 Temperature contour on xy planes along z(mm) axis

Besides the above direct observation, FLUENT also provides strong postprocessing tools for us to compute some important variables such as turbulence intensity, heat transfer coefficient and pressure drop. From the knowledge of fluid dynamics and heat transfer, it is known that the turbulence intensity is a variable that reflects the pulsating degree of flow. And usually where the turbulence intensity is higher, the heat transfer is better. In FLUENT, the turbulence intensity is defined as follows:

$$I = \sqrt{\frac{2}{3}k} \Big/ v_{ref}$$

where k is turbulence momentum; v_{ref} is reference velocity, commonly equal to the mean velocity.

Fig.10 shows the coupling relationship between turbulence intensity and surface total heat flux.

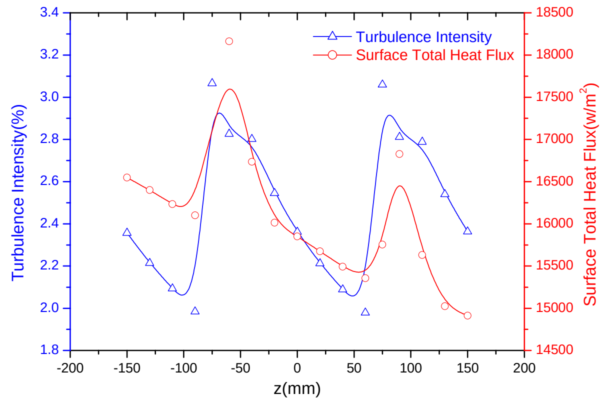


Fig. 10 Turbulence intensity and heat flux along z axis when Lb=150mm

From Fig. 10, it can be seen that when fluid passes the rods, the intensity gains great increase and decrease gradually after the fluid passes the rods. And at places where the turbulence intensity is higher, the surface total heat flux is also bigger, which shows the coupling relationship between them.

4 Optimal design of rod-baffle pitch

Applying the above simulation method, the characteristics of model under different working medium, different mass flux (or Re) and different L_b can be obtained. By using FLUENT post-processing tools, the variation of heat transfer co-efficient and pressure drop with L_b can be concluded. Fig.11 shows the variation of surface heat transfer coefficient with z coordinate when air flows at the Re of 18000 through the duct and L_b is 300mm.

From Fig.11, it can been seen that the surface heat transfer coefficient decreases along flow direction in the space between two rod baffles, which is caused by the weakening enhancement effect of rod on fluid flow after fluid passes the rod. By contrast with the model without rod, such conclusion can be drawn: at the place of z=100mm heat transfer coefficient declines to the value of model without rod baffle and the following area cannot benefit from the rod baffle, therefore, on this condition L_b is suggested not bigger than 250mm.

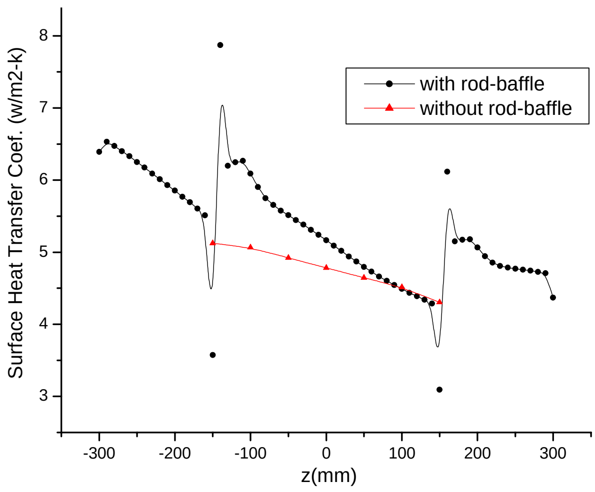


Fig. 11 Comparison of surface heat transfer coef. between model with rod baffleand model without rod baffle

By simulating models under different L_b on this working condition, heat transfer coefficient and pressure drop under a serial of L_b value is obtained as shown in Fig.12.

