2018 8th International Conference on Education, Management, Computer and Society (EMCS 2018)

Numerical Simulation of Hydrodynamics in the Diversion Region of Herringbone Plate Heat Exchanger

Shuwei Lv *
College of Mechanical Engineering
Jilin Engineering Normal University
Changchun,130052,China
851398465@qq.com

Zhuojuan Yang
College of Mechanical Engineering
Jilin Engineering Normal University
Changchun,130052,China
450818386@qq.com

Mei Yang
College of Mechanical Engineering
Jilin Engineering Normal University
Changchun,130052,China
1906231307@qq.com

***Abstract*—The effect of the geometry of the diversion region of a herringbone plate heat exchanger on the heat transfer performance is studied. Based on incompressible computational fluid dynamics numerical simulation theory a flow path model with a radius R=140 mm, an inclination angle φ=30°, and a spacing h=20 mm herringbone plate heat exchanger in the diversion zone and the heat exchange zone was designed. The numerical simulation software FLUENT was used to complete the entrance. The flow velocity characteristic data of 0.5 m/s, temperature of 350 K, outlet pressure of 101,325 Pa and reflow temperature of 300 K were calculated and analyzed. The results show that the average wall heat flux is 49.5 J, the average wall temperature is 339.6 W, the average fluid temperature is 339.5 W, and the average fluid velocity is 1.74 m/s. Through data compilation and calculation, the evaluation of heat transfer parameters is 131.22, and the evaluation of pressure drop parameters is 194,788 pa, which provides some basis for the design and optimization of herringbone plates.**

Keywords*—*Herringbone plate heat exchanger diversion area, Plate geometry parameters, Heat transfer performance, FLUENT

I. Introduction

The plate heat exchanger is a high-efficiency heat exchanger formed by stacking a series of metal plates with a certain corrugated shape. A plurality of flow passages with small flow sections are formed between the plates of the heat exchanger, and heat exchange is performed through. An important indicator for measuring plate heat exchangers in industry is heat transfer performance[1].The ripple shape of the existing plate heat exchanger plates at home and abroad is mostly herringbone shape. Most researchers have done research on herringbone corrugated plates [2].Lee [3] simulated the unit heat transfer node of the herringbone plate heat exchanger, and considered that the direction of fluid flow near the concave and convex plate is different. When the wave dip angle is large, the flow direction will seriously affect the flow structure, resulting in low Reynolds numbers achieve turbulent motion. Cui Lizhen [4]calculated and analyzed the mainstream area of herringbone corrugated plate heat exchanger, and used the calculated results to fit the relationship curve between the geometric parameters and Nusselt number and pressure drop . Liu Yang [5] used FLUENT software to analyze the two important structural parameters affecting the heat transfer and flow performance of the plate heat exchanger: the normal wave intercept, the included angle of the corrugation were numerically analyzed, and the temperature field and flow under different parameters were compared. The field and pressure fields show that decreasing the normal intercept or increasing the angle of the corrugation facilitates heat transfer enhancement, but also increases the flow resistance.

In order to study the influence of the geometric structure of the diversion region of the herringbone plate heat exchanger on the heat transfer performance, the computational fluid dynamics simulation of the diversion region of the herringbone plate heat exchanger was completed and the velocity distribution, temperature distribution and pressure distribution of the internal flow field were obtained. data. The simulation results were analyzed, the heat transfer performance parameters were evaluated and evaluated, and the pressure drop parameters were evaluated. Provides some basis for the design and optimization of the plate.

