Study on the Effect of Cooling Air Flow on Convection Heat Transfer of Helical Tube

Zihao Li, Fei Yao, Bo Li

Zhengzhou University of Aeronautics ZUA, China

Abstract

Spiral tube convection heat exchanger has the advantages of compact structure, self-cleaning, high heat transfer efficiency and low cost, so it is widely used in chemical, power, power and other industries. In this paper, a spiral tube heat exchanger is modeled by UG software, and then the tetrahedral mesh is divided by ICEM software. FLUENT is used to simulate the convection heat transfer performance of spiral tube heat exchanger under different cooling air flow conditions. The air flow rule and heat transfer coefficient in the spiral tube heat exchanger are studied and analyzed. This paper provides theoretical basis and data support for the calculation and design of the helical tube heat exchanger, and has a certain application value.

Keywords

Spiral Tube Convection Heat Exchanger; FLUENT Simulation; Convection Heat Transfer Coefficient.

1. Introduction

Spiral tube heat exchanger has the advantages of small size, large heat transfer coefficient, high heat transfer efficiency, compact structure, self-cleaning, etc. It is widely used in industrial equipment such as chemical industry, power, low temperature, power, petroleum and nuclear energy. The helical tube heat exchanger has a special structure. When the fluid flows through the heat transfer tube, it forms secondary flow, which enhances the turbulence effect and can enhance heat exchange and cleaning. Therefore, it has the advantages of compact structure and high heat exchange performance [1]. Therefore, it is necessary to strengthen the research on the flow and heat transfer of helical tubes.

A lot of research has been done on how to improve the efficiency of heat exchangers and the change of energy in the process of convection heat exchange. The research on flow-induced vibration of helical tube heat exchanger in literature [2] provides a reference for future design of helical tube heat exchanger. Reference [3] Tests on heat transfer performance of Y-type and K-type tube heat exchangers show that Y-type heat exchangers have larger total heat transfer coefficient, lower shell side pressure drop and higher shell side convection heat transfer coefficient within the test range and under the same Re condition by means of control variables. The results show that the comprehensive performance of Y-type heat exchangers is better than that of K-type heat exchangers. It provides a basis for the selection of types of spiral tube heat exchangers. Document [4] Through the study of design, manufacture and heat transfer mechanism, the spiral tube convection heat exchanger plays an increasingly important role in the practical application in the fields of chemistry, petroleum chemistry, power generation, metallurgy, food, pharmaceuticals and other fields. After the introduction of a new type of heat exchanger, the spiral tube heat exchanger has played a significant role in the production and processing efficiency. It has contributed to the popularization and application of helical tube convection heat exchanger. Reference [5] The heat transfer and resistance properties of helical elliptical tube heat exchanger are analyzed by CFD numerical simulation method. After simulating elliptical helical tubes with different structures under different working conditions, the existing

ISSN: 2414-1895 DOI: 10.6919/ICJE.202204_8(4).0077

criteria correlation is modified to improve the accuracy of criteria correlation. Document [6] By applying the helical tube heat exchanger to the solar-air source composite heat pump system, and analyzing the variation law of each parameter in the whole storage experiment through the experimental data, the storage characteristics of the heat exchanger are finally summarized. Literature [7] By studying the flow characteristics and resistance characteristics of the helical tube heat exchanger with coaxial annular channel, and comparing the theoretical values, the influence of fluid flow rate, pressure, flow rate and model geometry parameters on the flow characteristics of the coaxial annular channel is finally obtained. At the same time, it is pointed out that large errors will be caused by the resistance coefficient of the circular pipe, which provides a reference for the relevant experimental research. Document [8] calculates the flow field on the side of the shell of the component helical tube heat exchanger by using the hydrodynamic calculation method and simplifying and assuming. The results show that the resistance caused by the arrangement of three different components is different, which provides a reference for the study of resistance in the helical tube. Document [9] simplifies the design and calculation of spiral tube heat exchanger by using C# language program, which effectively improves the efficiency of calculation in design. At the same time, the reliability of calculation is verified by comparing the results of manual calculation, which greatly improves the efficiency of related calculation and further optimizes the design of spiral tube heat exchanger. Literature [10] The heat transfer characteristics of a new type of heat exchanger, spiral coiled finned tube heat exchanger, under dry surface conditions are studied experimentally. The effect of the inlet conditions of two workflows flowing through the heat exchanger on the heat transfer coefficient is discussed. The results show that the efficiency of the heat exchanger increases with the increase of water quality and flow rate. It also increases slightly with the increase of the inlet temperature. Literature [11] By investigating the effect of tube spacing on the heat transfer and pressure drop characteristics of two-phase flow on the shell side of LNG spiral wound heat exchanger, propane was used as the medium to set relevant parameters and analyze them. The results show that with the increase of pitch between longitude tubes, the heat transfer coefficient and pressure drop both decrease with a smaller deviation from the predicted data. It provides a reference for studying the influence of radial pitch of helical tube on heat transfer coefficient.