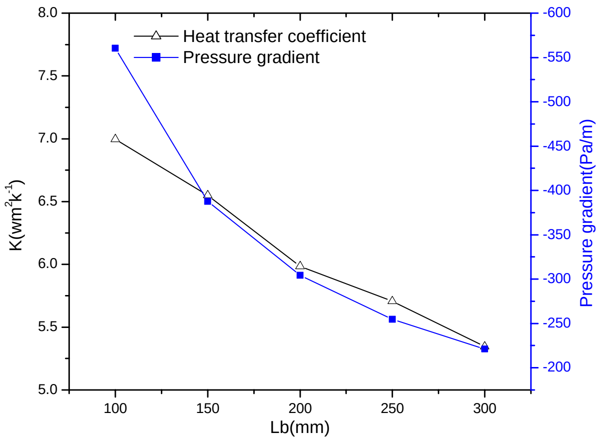


Fig.12 Heat transfer coefficient and pressure drop under different L_b

From Fig.12, it can be seen that the both heat transfer coefficient and pressure drop increase with the decreasing of rod-baffle pitch L_b. Particularly, when L_b is smaller than 150mm, the pressure drop increases more rapidly while the heat transfer coefficient has less increment. Therefore, taking full account of heat transfer and pressure drop, L_b is suggested not smaller than 150mm.

According to the above analysis, such conclusion is drawn: when air flows through the duct at Re of 18000, the value of L_b should be between 150mm and 250mm. Generally speaking, if power permits, smaller L_b is better.

Applying the same method, the optimal value range of L_b under different working media and different Re number is obtained as shown in Fig.13 and Fig.14.

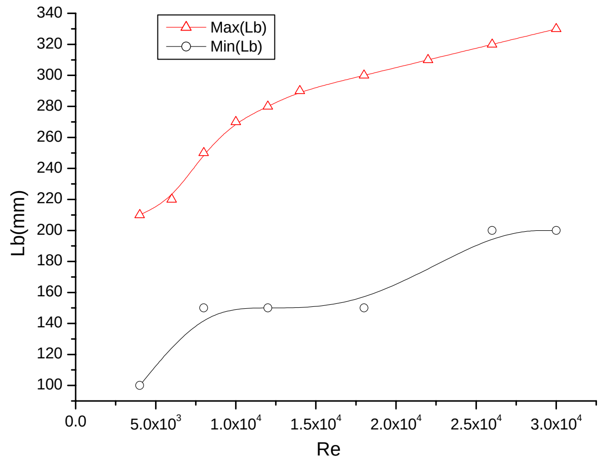


Fig. 13 The optimal value range of L_b under different Re for water

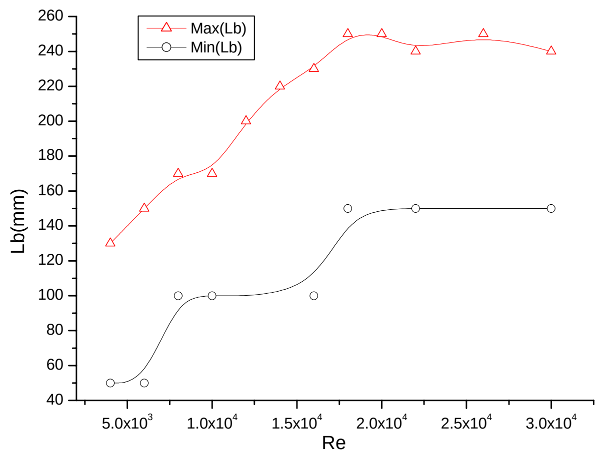


Fig. 14 The optimal value range of L_b under different Re for air

5 Conclusion

In this paper, CFD software FLUENT is employed to build the simplified model of key structure on the shellside of HELFSF. By simulation and analysis, detailed characteristics of fluid flow and heat transfer in periodic unit duct are obtained. Based on the comparison of simulation results under different geometric parameters and different working conditions, the optimal value range of rod-baffle pitch is given to instruct practical design of heat exchanger. Such method can also be used to study heat exchanger with different shellside structure.

References

[1] Dong QiWu, Liu MinShan, Guo ChaXiu, Hu QingJun, Wang XueSheng, Characteristic Research of the New Type Energy-Saving Tubular Heat Exchanger with Longitudinal Flow of Shellside Fluid[C], *Proceedings of ICEE*. London: Begell House Press, 1996, pp448-454.

[2] C.C. Gentry, ROD Baffle Heat Exchanger Technology[J], *Chemical Engineering Progress*, vol 86, No. 7, 1990, pp48-57.

[3] Yang Shimin, Tao Wenquan. *Numerical Heat Transfer .3rd ed*, Xi'an: Xi'an Jiaotong Universtiy Press, 2001.

[4] Wang Dingbiao, Application of Numerical Simulation Technology in Heat Exchanger with Longitudinal Flow of Shellside Fluid[dissertation]. Shanghai：East China University of Science & Technology, 1999.

[5] Wang Dingbiao, et al, Flow Field in Heat Exchanger with Longitudinal Flow of Shell-side Fluid. *Journal of Chemical Industry and Engineering (China)*, vol 55, No. 5, 2004, pp699-703.