II. Simulation calculation model

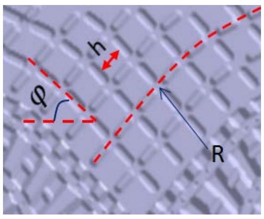

Fig. 1 Model parameters

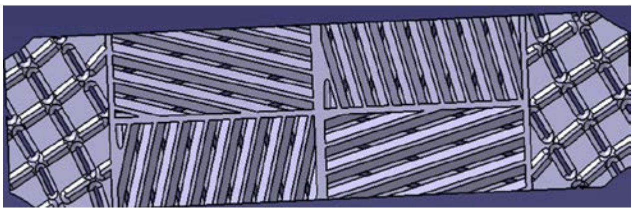

Fig. 2 Plate model

Draw a herringbone stamped sheet in the fast diversion region, as shown in Fig. 1. Specified plate model parameters: R is the radius of the arc, is the angle of inclination, is the peak spacing, these three parameters are the main factors affecting the heat transfer performance of the plate-type heat exchanger in the rapid diversion zone[6]. In the computational fluid dynamics analysis process for this problem, the heat transfer and pressure drop conditions of the fluid in this model are studied, and the flow channel model that can fully reflect the three parameters in the fast flow diversion region of the heat exchanger, ie, the

  DOI: 10.25236/emcs.2018.132

above factors, is selected. According to the heat exchange principle of the plate heat exchanger, an effective heat exchange area plate model of the heat exchanger is constructed, and the above-mentioned plate model is drawn by using a three-dimensional design software, as shown in Fig.2 .

III. SET SIMULATION PARAMETERS

A. Grid Division.

The object of the study is a herringbone plate heat exchanger plate with a trapezoidal groove. The effect of the corrugated geometry of the plate's rapid flow zone on the heat transfer performance is analyzed. The diversion area enters the heat exchange area again. An internal fluid model consisting of two plates with an arc radius of R=140mm, dip angle φ =30°, and peak spacing h=20 mm is optionally selected. The section is 200 mm long and 80 mm wide, and the internal fluid model is introduced into the calculation fluid. Dynamic Analysis Pretreatment Software.

The rationality of meshing is the key to solve the finite element problem. Therefore, the tetrahedral mesh is used to mesh the internal fluid model, and the partial area is refined. According to the size of this model, a grid with a size of 0.0001 is established. The maximum grid is 0.003. After refinement, a total of 466,957 grids are divided. After meshing, the internal fluid model is shown in Figure 3. According to the physical process of actual heat exchange of the heat exchanger, the boundary conditions such as inlet, outlet, and four closed wall surfaces are set.

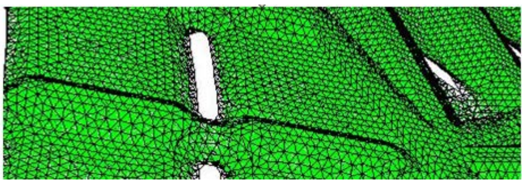

Fig. 3 Fluid fluid model

B. Parameter Setting.

The internal fluid grid file is imported into the computational fluid dynamics software. The structure of the model and the division of the grid are all three-dimensional. Enter the following specific values to simulate the simulation state of the fluid in this flow channel, calculate the specific values and analyze the results, as shown in Table 1:

TABLE 1 INPUT PARAMETERS

Entrance	Velocity	0.5 [ms^{-1}]	Export	Pressure	101325[pa]
	Temperature	350 [K]		Temperature	300[K]
Heat Exchange Wall Boundary Conditions	Free flow temperature	300 [K]	Adiabatic wall boundary conditions	Free flow temperature	0 [K]
	Heat exchange rate	50 [$w/m^2.K$]		Heat exchange rate	0[$w/m^2.K$]
Fluid properties	Density	1000 [kg/m^3]	Specific heat capacity	4182 [J/kg.K]	
	Thermal conductivity	0.67 [w/m.K]	Viscosity	0.001003 [kg/m.K]	

IV. COMPUTATIONAL FLUID DYNAMICS SIMULATION

The use of numerical simulation software FLUENT to calculate the basic process is as follows [7-9]: According to the idea of finite element calculations, the input physical state parameter is known, set the boundary conditions and other physical conditions, using the pressure solver speed , temperature, pressure, and other unknowns are solved. The solution process is: The solver solves the equation containing a single variable on all the grids, obtains the solution of the equation, and solves the solution of another parametric equation by solving the loop. Because this equation is a nonlinear control equation, and they are coupled to each other, multiple operations can be performed. The iterations eventually converged the equations and the calculation process ended.

