Study on the Influence of Tube Curvature on Heat Transfer Characteristics of High Efficiency Air Pre-cooler

Zhiqiang Cao*, Chunan Ai, Xueren Wang and Jian Li
Rocket Force University of Engineering, Xian, China

*Corresponding author e-mail: czq_1015@163.com, aca201601@163.com, alittlepanda@163.com, lijian_206@qq.com

Abstract. High-efficiency pre-cooler is one of the key components of RBCC combined engine. To explore the influence of tube row curvature on heat transfer, a convection heat transfer simulation of pre-cooler composed of different curvature tube rows is established. The model and numerical calculations show that when the curvature is small, the heat transfer efficiency increase slowly with curvature increasing, but the flow resistance increase obviously by increasing curvature. However, when the curvature is large, the effect of curvature increasing on the heat transfer efficiency is obvious and the increase of flow resistance get slower. Based on this, a ring pre-cooler is designed and the heat transfer effect is simulated. It has reference application value for the design of high efficiency pre-cooler.

1. Introduction
As one of the key components of combined engine, the high-efficiency air pre-cooler has many advantages of light and high efficiency, compact structure and high heat exchange efficiency. The precooling device assembled after the inlet of the combined engine can quickly cool the incoming air, increase the intake density of the compressor, and then increase the intake air flow and increase the thrust, effectively increase the thrust ratio of the engine and expand the flight envelope[1], which has great development advantages and technical potential. The heat transfer and resistance characteristics of the heat exchanger are measured by forced convection. Two important indexes [2], which have been studied deeply and carefully by many scholars at home and abroad, such as Myeong-Gie Kang's experimental study on the effect of tube spacing on boiling heat transfer of numerical tube bundles [3], Bacellar et al., through a large number of experimental data, gave the air side friction heat transfer characteristic equation of bare tube air-cooled heat exchanger with diameter of 0.5 ~ 2mm [4]. Kong et al studied the gas side flow and heat transfer performance of plate finned tube bundles and slotted finned tube bundles [5]. The effects of geometric parameters such as tube spacing, pipe diameter and tube type on heat transfer efficiency and flow resistance were investigated by experimental methods. Influence. In the aspect of numerical simulation, Dong Qiwu et al used CFD method to simulate the Rukauskas experiment [6], and the results match well with the experiment. Ahn et al studied the variation of unstable heat transfer on cylindrical surface with turbulence intensity and Reynolds number in detail by using FVM numerical simulation method [7]. Wang et al studied the thermodynamic properties of shell and tube heat exchanger on air side and shell side [8], and analyzed the relationship between fluid flow and heat transfer.
At present, the research on tube row heat exchanger mainly focuses on tube shape, pitch, wing shape and incident angle, the purpose of which is to explore the influence mechanism of tube shape, pitch, wing shape and incident angle on turbulent enhanced heat transfer, and the research object is usually uniformly arranged straight row tube bundles, but the research on tube heat transfer with curvature is less. However, the high efficiency pre-cooler in practical application often has higher requirements for the axial shape of the micro-tube, such as figure 1 is the pre-cooler model used in SABRE engine [9], in which the heat exchanger pipe is a curved pipe with cross arrangement, which can not only make the structure of the heat exchanger more compact, but also increase the heat transfer area. But its internal turbulent side In order to study the effect of the tube bundle curvature on the flow and heat transfer in the pre-cooler in the engineering application, the numerical simulation of the convective heat transfer in the pre-cooler of the tube bundle with curved arrangement is carried out in order to study the effect of the tube bundle curvature on the heat transfer in the pre-cooler. In this paper, the flow heat transfer characteristics of gas cross-swept different curvature pipe rows are discussed.

