ISSN 2616-5767 Vol.7, Issue 3: 173-177, DOI: 10.25236/AJETS.2024.070324

Simulation on Characteristics of Multiple-Tube Side Heat Exchanger in Airflow Distribution Uniformity

Xiong Li

Automotive and Aviation Institute, Hubei Communications Technical College, Wuhan City, Hubei Province, 430202, China

Abstract: *Aiming at the flow performance of gas-water multiple-tube side heat exchanger, a numerical simulation based on FLUENT software was carried out for airflow distribution uniformity of the split range channels, with the influence analysis on the ununiformity and energy loss under different conditions of the heat exchanger tube arrangement number, vertical section guide angle. The simulation results show that heat exchanger tube arrangement number have important impact on the ununiformity and energy loss, the left guide angle have significant impact on ununiformity, but the right guide angle have little influence when single side angle change, and with the increase of them, the energy loss increase slightly. When both side angle change, ununiformity and energy loss become fluctuate, and overall evenly distributed effect is lower than single side angle. Within the scope of this study, the structure of split range channels is recommended that left angle is 78.7°, and right angle is 90°. The numerical simulation can provide the basis for similar compact multiple tube side heat exchanger diversion technology development.*

Keywords: *tubular heat exchanger, split range channels, distribution uniformity, numerical simulation*

1. Introduction

Tubular heat exchanger has the advantages of firm structure, high operating flexibility, strong adaptability, heat transfer area per unit volume is large, and the heat transfer effect is well [1]. In order to increase the efficient heat exchange area, the heat exchanger tube number was added, but the flow rate of the medium in the tube bundles become down, and lead to the surface heat transfer coefficient decreased, so as to increase the number of heat exchanger tube may not reach the requirements, then, to keep the larger velocity of fluid in the tube bundles, can divide tube bundles into a number of processes [2]. The curved path exists in split range channels with multiple tube side heat exchangers. When airflow run through tube side, because of the low density, streamline direction will change after through split range channels, it can lead to mainstream away from the tube wall and make the flow field maldistribution [3]. The uneven distribution of velocity field leads to temperature field maldistribution in the tube side, and the heat transfer performance of bundle will be affected. It seemed that the structure of split range channels needs to optimization.

The predecessor has some research on fluid flow characteristics in the curved path. Ding Jue has research on turbulent flow in square-sectioned 90-degree bend with numerical simulation, and fluid separation was discovered [4]. Mao Jianfeng has do the simulation and optimization on the entrance channel of heat pipe exchanger, and discovered that suitable place of the baffle can enhance the distribute of velocity field and pressure field, as well as heat transfer performance [5]. This paper simulation on the structure of split range channels, and search for optimum size, the numerical simulation can provide the basis for similar compact multiple tube side heat exchanger diversion technology development.

2. The geometry model &basis theoretical calculation

2.1. The geometry model of heat exchanger

The structure of the multiple-tube side heat exchanger as shown in Figure 1, heat transfer component was bundle, and heat transferred from tube side hot air flow to outside cooling water. The length of the bundle is 100mm, copper material, wall thickness is 0.5mm, and arrangement for Parallel Square, heat exchange tube pitch generally not less than 1.25 times diameter [2]. Because of the split range channels has consistent structure in depth direction, it can be simplified two-dimensional model as shown in Figure 2. The analysis level of α_1 (α_2) are 68.2°, 73.3°, 78.7°, 84.3°, 90°, the heat exchanger tube arrangement number are 4×4, 5×5, 6×6.

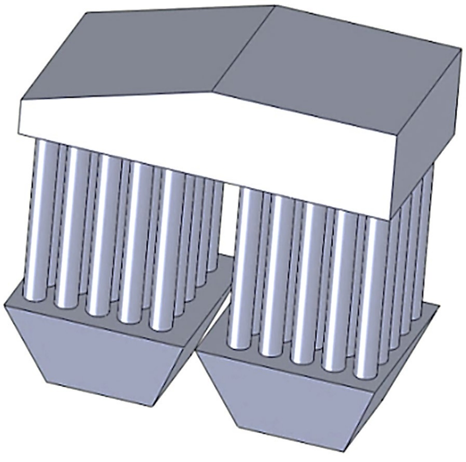

Figure 1. 3D structure of heat exchanger.