A. Iterative Solution.

Each iteration consists of the following steps[10].

I. Based on the results of the previous round of analysis, the liquidity variable is updated again.

II. Find the solution of the momentum equation to obtain the velocity field.

III. If the velocity solution does not satisfy the continuous equation, a poisson-type pressure correction equation is constructed using a momentum equation and a continuous equation, and the pressure correction equation is solved to obtain the corrected values of the velocity field and the pressure field.

IV. Uses the correction values of the velocity field and the pressure field to solve the temperature control equation.

V. Check the convergence of the system of equations. If the equations do not converge, repeat the convergence process.

The model convergence criteria are as follows:

I. The residual I satisfies the requirements. The parameters are: energy equation 10-6, other parameters: 10-4.

II. In the calculation area , with the iteration, the speed, temperature, and pressure do not continue to change.

III. Exit, with the iteration, speed, temperature, pressure does not continue to change.

IV. The calculation results satisfy the conservation of the mass and the conservation of energy.

The residual plot was activated and iteratively solved. After iteration 151, the data converged. The convergence process curve is shown in Fig.4.

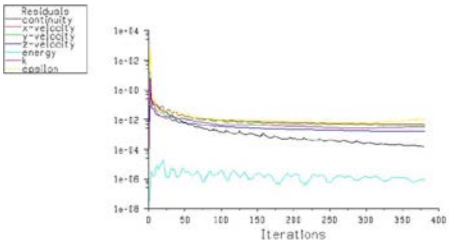

Fig. 4 Data convergence process curve

B. Numerical Simulation Results.

After numerical calculations, the velocity distribution, temperature distribution and pressure distribution data of the internal flow field were obtained. Among them: the profile of the velocity profile of the internal fluid is shown in Fig.5, and the full model is shown in Fig.6. Water flows in the positive direction from the Y-axis and flows out from the other side. From red to blue, the velocity of the fluid in the flow channel is high to low. As can be seen from the figure, the speed is lowest around the corrugated intersecting grooves of the two plates, and there is no intersecting groove along the fluid inflow direction and the flow path is high.

After numerical calculations, the velocity distribution, temperature distribution and pressure distribution data of the internal flow field were obtained. Among them: the profile of the velocity profile of the internal fluid is shown in Fig.5, and the full model is shown in Fig.6. Water flows in the positive direction from the Y-axis and flows out from the other side. From red to blue, the velocity of the fluid in the flow channel is high to low. As can be seen from the figure, the speed is lowest around the corrugated intersecting grooves of the two plates, and there is no intersecting groove along the fluid inflow direction and the flow path is high.

Fig. 5 Cross-section model of velocity distribution in flow field

Fig. 6 Full model of velocity distribution in flow field

The pressure distribution cross-section model diagram of the internal flow field is shown in Fig.7, and the full model Fig.8 is shown. Red to blue in the figure represents the pressure from high to low. The pressure drop changes in the direction of flow, and the pressure in the positive X direction is high. A pressure drop is formed in the chute at the inlet.

Fig. 7 Cross-section model diagram of flow field pressure distribution

Fig. 8 Full model diagram of flow field pressure distribution

The cross-section model of the temperature distribution in the inner flow field is shown in Fig.9, and the full model is shown in Fig.10. The color from red to blue, in turn, represents the temperature from high to low.

