Computational Fluid Dynamics Model Based on Thermal Energy Application

Yang Hu

West China Hospital of Sichuan University, Chengdu, 610041, China

Keywords: Heat exchanger; Fluid dynamics; Heat transfer characteristics

Abstract: Heat exchanger is an indispensable equipment to transfer heat, and its efficiency directly affects the overall energy utilization rate and production economic benefits of the equipment. Because the plates of plate heat exchanger are different and the flow channels are different, there is great uncertainty in the design process of plate heat exchanger. The basic principle of the heater is to use different heat transfer media for heat exchange, which is a great embodiment of better utilization of resources. Therefore, the methods of improving heat exchange efficiency and improving energy utilization rate are the prerequisite problems in heat exchanger research. In actual production conditions, due to the limitations of the volume, weight and processing technology of the heat exchanger, the method of simply increasing the plate area to improve the heat transfer performance has limitations. Therefore, improving the flow channel design inside the plate heat exchanger to improve the heat transfer rate per unit volume is an important method to improve the heat transfer performance. In this paper, a computational fluid dynamics model based on thermal energy application is studied to analyze the heat transfer characteristics of plate air heaters and formulate optimization strategies.

1. Introduction

Heat exchanger is a widely used energy-saving equipment, which is used to realize heat transfer between materials in metallurgy, electric power, petroleum, chemical industry, light industry, food and other industries [1]. Its function is to make materials at different temperatures contact directly or indirectly to complete heat transfer. The advantages of plate heat exchanger, such as compact structure, light weight, excellent heat transfer characteristics and low manufacturing cost, have attracted more and more attention. The existing forms of non-renewable energy mainly include coal, natural gas and oil, and most of the reserves are quite limited [2-3]. Facing the increasingly severe development environment, how to make rational use of the remaining energy and improve the utilization rate of energy has become a key research issue for relevant people and scholars in the industry [4]. How to make good use of energy, the main research content is the efficient conversion of primary energy and the improvement of the advanced nature and efficiency of technology in each stage [5].

The main factors affecting the performance of plate heat exchanger are: plate shape parameters, plate collocation form, medium flow mode, etc., and the flow channel shape of plate heat exchanger is complex, and the installation and stacking methods are diverse, and the experimental research period is long and the cost is high [6]. The flow channel between plates of plate heat exchanger is composed of many small flow sections. Under the same working conditions, the heat transfer coefficient is much higher than that of conventional shell-and-tube heat exchanger [7]. The new type of plate air heater has various plate forms, and different structural dimensions will have different effects on the heat transfer effect. In the actual production conditions, due to the limitations of the volume, weight and processing technology of the heat exchanger, the method of simply increasing the plate area to improve the heat transfer performance has limitations [8]. Therefore, it is an important method to improve the heat transfer performance by improving the flow channel design inside the plate heat exchanger and increasing the heat transfer rate per unit volume.

In terms of heat exchanger design, developing compact heat exchanger is the direction of heat exchanger optimization. If a more efficient heat transfer effect can be obtained in a smaller heat

  DOI: 10.25236/icmmct.2023.038

exchanger, the production of raw materials will be saved and the effect of energy saving and emission reduction will be achieved [9]. During the investigation of existing data, it is found that there are few topics about the research and calculation of plate size in the design of internal flow channel of plate heat exchanger, and the summary of the law that plate flow channel structure affects fluid flow characteristics and heat transfer characteristics is not perfect, but considering these aspects is the key to improve the efficiency of heat exchanger. It is one of the effective ways to reduce the cost and shorten the cycle to study the flow and heat transfer characteristics between plates by numerical simulation. In this paper, a computational fluid dynamics model based on thermal energy application is studied to analyze the heat transfer characteristics of plate air heaters and formulate optimization strategies.

2. Computational fluid dynamics theory

Fluid dynamics mainly takes fluid as the research object and studies its mechanical motion law. In other words, it is to study the relationship between the external force and the application of fluid. Fluid dynamics takes the interaction between moving fluid and solid as the main research problem. The shape of flow channel in most types of plate heat exchangers is complex, and the cross-sectional shape in the direction of fluid flow constantly changes periodically. The two-dimensional or three-dimensional flow of fluid in the channel of plate heat exchanger and the small hydraulic diameter of the channel will greatly enhance the disturbance of fluid, so the plate heat exchanger can form a turbulent flow at a very low Reynolds number. From the microscopic point of view, because there are gaps between fluid molecules, the distribution of physical quantities representing fluid is discontinuous in space, and at the same time, under the influence of random motion of molecules, the physical quantities at a point in space are not continuous in time [10]. Fluid is composed of molecules that do thermal motion in one direction, and has compressibility. The compressibility of gas is large, while the compressibility of liquid is relatively small. When a series of morphological changes occur, there is a certain flow resistance inside, which is viscous.