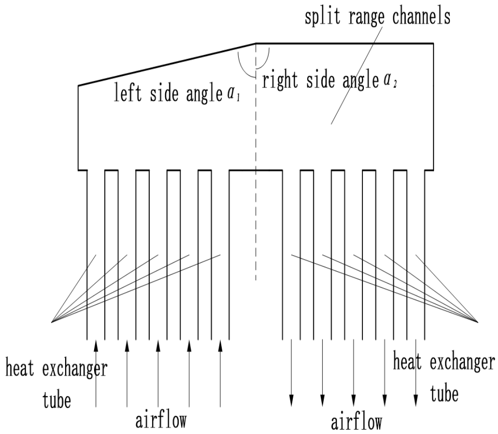

Figure 2. 2D simplify structure of heat exchanger.

2.2. Examination index

The uneven flow in tube 2 will impact on tube 1, and its uniformity cannot be unified. Its need to make the appraisal to the tube 1 and tube 2 distribution uniformity of flow in the meantime, the evaluation standard can take for ununiformity S values [6], its formula as shown in Eq. (1), and the lower S values, flow distribution more uniformity. S_i is the ununiformity of tube 1 and tube 2, u_i is the average velocity of tube 1 and tube 2, u_{ij} is the average velocity of each tube cross section (the value range of i is 1 and 2, the value range of j is 1-36). When increase the uniformity of the flow at the same time, also can increase the flow resistance of the heat exchanger, yet can also take the E value as the evaluate standard for energy loss in the tube side [7], its formula as shown in Eq. (2), p_{1j} and p_{2j} is the total pressure of each tube, L is outlet gridding length, ρ is the average density of airflow.

$$S_i = \left[\frac{1}{n-1}\sum_{j=1}^{n}\left(\frac{u_{ij}}{u_i}-1\right)^2\right]^{\frac{1}{2}} \tag{1}$$

$$E = \sum\left[\rho u_{1j} L\left(\frac{p_{1j}}{\rho}+\frac{u_{1j}^2}{2}\right)\right] - \sum\left[\rho u_{2j} L\left(\frac{p_{2j}}{\rho}+\frac{u_{2j}^2}{2}\right)\right] \tag{2}$$

3. Numerical simulation and boundary conditions

The entire heat exchanger's tube side was selected to simulate calculation. Modeling and mesh generation by using GAMBIT, numerical calculation is done by FLUENT software. In order to improve the calculation accuracy of flow boundary and thermal boundary near the wall, it can adopt hexahedral grid refinement mesh. The total grid number is 687760. Data collected section selected as the center of the tube 1 and tube 2, Figure 3 is the top view of cross section of the heat exchanger (it's an example of 5×5 bundle arrangement), bundle numbering sequence of each tube is from left to right, from top to bottom.

The inlet flow pattern is turbulence, so the calculation model is selected to standard k-ε model. The inlet boundary conditions are selected to velocity-inlet [8], because this kind of heat exchanger used in automotive, entrance flow is set to 20m/s [9], and temperature is 473K. The outlet boundary condition is selected the outflow option, the outer wall temperature condition is set to constant wall temperature of 373K. The coupling of pressure and velocity using SIMPLE algorithm on staggered grids, momentum, energy and turbulent diffusion rate is solved by a first-order upwind scheme, define the convergence condition for mass and energy calculated residual absolute value respectively 10^{-4} and 10^{-6} precision.