Figure 1. SABRE engine pre-cooler

2. Basic hypothesis and Mathematical Model

Li [10] et al., when using RANS method to calculate the convective heat transfer wall effect of swept tube bundles, pointed out the standard \( k-\varepsilon \) model can accurately simulate the convective heat transfer of fluid traversing tube bundles, so the standard \( k-\varepsilon \) is adopted in this paper. The model is calculated by using enhanced wall function [11], the discrete equation is coupled by finite volume method, the pressure velocity is coupled by SIMPLE algorithm, the pressure interpolation is carried out by standard scheme, and the others are treated by second order upwind scheme. In the course of the calculation, the following assumptions are made:

The main results are as follows:

(1) the fluid is incompressible, neglecting the influence of temperature on physical properties, assuming that the physical parameters are constant;
(2) the wall is assumed to be non-slip condition;
(3) the process of flow and heat exchange is steady state;
(4) the temperature of air and pipe wall is stable and the inlet speed is constant.

Based on the above assumptions, the governing equations of convective heat transfer in Cartesian coordinate system are as follows:

Continuous equation:

\[
\frac{\partial (u_i)}{\partial x_j} = 0
\]  

Momentum equation:

...
\[
\frac{\partial \left( \rho u_i u_j \right)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \frac{\partial u_i}{\partial x_j} + u_i \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j} \right) \right]
\]  
(2)

\[ \frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho ku_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} + \mu_t \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j} \right) \right] - \rho \varepsilon
\]  
(3)

\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{c_p}{k} \mu_t \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j} \right) - c_2 \rho \frac{\varepsilon^2}{k}
\]  
(4)

Energy equation:
\[
\frac{\partial \left( \rho c_p T u_j \right)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \lambda + \frac{c_p \mu_t}{\sigma_t} \right) \frac{\partial T}{\partial x_j} \right]
\]  
(5)

In the form: \( \mu_t = \frac{c_p \rho k^2}{\varepsilon} \);

\( u \) —— The velocity component, m/s;

\( i, j \) —— Tensor index;

\( T \) —— temperature, K;

\( \rho \) —— density, kg/m\(^3\);

\( \mu \) —— Dynamic viscosity, Pa·s;

\( c_p \) —— Constant pressure specific heat, J/(kg·K);

\( \mu_t \) —— Turbulent viscosity;

\( \sigma_t \) —— Energy Trump number, take 1.0;

\( \sigma_\varepsilon \) —— Pulsatile energy dissipation rate Plantt number, take 1.3;

Coefficient \( c_1, c_2, \mu_t \), Take 1.44, 1.92 and 0.09.

Reynolds number \( Re = \frac{u_{max} D}{\nu} \), Disfeature length, \( \nu \) is Dynamic viscosity, \( u_{max} \) Take the speed at the minimum section of the first row of tubes.

Average Nu number \( Nu = \frac{hD}{\lambda} = \frac{qD}{\lambda(T - T_w)} \), \( q \) is the heat transfer rate, \( D \) is the characteristic length, and the characteristic length is the outer diameter of the pipe when sweeping the circular tube, \( \lambda \) is the thermal conductivity of the fluid, \( T \) is the characteristic temperature, \( T_w \) is the wall temperature.

Definition of differential pressure resistance coefficient: \( f = \frac{\Delta P}{\rho u^2} \), \( \Delta P \) pressure differential, \( u \) is entrance velocity.
3. Establishment and Verification of Air Crossing Fork Tube Bundle Model

3.1. Physical model and boundary conditions

Fig. 2 is a schematic diagram of the arrangement of air traversing fork tube bundles, which is arranged in a triangular angle and a pitch of 2.5D (D is a circular tube diameter 2mm). Grimson's experimental results [12] point out that when the number of tube rows is more than 10 rows, the rear row tube will no longer be disturbed by the wake of the front row tube, so the 10 row tube bundles will be selected as the research object. The boundary condition of the velocity inlet is taken, the velocity direction is perpendicular to the intake surface, and the inlet temperature is 300K, the wall temperature is 100K, the outlet is the pressure outlet, and the symmetrical boundary condition is used in the axial direction.

![Figure 2. Bundle arrangement diagram](image)

3.2. Grid Generation and Independence Test

The structured grid is suitable for fluid convection heat transfer calculation because of its fast generation speed and good mesh quality. In order to accurately calculate the fluid change of the boundary layer of the tube bundle wall, the mesh near the wall surface shall be encrypted, and the 5-layer dense grid shall be divided outside the pipe, and Fig. 3 is the model grid division and partial enlarged drawing.