In summary, simulation calculation and experimental study are mostly carried out on the flow rate, temperature and material of the helical tube, while the structural optimization of the helical tube convection heat exchanger is less studied. This paper provides theoretical basis and data support for the calculation and design of the helical tube heat exchanger, and has a certain application value.

2. Physical Model Establishment and Griding of Helical Tube Heat Exchanger

In this paper, UG three-dimensional modeling software is used to build the physical model of spiral tube heat exchanger. It is a very powerful modeling software with the advantages of simple operation, easy modification and many modules. It can easily construct various complex entities and shapes..

2.1 Model Establishment

The spiral tube convective heat exchanger studied in this paper is mainly composed of two parts: cylinder and spiral tube. The cooling fluid flows in through the cylinder inlet, and the hot fluid flows in through the spiral pipe inlet. The cooling fluid inlet and the hot fluid inlet are located at both ends of the heat exchanger respectively, which is a countercurrent heat exchanger. In addition, in order to facilitate the inlet and outlet of the fluid in the spiral pipe, the inlet and outlet of the spiral pipe are also considered in the modeling, that is, the pipe bends inward through the inner wall of the cylinder. The whole spiral tube convection heat exchanger model is shown in Figure 1.

ISSN: 2414-1895 DOI: 10.6919/ICJE.202204_8(4).0077

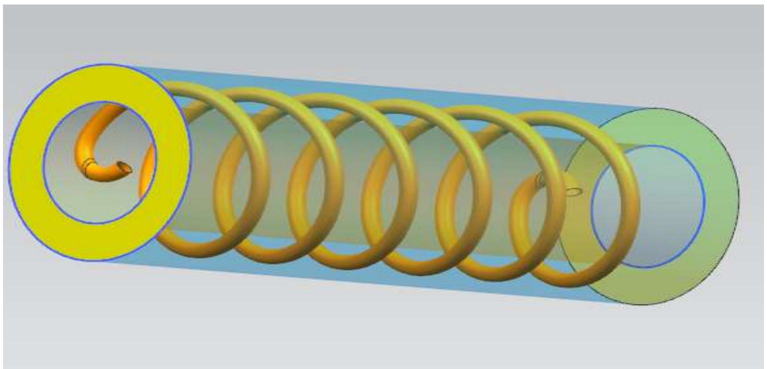

Figure 1. Spiral tube heat exchanger model

2.2 Grid Division

(1) Model import and processing

1) Model import. Export UG model file format to X_ T format and import ICEM.

2) Model repair. Since there are only faces after importing ICEM, use the repair model function to check the model and repair the missing point and line features of the model.

3) Create a body. The research object of this paper is the fluid in spiral tube and cylinder. Use the two-point method to establish the body in the spiral tube and the cylinder respectively: take two points in the spiral tube to establish fluid1 respectively; Take two points in the cylinder to establish fluid2.