ISSN 2616-5767 Vol.7, Issue 3: 173-177, DOI: 10.25236/AJETS.2024.070324

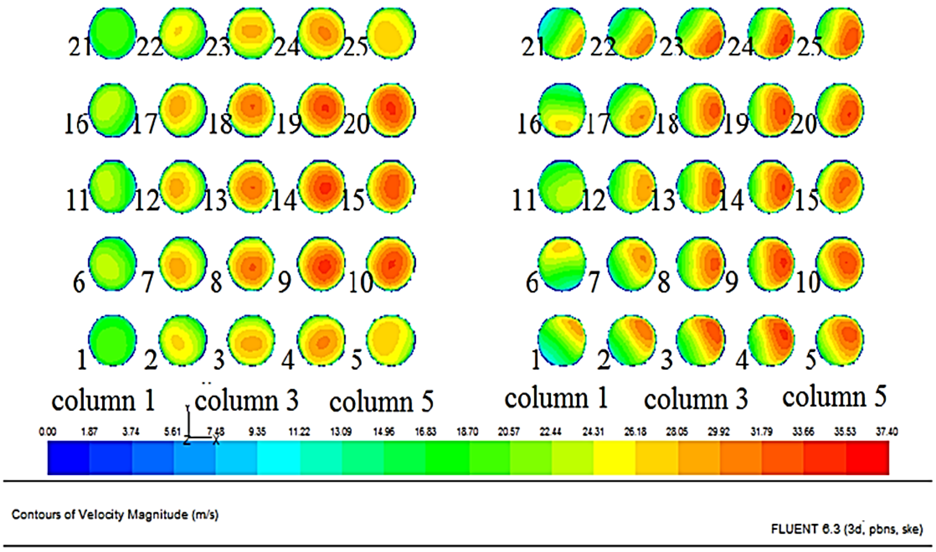

Figure 3. Velocity gradient map of bundle.

4. Numerical simulation results and analysis

4.1. The flow distribution in different split range channel structure

Figure 3 shows the velocity gradient map of bundle which $\alpha_1=\alpha_2=68.2°$. Figure 4 shows the average flow distribution of different split range channels in each column which abscissa is the column number (see Figure 3). It can be seen from the Figure 3 that flow distribution very different between each column, and the flow increased from left to right either of the tube 1 and tube 2. The reason is that: the dynamic pressure increase and static pressure decrease from left to right in curved path of the tube 1, and the tube 2 is the opposite, which make pressure difference increasing from left to right in the split range channels. As can be seen from Figure 4, the difference of velocity distribution was not obvious between different structures, in order to get the quantitative difference, needs the *S* value as the criteria [10].

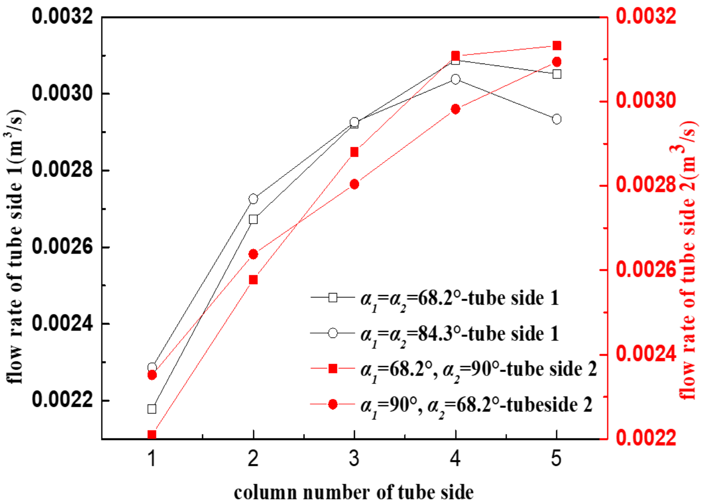

Figure 4. The average flow distribution of each column.

4.2. Influence of single side angle on maldistribution and energy loss

Figure 5 and Figure 6 shows the maldistribution of the different column number when single side angle changes (when one side angle changes, other side angle keep 90 degree). From the Figure 5, with the increase of the α_1, the ununiformity S_1 and S_2 decreased, and the reduction of relative between

0.3%-17.8%, as well as the decreasing amplitude decreases gradually. Because with the increase of $\boldsymbol{\alpha_1}$, flow section of the split range channels added, and lead to velocity decrease, the pressure changes corresponding decreases. The more bundles number, the higher $\boldsymbol{S_1}$ and $\boldsymbol{S_2}$ by comparison of ordinate values. Because the velocity of airflow become low when it through the tube side, and the flow resistance loss is reduced [11].

From the Figure 6, with the increase of $\boldsymbol{\alpha_2}$, S1 basically unchanged, limit the relative value is 1.5%; $\boldsymbol{S_2}$ rise slightly, the highest relative value is 8.4%. Because cross-sectional area of split range channels increases gradually from left to right, flow resistance was increased, as well as the static pressure.