In general, even in a small space, there are a lot of fluid molecules, and fluid mechanics is a science that studies the force and motion of fluid macroscopically. It studies the macroscopic characteristics of fluid, that is, the average statistical characteristics of macromolecules, so there are enough reasons to regard fluid as composed of continuous distribution of fluid particles. Like other disciplines, fluid mechanics is based on theoretical analysis and experimental research. Theoretical and experimental studies on fluids have been conducted for a long time. Among them, theoretical analysis is to obtain a definite solution by using mathematical methods under assumed conditions, but the problems that theoretical analysis can solve are often few and the calculation is very complicated. The physical model considered in fluid mechanics is a continuum model. In terms of characteristic size, compared with the size of the studied problem, the size of fluid particles in the continuous medium is small enough, that is, macro is small enough, but it contains a large number of fluid molecules, reflecting the average properties of a large number of molecules, that is, fluid micelles are large enough in micro.

Any closed surface in the flow field can be expressed by the integral of the infinitesimal mass $\rho d\tau$ in the control volume:

$$m = \vartheta_{\tau} \rho d\tau \tag{1}$$

Where τ is the volume, ρ is the fluid density and m is the fluid mass. The amount of change per unit time can be expressed as:

$$\frac{\partial m}{\partial_t} = \frac{\partial}{\partial_t} \vartheta_{\tau} \rho d\tau \tag{2}$$

Any fluid flow should satisfy the law of conservation of mass, and the mass flowing into the control body through the closed surface must be equal to the increase of the mass of the control body. If the inflow mass of fluid in the control body is greater than the outflow mass per unit time, that is:

$$\vartheta_{A} \rho v dA < 0 \tag{3}$$

The unit vector in the direction of dA outer normal is denoted as n, and v is the velocity vector. The fluid quality in the control body will become larger, that is:

$$\frac{\partial}{\partial t} \vartheta_{\tau} \rho d\tau > 0 \tag{4}$$

Compared with the experimental method, the numerical simulation method has the advantages of low cost and high speed, and it can simulate a large number of different models and working conditions without real experimental models. With the rapid development of computer software and hardware technology, computational fluid dynamics plays an increasingly important role. Considering the characteristics of the fluid continuum model, because the fluid points occupy every point in the flow space at any time point, and the variables representing fluid characteristics and motion performance in the model are usually continuous functions in time and space, this provides a basis for using the continuous function method to solve and analyze fluid problems [11]. The purpose of computational fluid dynamics is to approximate the continuous field in time and space with enough discrete points, establish algebraic equations of physical quantities at discrete points through the laws of mass conservation, momentum conservation and energy conservation, and obtain approximate solutions of the whole physical field by solving the algebraic equations. In this way, we can solve the particularly complicated flow problem and get the velocity, pressure, temperature and concentration of each scattered point in the whole calculation area. The fluid particles in a moving fluid at any time and any space point will change their kinematic parameters and thermodynamic parameters in a certain way when they exchange heat with the outside world under the action of various external forces, and the changes of each parameter are interrelated.

3. Heat transfer analysis of plate heat exchanger

When designing a heat exchanger, it is necessary to consider how to increase the heat exchange capacity. The usual method is to effectively improve the heat transfer coefficient and achieve the purpose of strengthening heat transfer. The flow in the deep groove of asymmetric plate heat exchanger presents a cross flow, that is, the fluid mainly flows along the deep groove, and continues to flow along the deep groove without folding into the opposite plate when flowing through the cross flow channel until the two sides of the plate edge, and then turns back into the deep groove of the opposite plate in a cross shape. However, the fluid flowing into the shallow groove is obviously dragged by the fluid in the deep groove when it flows through the cross channel, resulting in local vortex. The specific plate type of plate heat exchanger and its structural design parameters have obvious influence on its heat transfer and flow resistance. Different plate types make the fluid flow between plates have significant differences. The heat in the thermal boundary layer is conducted by heat conduction, and the local heat density is expressed as:

$$q_x = \Gamma\left(\frac{\partial t}{\partial y}\right) \tag{5}$$

Where Γ is the thermal conductivity of the fluid. The convection heat transfer phenomenon is expressed by Newton's cooling formula:

$$\frac{\partial}{\partial t} \vartheta_{\tau} \rho d\tau > 0 \tag{6}$$

Integrated:

$$\frac{\partial}{\partial t}\underset{\tau}{}\vartheta \rho d\tau > 0 \tag{7}$$

Where h_x is the heat transfer coefficient of the wall surface, which can be obtained by the temperature gradient on the wall surface.

