
Heat exchanger mesh and numerical calculation

Lujie Sun ^a, Zhe Li ^b, Qiqi Fu ^c and Luyao Zou ^d

School of Shandong University of science and technology; Shandong 266590, China;

^a838457922@qq.com, ^b1214164853@qq.com, ^cfuqiqi1223@163.com,
^dzouluyaoxf@163.com

Abstract

Heat exchanger is an indispensable process equipment in modern industry. The performance of heat exchanger determines the quality of the whole project. Therefore, the study on heat exchanger performance is more and more important. According to the heat exchanger performance, the model of heat exchanger was meshed and calculated by computer software.

Keywords

Heat exchanger, Grid division, Numerical calculation.

1. Introduction

Heat exchanger is fluid general technology of heat exchange equipment, widely used in chemical, petroleum, power, light industry, metallurgy, nuclear power, shipbuilding, aviation, heating and other industries, especially in petroleum refining and chemical processing device has a machine in the important position. Heat exchanger is also because of its unique advantages, in order to save energy environmental protection received great attention from all walks of life, in recent years, various kinds of heat exchanger with new makes heat exchanger in the technical level of the quality^[2].

The numerical simulation method provides powerful measures for the design, evaluation and transformation of heat exchangers for economic safety^[2]. It has great advantages in designing, evaluating and analyzing new heat exchange equipment. In this paper, the geometric structure model of heat exchanger was established by using CFD software, and the numerical simulation analysis of heat exchanger was carried out by using FLUENT commercial CFD software.

2. Establishment of geometric model and grid division

The heat exchanger introduced in this paper is a common process equipment. Its graphic size is shown in figure 1, and the external wall of the heat exchanger is hidden for display. High temperature fluid ($T = 100\text{ }^{\circ}\text{C}$) imports from spiral pipe flow and the pressure of 20000 pa cryogenic fluid ($T = 0\text{ }^{\circ}\text{C}$) inflows from below, the pressure is 1000 pa.

Use CFD to create geometric models:

- (1) Point creation: create the required point by directly entering coordinates and coordinate increment. Each point is a marker that needs to be further processed later.
- (2) Create curves: the curve function in the collection menu closes the point connection.
- (3) Founder: define the stretching vector with the curve and point created, create the inner and outer wall surfaces of the device, and close the upper and lower bottom surfaces.
- (4) Create spiral pipe: create multiple points, defined $X=0.75 \times \cos(m \times 720^{\circ})$ in the X text box. Defined $X=0.75 \times \sin(m \times 720^{\circ})$ in the Y text box. $Z=0.5+m$ is defined in the Z text box to generate the spiral line, which is used to further generate the spiral pipe.

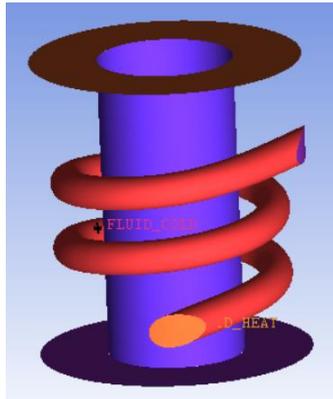


Figure. 1 geometric model of heat exchanger (no external wall is shown)

Define grid parameters:

- (1) Global parameters of the grid are defined, Scale factor=1 and Max element=0.2 are set, Tetra/ is selected for Mesh Type, and Robust is selected for Mesh Mixed Method.
- (2) Prismatic mesh was adopted, and the parameters were respectively defined as Min prism quality=0.1, Ortho weight=0.50, Fillet ratio=0.10, Max prismangle=180, Max height over base=1.0, according to the model.
- (3) Continue to define the grid parameters of each Part as shown in table 1, and set the maximum grid size of each Part. This completes the setup of the grid and further generates the grid as shown in figure 2.

Table 1 grid parameters of each part

| Edge | Max size | height | Height ratio |
|-----------|----------|--------|--------------|
| C_IN | 0.15 | 0 | 0 |
| C_OUT | 0.15 | 0 | 0 |
| H_IN | 0.05 | 0 | 0 |
| H_OUT | 0.05 | 0 | 0 |
| WALL_PIPE | 0.18 | 0.01 | 1.2 |
| WALL_IN | 0.05 | 0.01 | 1.2 |
| WALL_OUT | 0.15 | 0.01 | 1.2 |

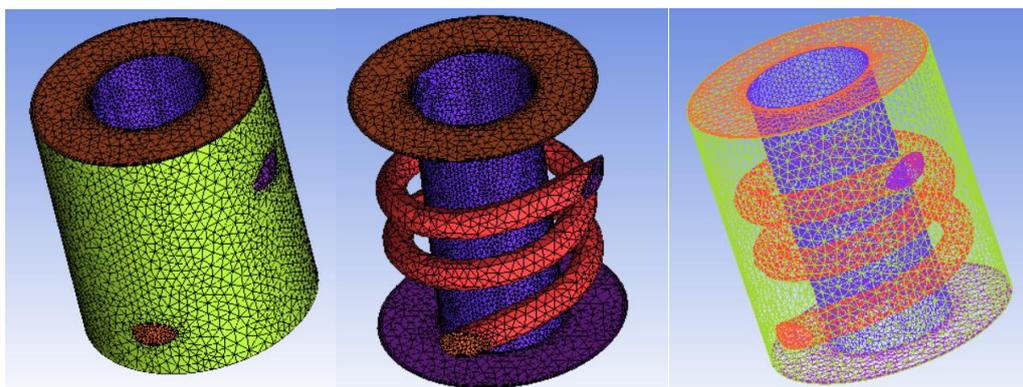


Figure. 2 Model grid

- (4) After grid generation, check the grid quality according to the unit type, as shown in figure 3.