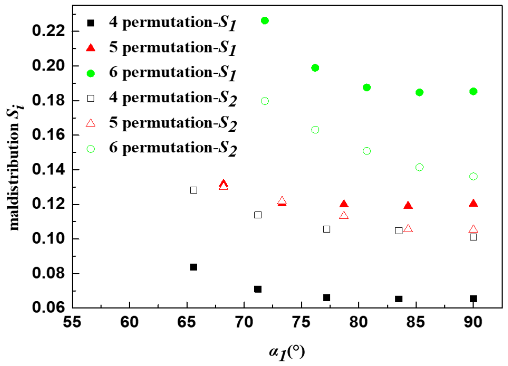

Figure 5. Influence of α_1 on S_i.

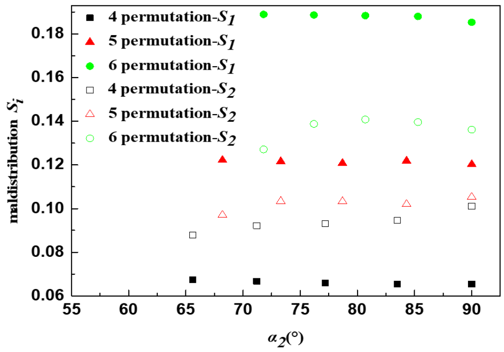

Figure 6. Influence of α_2 on S_i.

Figure 7 shows the energy loss of the different column number when single side angle changes. From the Figure 7, with the increase of the $\boldsymbol{\alpha_1}$, energy loss E went up after dropping slightly, there have a minimum value between 77.2 and 80.7 degrees, the highest relative value is 5.2%-9%. Because the vortex intensity of airflow gets stronger in the split range channels, which leading to increase of energy loss. With increase of $\boldsymbol{\alpha_2}$, $\boldsymbol{E}$ showed a trend of reducing, reduction of relative value is between 0.3%-8.19%. With increase of the bundle number, the energy loss increase, which because the velocity of airflow decreased when it through the tube side, and the flow resistance loss is increased.

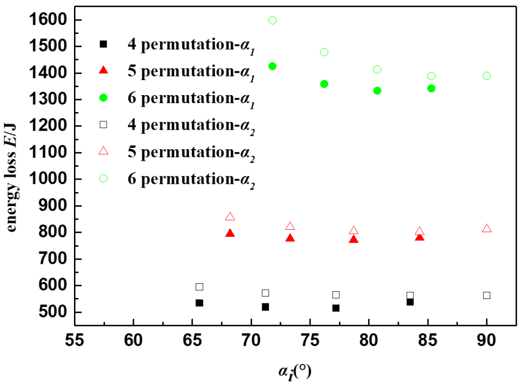

Figure 7. Influence of α_1 &α_2 on energy loss.

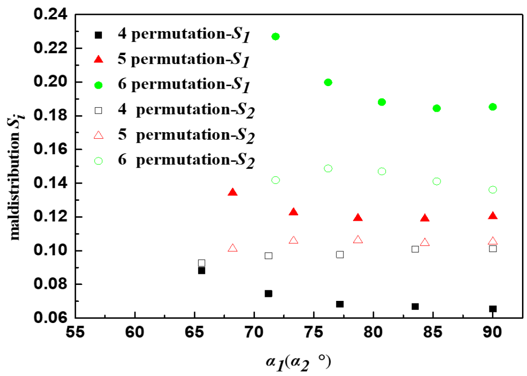

Figure 8. Influence of both angles on S_i.

4.3. Influence of both side angle on maldistribution and energy loss

Figure 8 shows the maldistribution of the different column number when both side angle changes ($\boldsymbol{\alpha_1}$=$\boldsymbol{\alpha_2}$). From the figure, with increase of the angle, $\boldsymbol{S_1}$ decreased, and the reduction of relative between 0.2%-18.3%, which because cross-sectional area was increased, the range of velocity was decreased, static pressure reduced corresponding. $\boldsymbol{S_2}$ rise first and then fall, and there have a minimum value between 76.2 and 83.5 degrees.