Horizontal corrugated plate and straight corrugated plate are the two most common plate types at present. The flow in the plate heat exchanger with horizontal corrugated plate is a typical three-dimensional network flow, while the flow in the plate heat exchanger with straight corrugated plate is two-dimensional. The fluid flows in the shallow groove of the opposite plate, and the whole flow field is continuous and parallel with small ripples, that is, tortuous flow. The reason for this different flow state is that the deep groove channel is wide, and the hot side fluid mainly flows along the deep groove, and its momentum component along the mainstream direction is larger, which is less affected by the drag of the fluid in the opposite plate groove, and the turning-back points appear at the left and right edges of the plate, so the flow field presents a cross flow. Other things being equal, with the increase of plate spacing, the heat transfer performance of plate heat exchanger decreases. The research shows that when the flowing working medium is water, the fluid in the horizontal corrugated plate heat exchanger will enter the ruiliu state at the time of production. Therefore, evenly and reasonably arranging the nodes on the plate and appropriately reducing the spacing can not only improve the heat transfer efficiency, but also strengthen the overall strength of the plate and bear a large pressure drop.

4. Conclusions

Plate heat exchanger plays an important role in modern industrial society. As an important heat exchanger in chemical equipment industry, its performance has an important impact on energy saving and production efficiency improvement. With the wide application of plate heat exchanger in industrial field, the requirements for plate heat exchanger are becoming more and more diversified. In this paper, a computational fluid dynamics model based on thermal energy application is studied to analyze the heat transfer characteristics of plate air heaters and formulate optimization strategies. The flow channel between plates of plate heat exchanger is composed of many small flow sections. Under the same working conditions, the heat transfer coefficient of plate heat exchanger is much higher than that of conventional shell-and-tube heat exchanger. The flow channel of plate heat exchanger is relatively narrow, and the plate is composed of diversion area and corrugated area, and the cross-section distribution is complex, which makes the flow direction and velocity of medium change constantly and increases the disturbance of medium flow. The cold and hot channels of asymmetric plate heat exchanger have different structures and different thermal characteristics. Scientific analysis and reasonable design for specific asymmetric working conditions can make full use of the allowable pressure drop of media on both sides, improve the overall heat exchange efficiency of the heat exchanger, reduce the heat exchange area and give full play to the energy-saving potential of the heat exchanger.

References

[1] Bugaj M A, Mik K.Effect of the silicone-based heat exchanger construction material modified with PCM on thermal working conditions of flexible photovolaic modules[J].Przemysl Chemiczny, 2019, 1(8):79-83.

[2] Xiao D, Liu J, Yang R,et al.Research on economic length of the horizontal section of downhole coaxial heat exchanger geothermal system based on net present value[J].International Journal of Energy Research, 2022, 46(5):6351-6369.

[3] Zhang W, Xue X, Liu F,et al.Modelling and experimental validation of advanced adiabatic

compressed air energy storage with off-design heat exchanger[J].IET Renewable Power Generation, 2020, 14(3):389-398.

[4] Navickait K, Penzel M, Bahl C R H,et al.Performance Assessment of Double Corrugated Tubes in a Tube-In-Shell Heat Exchanger[J].Energies, 2021, 14(5):1343.

[5] Triscari G, Santovito M, Bressan M,et al.Experimental and model validation of a phase change material heat exchanger integrated into a real building[J].International Journal of Energy Research, 2021, 45(12):18222-18236.

[6] Yang L H, Liang J D, Hsu C Y,et al.Enhanced efficiency of photovoltaic panels by integrating a spray cooling system with shallow geothermal energy heat exchanger[J].Renewable Energy, 2019, 134(4):970-981.

[7] Venkateswaran S, Wilhite B, Kravaris C.Identification of Optimal Catalyst Distributions in Heat-Exchanger Reactors[J]. Industrial and Engineering Chemistry Research, 2020, 59(13):5699-5711.

[8] Yaqiong H, Guihua T.Thermal-Hydraulic-Structural Analysis and Design Optimization for Micron-Sized Printed Circuit Heat Exchanger[J].Journal of Thermal Science, 2019, 28(02):96-105.

[9] Majumdar S, Moisseytsev A, Natesan K.Assessment of next generation nuclear plant intermediate heat exchanger design.[J].Journal of Nuclear Materials, 2018, 392(2):307-315.

[10] Amanuel T, Mishra M.Experimental study and exergy analysis of three-fluid tubular heat exchanger with nanofluids[J].International Journal of Exergy, 2021, 35(2):222.

[11] Araiz M, Casi L,Leyre Catalán,et al.Thermoelectric Generator with Passive Biphasic Thermosyphon Heat Exchanger for Waste Heat Recovery: Design and Experimentation[J].Energies, 2021, 14(18):5815.