4) Part division. The inner and outer pipe walls of the cylinder, the inlet and outlet of the cylinder, the wall surface of the spiral pipe and the inlet and outlet of the spiral pipe are named and established according to different types of boundaries.

(2) Meshing method and process

The specific operation and process of grid division are as follows:

1) Global mesh parameter settings. Set the global mesh size and volume mesh parameters respectively.

2) Division of face mesh. Select each face on the model and set the mesh size reasonably according to the model size, run the mesh generation of the face and check whether the mesh generation is reasonable.

3) Meshing of cylinder. Select volume fluid2, select Volume 2 and the part contained in the spiral tube in the body, set parameters, and run volume meshing.

4) Spiral tube meshing. Close the completed cylinder grid, select fliud1 and part of spiral pipe, set parameters and run grid division.

5) Output grid. After meshing, output the solver, select ANSYS fluent and save it. The model after meshing is shown in Figure 2.

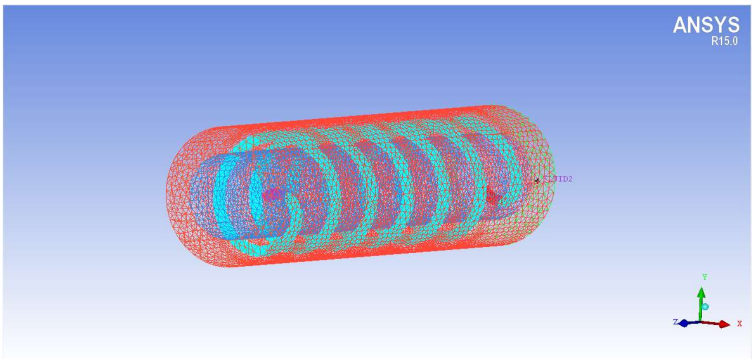

Figure 2. Model grid

 Volume 8 Issue 4, 2022
DOI: 10.6919/ICJE.202204_8(4).0077

3. Simulation Calculation

3.1 Model Simplification

The following assumptions are made for the numerical simulation of the helical tube heat exchanger:

(1) The materials of heat exchangers are uniform.

(2) The heat exchanger is mainly convection heat transfer in the simulation process, ignoring radiation heat transfer and natural convection heat transfer.

(3) The wall of the fluid passage is an adiabatic interface, and the wall is smooth. There is no slip when the fluid flows on the wall.

(4) The system is stable, the fluid is incompressible, and the intensity of turbulence is uniform at all points, ignoring the influence of gravity.

3.2 Simulation Calculation Method

(1) Conservation equations of flow and heat transfer

In the simulation calculation of spiral tube heat exchanger, its conservation equations need to be solved. Among them, the mass conservation equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

Momentum conservation equation:

$$\begin{aligned}
\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)&=F_x-\frac{\partial p}{\partial x}+\eta\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)\\
\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)&=F_y-\frac{\partial p}{\partial y}+\eta\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)\\
\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)&=F_z-\frac{\partial p}{\partial z}+\eta\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)
\end{aligned} \tag{2}$$

Energy conservation equation:

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}=\frac{\lambda}{\rho C_P}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right) \tag{3}$$

Formula: u is the x-axis velocity component, v is the y-axis velocity component, w is the z-axis velocity component, ρ for air density, C_{p} for constant pressure specific heat capacity, λ Is the thermal conductivity.

(2) Calculate the Reynolds number of a fluid to determine whether it is laminar or turbulent.

$$Re=\frac{\rho V \mathrm{d}}{\mu}=\frac{q_m d}{A\mu} \tag{4}$$

 Volume 8 Issue 4, 2022
DOI: 10.6919/ICJE.202204_8(4).0077

Formula: q_m for fluid mass flow, d for hydraulic diameter, A for fluid inlet area, μ for air viscosity.