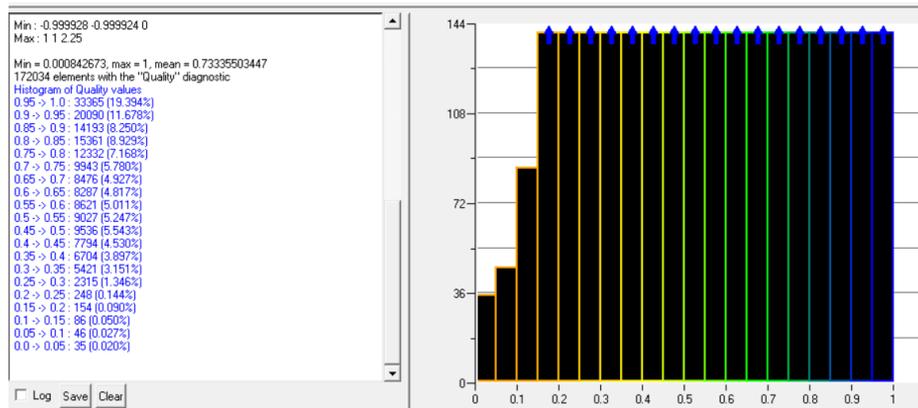


Figure. 3 grid quality

Therefore, the mesh quality of the model meets the requirements of numerical calculation and post-processing in the next step.

3. Numerical calculation

Open the completed model above in FLUENT software, as shown in figure 4.



Figure. 4 model grid

(1) Firstly, the grid length unit was defined as m in the Scaling column, and the grid quality report was generated by continuing to check the grid in FLUENT. The value of Minimum Volume was observed to be greater than 0, so the next step was carried out.

(2) The solution model is defined. The flow and heat transfer of the fluid are governed by the laws of physical conservation, so the shell flow of the heat exchanger must satisfy the following three basic control equations^[3]:

Mass conservation equation:

$$\nabla u = 0 \tag{1}$$

Momentum conservation equation:

$$\rho \left(\frac{\partial u}{\partial t} + u \nabla u \right) = \rho f - \nabla p + \mu \nabla^2 u \tag{2}$$

Energy conservation equation:

$$\rho c_p \left(\frac{\partial T}{\partial t} + u \nabla T \right) = \lambda \nabla^2 T + \rho q \tag{3}$$

A three-dimensional steady-state solver was selected, Laminar Viscous Model was selected in the Viscous Model panel, and the water-liquid parameter in FLUENT Database was used to create the material. Both fluid-cold and Fluid-hot materials are defined as water-liquid.

(3) Define the boundary conditions. According to the flow conditions of the problem itself, set the Total Pressure of Gauge =1000Pa, Supersonic/Initial Gauge Pressure=800Pa, and Total Temperature=273K to define the cold water inlet. Gauge Total Pressure=20000Pa, Supersonic/Initial Gauge =10000Pa, Total Temperature Pressure= 373K define hot water inlet, Gauge Pressure=0Pa, Backflow Total Temperature=300K define wading outlet and cold water outlet, and set wall material and Temperature according to the actual problem conditions.

(4) Define the solver control parameters, select SIMPLE Pressure in Velocity Coupling Scheme, and First Order Upwind in Momentum. The residual values of the monitor are set to 1×10^{-6} and the temperature change of the cold water outlet is monitored.

The maximum number of solving steps is defined as 2000, and the calculation results are shown in figure 5.

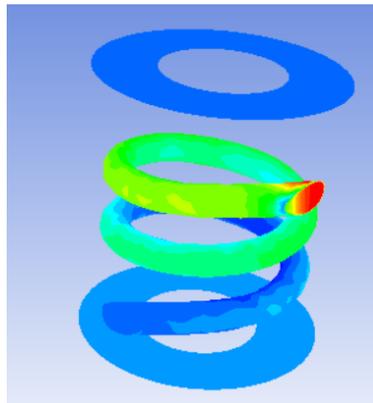


Figure. 5 contour of outlet temperature change

4. Conclusion

At present a lot of process equipment research and development plan and the research is based on the computer, on the basis of a model of grid generation accounted for most of the time of the entire project cycle, so the grid generation is more and more important, the quality of the grid also means that the follow-up on the accuracy of the numerical calculation.

Based on the heat exchanger and the process of numerical calculation, the build quality is reliable, on the basis of grid, for subsequent numerical calculation of the heat exchanger can play a big help, vary according to the outlet temperature of the heat exchanger, should be in hot water inlet and the cooling water inlet pressure to strengthen control, control of the liquid flow rate so as to be able to better energy exchange.

References

- [1] Lu minglu, Yang xin, zhang yao, solemn. Analysis and classification application of heat exchangers [J]. Contemporary chemistry, 1998, 3:582-584.
- [2] Tao Wenquan. Numerical heat transfer[M]. Xi'an: Xi'anJiao Tong University Press, 1990: 79-83(in Chinese).
- [3] Wang fujun. Computational fluid dynamics analysis [M]. Beijing: tsinghua university press, 2004: 7-10.