Fig. 9 Cross-section model of temperature distribution in flow field

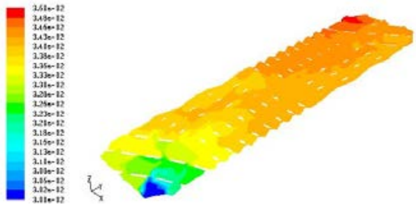

Fig. 10 Full model of temperature distribution in flow field

C. Data Processing.

Derive the internal fluid model to calculate the value of each node, import the numerical calculation software to perform numerical calculation, find the average wall heat flux (q), the average wall surface temperature (Tw), the average fluid temperature (Tf), the average fluid velocity (u), specific values such as the table II shows.

TABLE II ANALYSIS PROCESS PARAMETERS AND ANALYSIS RESULTS

Equivalent diameter（d_e） 0.006 [m]	Coefficient of friction（f） 0.2
Average wall heat flux（q） 49.5 [J]	Flow path length （L） 0.05 [m]
Average wall temperature（T_w）339.6 [K]	Average fluid temperature （T_f）339.5 [K]
Density （ρ）1000 [kg/m^3]	Thermal conductivity（λ）0.67 [w/m.K]
Average fluid velocity （μ）1.74 [m/s]	

Analyze the simulation results, solve and evaluate the heat transfer parameter values Nu, and evaluate the pressure drop parameters Δp.

$$N\mu = \frac{q d_e}{\lambda (T_w - T_f)} = 131.22$$

Δp=194766.0 (pa)

V. CONCLUSION

According to the parameter range of plate-type heat exchanger plate rapid diversion region, an internal fluid model consisting of two plates with an arc radius R=140 mm, dip angle=30°, peak spacing h=20 mm, fast diversion zone and heat exchange zone was established. An internal fluid model with a length of 200 mm and a width of 80 mm was intercepted and used as a fluid dynamics simulation object. After computational fluid dynamics analysis, the velocity distribution, temperature distribution, and pressure distribution data of the internal flow field were obtained. The data showed that: Evaluation of heat transfer parameter values Nu is 131.22, the evaluation pressure drop parameter Δp is 194,788 pa, which provides some basis for the next chevron design and optimization work.

REFERENCES

[1] W.H. Tao:Numerical heat transfer [M]. Xi'an: Xi'an Jiaotong University Press, 1995.(In Chinese).

[2] Wang Hao, Han Huaizhi, Li Bingxi. Flow and heat transfer mechanism of corrugated passage in plate heat exchanger[J]. CIESC Journal, 2017, 68(S1).(In Chinese).

[3] LEE J Y,LEE K S. Flow characteristics and thermal performancein chevron type plate heat exchangers [J].International Journal of Heat and Mass Transfer,2014,78:699-706.

[4] Xu Zhiming, Wang Yueming, Zhang Zhongbin. Numerical simulation of the performance of plate heat exchangers[J]. Chinese Journal of Power Engineering, 2011, 3:189~202.(In Chinese).

[5] LIU Yang. Numerical analysis of plate heat exchanger using fluent software[J]. Guangzhou Chemical Industry,2013, 41(13)：221-223.(In Chinese).

[6] Yang Chonglin. Plate heat exchanger engineering design manual [M]. Beijing: Mechanical Industry Press, 1998: 78-82.(In Chinese).

[7] M AMehrabian,R Poulter．Hydrodynamics and thermal characteristics of corrug channels：computational approach[J]．Applied Mathematical Modeling,2000.24:343~364.

[8] Koen Grijspeer,Birinchi Hazarika,Dean Vucinic．Application of computational fluid dynamics to model the hydro dynamics of plate heat exchangers for milk processing [J]． Journal of Food Engineering. 2003.

[9] Wu Jinxing, Wei Xinli et al. Performance Analysis of Baffle Bar Heat Exchanger Based on CFD Technology [J]. Light Industry Machinery. 2006.6.Vol. 24, No. 2:61~63.(In Chinese).

[10] Lv Shuwei. Analysis of flow characteristics in the plate diversion zone of plate heat exchanger (Ph.D.,Changchun City, Jilin Province, 2055 Yan'an Street: Changchun University of Technology,China 2015),p27.