(3) Setup of boundary conditions

In this simulation experiment, mass flow inlet and free outlet are used for air inlet and air outlet. The formulas for calculating the entrance parameter turbulence intensity and hydraulic diameter are as follows:

Turbulence intensity:

$$I = 0.16Re^{-\frac{1}{8}} \tag{5}$$

Hydraulic diameter:

$$d_e = \frac{4A}{C} \tag{6}$$

In formula A is the cross-sectional area of the fluid-filled part and C is the wet perimeter of the fluid.

3.3 Solver and Residual Setting

According to the research content and condition parameters of this paper, due to the low speed, the solver adopts the pressure based solver, the algorithm selects the simple algorithm, the difference format adopts the second-order upwind format, and the relaxation factor adopts the default setting. According to the research content and condition parameters of this paper, set the iteration steps of the numerical simulation results to 2000, and then the calculation results can converge without further calculation.

4. Data Analysis

4.1 Cloud Image Analysis When Cooling Fluid Flow is 0.5kg/S

Due to the inconvenient display of the three-dimensional model, two special sections are used in the analysis of flow and heat transfer: the section of the cylinder along the central axis, that is, the longitudinal section; The transverse section in the middle of the cylinder, i.e. the cross section.

Open the data on the two sections in Tecplot software to view the distribution of temperature, pressure and speed. When the mass flow of cooling fluid is 0.5kg/s, the pressure, velocity and temperature nephogram are shown in figures 3, 4 and 5 respectively.