Figure 9 shows the energy loss of the different column number when both side angle changes. With the increase of the angle, energy loss $\boldsymbol{E}$ went up after dropping, there have a minimum value between 77.2 and 85.3 degrees, the highest relative value is 7.1%-19.0%. Because with increase of angle, guiding angle of split range channels will decrease, and the vortex area become larger in the leftmost and rightmost when airflow pass through.

ISSN 2616-5767 Vol.7, Issue 3: 173-177, DOI: 10.25236/AJETS.2024.070324

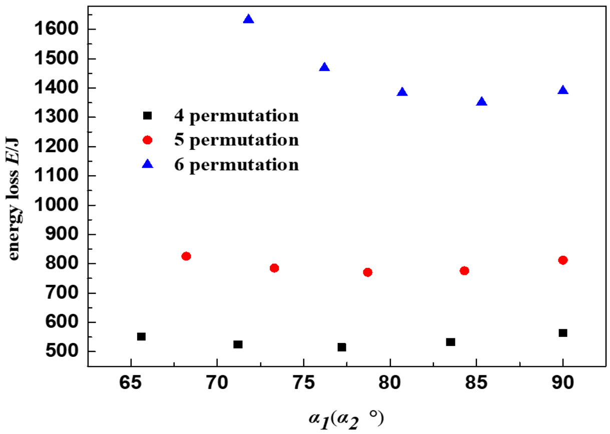

Figure 9. The average flow distribution of each column.

5. Conclusions

With the influence analysis on the ununiformity and energy loss under different conditions of the heat exchanger tube arrangement number, vertical section guide angle in this paper, the following conclusions are made:

1) When single side angle changes, with increase of the α_1, the ununiformity decreased; with the increase of α_2, ununiformity basically unchanged; E was changed slightly. The more bundles number, the higher ununiformity and lower energy loss.

2) When both side angle changes, with increase of the angle, S_1 decreased, S_2 rise first and then fall, and overall evenly distributed effect is lower than single.

3) Within the scope of this study, the structure of split range channels is recommended that α_1 is 78.7°, and α_2 is 90°, relative value of ununiformity will decrease 17.8%, as well as energy loss will decrease 8.19% on this condition.

References

[1] Y H Wang. (2019) Research and Development a Strengthened Heat Transfer of Tubular Heat Exchanger. Refrigeration., 40: 53-57.

[2] S J Qin, W B Ye. (2013) Heat Exchanger. Chemical Industrial, Inc., Shandong province. pp. 80-83.

[3] G C Dai, M H Chen. (2015) Fluid Mechanics in Chemical Engineering. Chemical Industrial, Inc., Shandong province.pp. 154-155.

[4] J F Mao. (2020) Numerical simulation and optimization of heat pipe exchanger under variously velocity distribution inflow [D]. Changsha: Central South University.

[5] M A Habib, (2011) M R Ben, A M Said, et al. Evaluation of Flow Maldistribution in Air-Cooled Heat Exchangers. Applied Thermal Engineering., 22: 1-7.

[6] Z M Zhou, W G Fan, L C Dong. (2021) Design and Numerical Simulation for Flow Deflector in the Gas Flue of Selective Catalytic Reduction Denitrification System. Journal of Chongqing University., 32: 1187-1196.

[7] X Li, W J Liu, P Tang, et al. (2014) Numerical Simulation and Analysis on the Dispersed Flow Characteristics in Tee Branch Pipe of Exhaust System. Journal of Hebei University of science and technology.,35: 272-278.

[8] C L Xu, W J Liu, W P Hu. (2014) Numerical Simulation and Analysis of Enhanced Heat Transfer Tube for Heat Recovery of Vehicle Exhaust. Journal of Hebei University of science and technology., 34: 589-596.

[9] X M Fan, Y Xu, Q L Yang et al. (2020) Simulation and analysis of engine exhaust pressure wave. Computer Simulation., 27: 282-285.

[10] T Q Luo, Y X Cheng, Y Y Xie. (2017) Fluid mechanics[M].Mechanical industry press, Inc., Beijing, pp. 210-214.

[11] LIU Mengqin, LI Yanming, LIU Chengliang. (2010) Numerical simulation of flows in automotive catalytic converters with a novel defector and its structure optimum design. Machine Design & Research., 26: 59-63.