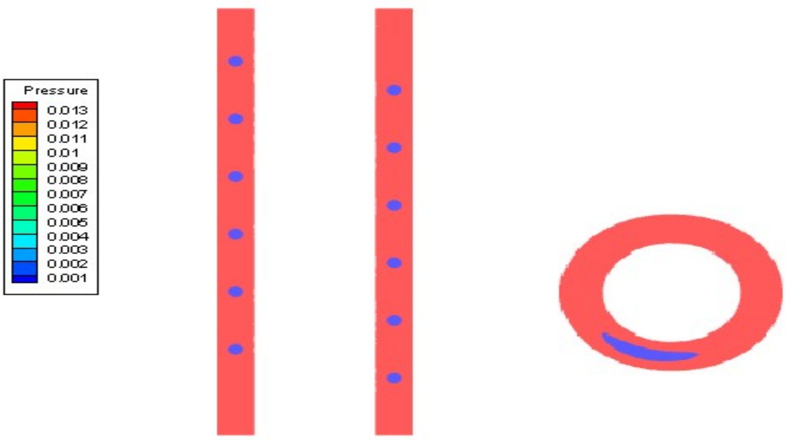

Figure 3. Cloud chart of 0.5kg/s pressure

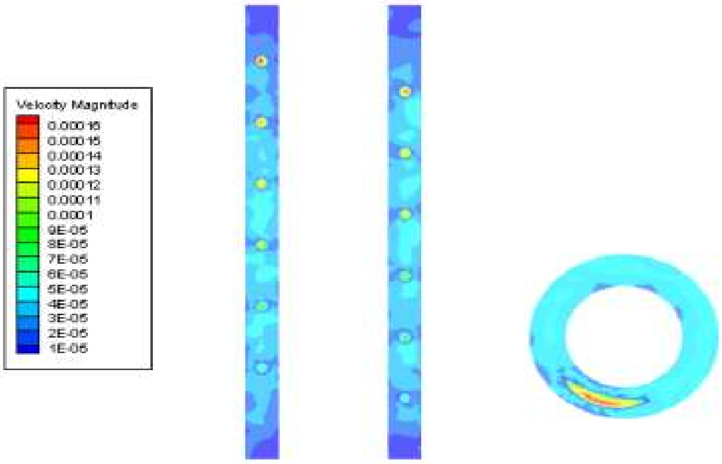

Figure 4. Cloud chart of 0.5kg/s speed

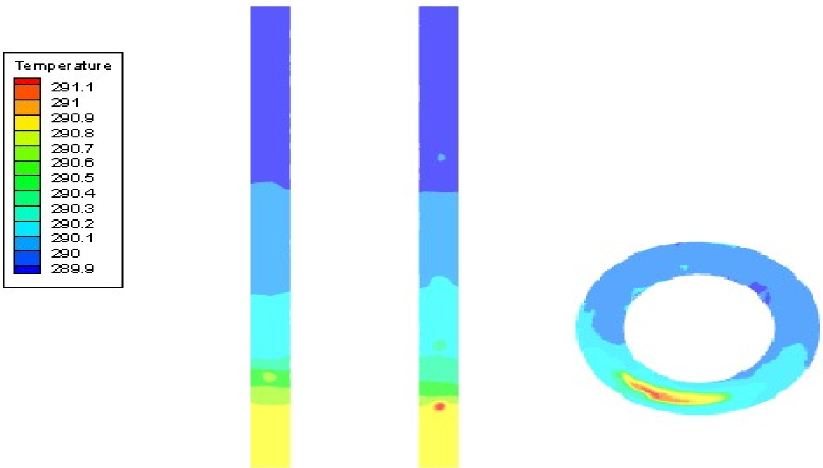

Figure 5. Cloud chart of 0.5kg/s temperature

It can be seen that when the cooling air flow is 0.5kg/s, the zoning of cold and hot fluid velocity and pressure nephogram on the two sections is obvious, but the values change little; The range of temperature change is obvious. The temperature of cold fluid increases gradually from inlet to outlet along the fluid flow direction, that is, the longitudinal section. The level is obvious. On the cross section, the temperature on one side close to the heat source (spiral tube) is higher and the temperature on the other side is lower. Therefore, the subsequent comparative analysis of cloud images mainly focuses on temperature cloud images.

4.2 Comparative Analysis of Temperature under Different Cooling Mass Flow

In this paper, the flow and heat transfer characteristics of cooling fluid with different inlet flow are simulated. The inlet flow of cooling fluid is 0.5kg/s, 1.0kg/s and 1.5kg/s respectively, and the temperature is 288.15k; The inlet flow of hot fluid is 0.05kg/s and the temperature is 450k. Through calculation and analysis, when the inlet flow of cooling fluid is different, the temperature cloud distribution on the longitudinal section and cross section are shown in Fig. 6 and Fig. 7 respectively.

 Volume 8 Issue 4, 2022
DOI: 10.6919/ICJE.202204_8(4).0077

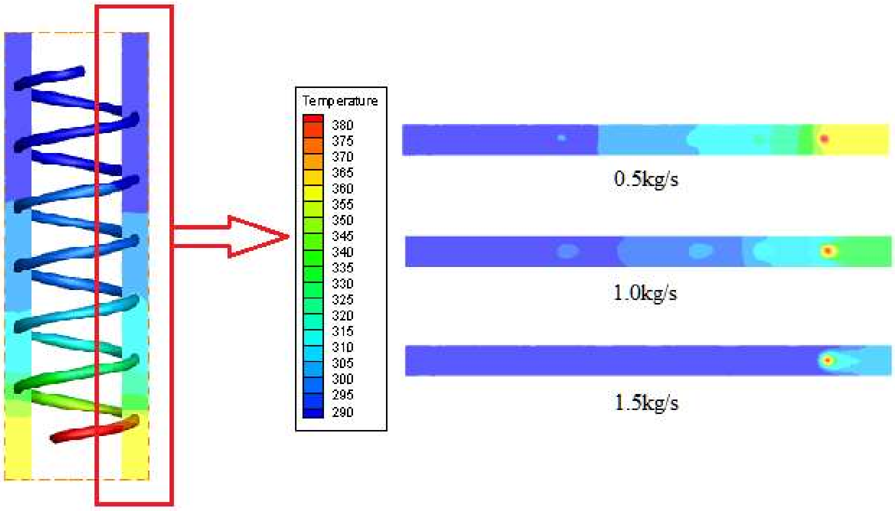

Figure 6. Temperature cloud diagram under different cooling air flow (longitudinal section)

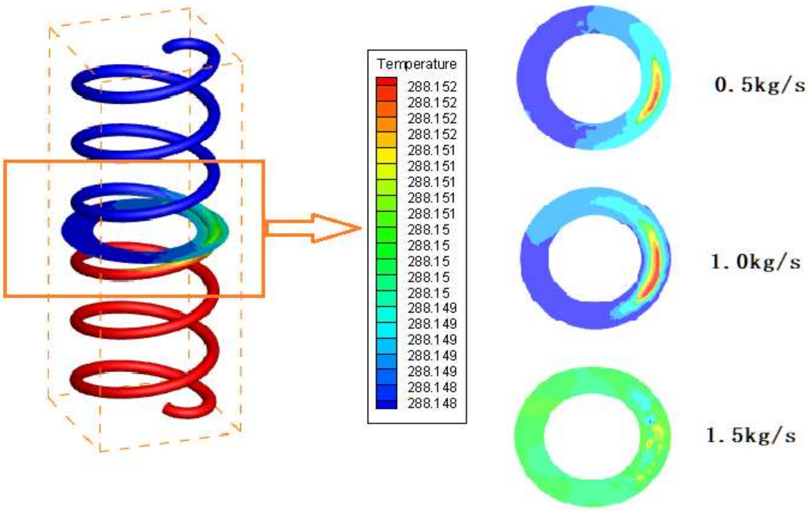

Figure 7. Temperature cloud diagram (cross section) under different cooling air flow

It can be seen that when the cold fluid flow is 1.5kg/s, the heat transfer effect of spiral tube convective heat exchanger is the best; When the fluid flow is 0.5kg/s, the heat transfer effect of spiral tube convective heat exchanger is the worst. This is because the larger the flow, the stronger the turbulence intensity, the faster the heat entering the cold fluid can be discharged, and the more obvious the convective heat transfer effect is. When the cooling fluid flow is 1.5kg/s, its Reynolds number Re is the largest. The violent movement of the air in the heat exchanger makes the cold air fully contact and exchange heat with the heat source, and the heat exchange effect is the best. The factors affecting the convective heat transfer coefficient include heat transfer temperature difference, medium type, cooling fluid velocity, turbulence and laminar flow. The fluid is in turbulent state, and its air stratification is serious. The stratification of 1.0kg/s is slightly better than 1.5kg/s. The reason is that the turbulence intensity of 1.5kg/s is slightly higher, resulting in poor stratification.

4.3 Curve Comparison and Analysis

For the data in the cloud map, the parameters of some points, including position coordinates, pressure and temperature, will be taken, and the broken line diagram will be drawn and analyzed by Origin software. Select multiple points (along the flow direction of cooling fluid) on the cross section of the cloud diagram, and input their coordinates and corresponding temperatures into the origin table, and finally obtain the temperature broken line diagram of cooling fluid with different inlet flow, as shown in Figure 8.

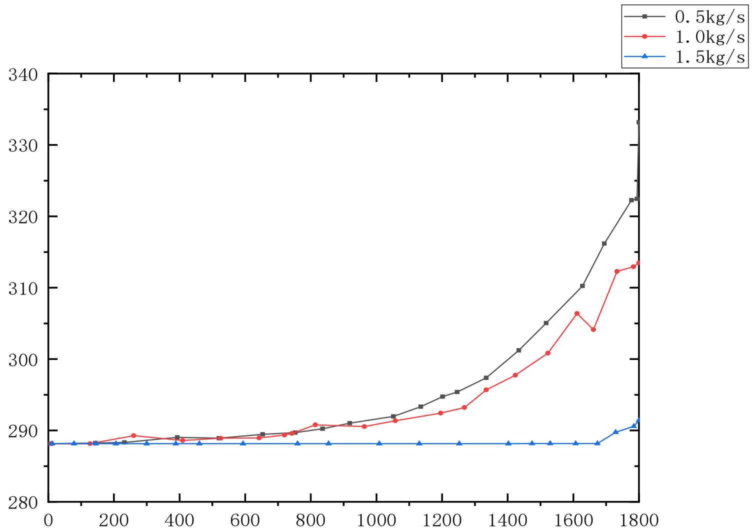

Figure 8. Broken line diagram of cooling fluid temperature under different inlet flow

On the whole, it can be seen from the figure that the convective heat transfer process is divided into three stages: the temperature change rate of cooling air at 0-1000 mm is the lowest, because the cold fluid has not yet flowed to the position of spiral pipe, and the flow and heat transfer is weak; There is a drastic temperature change at 1000-1600 mm. At this stage, the cooling air has a drastic convective heat transfer with the heat source, resulting in a rapid rise in the temperature of the cooling fluid; At 1600-1800 mm, the cold fluid is continuously heated due to its proximity to the heat source, and the temperature change is the largest. In contrast, when the inlet flow is 1.5kg/s, the overall temperature change is the smallest, the temperature of hot fluid is the lowest, and its heat exchange performance is the strongest.

5. Conclusion

In this paper, the convective heat transfer coefficient of spiral tube heat exchanger is studied and analyzed according to the parameter of cooling fluid inlet flow. The physical model, ICEM software and fluent software are established through UGNX for grid division and simulation calculation, and the post-processing of simulation data is carried out through Tecplot and Origin software. The following conclusions are drawn:

(1) Under the same other conditions, the higher the turbulence intensity of the cooling medium, the better the heat transfer effect.

(2) Under the same other conditions, the greater the mass flow of cooling medium, the greater the pressure in the heat exchanger.

(3) Under the same other conditions, the greater the turbulence intensity of the cooling medium, the less obvious the stratification in the heat exchanger and the vortex will appear.

References

[1] Liu Yan, Kong Qiaoling, Huang Weiguang, et al Experimental study on convective heat transfer of spiral tube molten salt helium heat exchanger [J] Journal of solar energy, 2019, 40 (08): 2266-2271.

[2] Wu Jianbang, Zheng Mingguang, Wang Minglu, et al Study on flow induced vibration of spiral tube heat exchanger [J] Atomic Energy Science and technology, 2020, 54 (02): 250-256.

[3] Liu Shaolin, Wu Jinxing, Tian Qianhui Experimental study on heat transfer performance of two types of wound tube heat exchangers [J] Pressure vessel, 2018, 35 (03): 8-13.

[4] Xu Xuecai Design, manufacture and engineering application of spiral wound tubular heat exchanger [J] Pharmaceutical engineering design, 2012, 33 (06): 1-3.

[5] Wei Lifeng, Gao Wenzhi, Li Yong, et al Study on heat transfer and resistance characteristics of spiral elliptical tube heat exchanger [J] Chemical engineering, 2019, 47 (04): 23-28.

[6] Dai Yuande, Yi Yong, Wen Hong Cold storage characteristics of spiral tube phase change energy storage heat exchanger [J] Science, technology and engineering, 2016, 16 (27): 141-146.

[7] Zhao Zhenxing, Yang Zhen, Guo Qinqin, et al Experiment on flow characteristics of coaxial annular channel of spiral tube heat exchanger [J] Journal of Chongqing University, 2011, 34 (01): 116-120.

[8] Shen Yaou, Duan riqiang, Jiang Shengyao Calculation of shell side flow field of modular spiral tube heat exchanger [J] Journal of Tsinghua University (NATURAL SCIENCE EDITION), 2009, 49 (03): 396-398.

[9] Xiang Pengfei Optimal design of spiral wound tube heat exchanger [D] Wuhan University of engineering, 2015.

[10] Kwanchanok S, Somchai W. Heat transfer characteristics of a new helically coiled crimped spiral finned tube heat exchanger[J]. Heat and Mass Transfer, 2009, 45(4): 381-391.

[11] Ding C, Hu H, Ding G, et al. Influences of tube pitches on heat transfer and pressure drop characteristics of two-phase propane flow boiling in shell side of LNG spiral wound heat exchanger[J]. Applied Thermal Engineering, 2017, 131.