![Figure 3. Diagram of mesh generation](image)

In order to verify the rationality of grid division, five sets of grids with the same topology but different grid parameter distribution are selected to test the 80m/s stream, and the average Nussel number is calculated. The calculated results are shown in figure 4. It can be seen that when the number of grid reaches 1.4 million, the influence of increasing the number of grid on Nu number is very small. It can be considered that the grid has obtained an independent solution, considering the accuracy and economy, 1.4 million grid is used in the subsequent calculation.
3.3. Verification of accuracy of Simulation calculation Model
In the critical bypass interval, Zukauskas’s relationship with respect to the mean surface heat transfer coefficient of the fluid-swept fork bundle [13], \((s_1 < s_2)\):

\[
\text{Nu}_f = 0.35 \text{Re}^{0.6} \text{Pr}_f^{0.36} \left( \frac{\text{Pr}_f}{\text{Pr}_w} \right)^{0.25} \left( \frac{s_1}{s_2} \right)^{0.2}
\] (6)

The qualitative temperature is the average temperature of import and export, \(\text{Pr}_w\) is tube bundle wall temperature; The characteristic length is the diameter of the tube and the Re number is the average velocity at the minimum section. The numerical simulation results are shown in Fig. 5, and the Nu increases with the increase of Re in the tube heat transfer with the increase of Re, and the trend obtained by the two methods is consistent with that obtained by the two methods, and the numerical simulation results are as follows: figure 5 shows that the numerical simulation results increase with the increase of Zukauskas correlation.

![Figure 5. Relationship between Nu and Re number](image-url)
It can be seen from Table 1 that the maximum error at the Reynolds number is 4.76%, the result of the numerical simulation is smaller than the correlation calculation result, and the calculation result of the model and method is credible within the allowable range of the error.

| Re     | Numerical simulation Nu | Correlation calculation Nu | Deviation δ |
|--------|-------------------------|---------------------------|-------------|
| 8000   | 65.42                   | 65.99                     | 0.86%       |
| 12000  | 80.16                   | 84.17                     | 4.76%       |
| 16000  | 101.61                  | 100.02                    | 1.59%       |
| 20000  | 116.34                  | 114.35                    | 1.74%       |
| 25000  | 134.73                  | 130.73                    | 3.06%       |

4. Establishment of Model of curved Pipe Row pre-cooler and numerical Analysis of Convective Heat transfer

4.1. Convection and heat transfer characteristics of curved tube

The tube row with curvature of 12.6 is selected for simulation under Re=20000 condition, and the velocity cloud diagram is shown in Fig. 6. It can be seen that the velocity around the tube changes dramatically, and the velocity decreases to the lowest point after the tube, and then increases gradually. The average velocity between the rows increases gradually with the flow, which is due to the gradual reduction of the flow cross section in the curved flow field, and the larger the velocity gradient is.

![Figure 6. Velocity distribution map](image-url)

It can be seen from Fig. 7 that forced convection heat exchange occurs when air passes through the cooling pipe, and the tail mark is formed behind the pipe, the temperature is lower, and the wake of the rear tube bundle is longer than that of the front row pipe. As the flow proceeds, the wake air is mixed
with the hot air, and the temperature rises. The average Nu number of tube wall heat exchange is 134.48, which is 15.59% higher than that of straight tube row heat exchanger.

In order to investigate the change of heat transfer capacity of each row in the flow direction, each tube row is numbered along the flow. The heat transfer coefficient of each tube is shown in Fig. 8. The heat transfer coefficient of the rear tube is larger than that of the front tube, and the heat exchange coefficient increases greatly after the sixth row of pipe becomes stable. It is pointed out in [12] that, due to the rectification of tube banks in the straight tube bundle, when the number of tubes is more than 10, the heat transfer capacity of the rear tube bundle is significantly higher than that of the front row when the number of tube banks exceeds 10, the heat exchange energy can be greatly enhanced by increasing the number of tube banks. So in the design of tube row heat exchanger with bending arrangement, more number of tube banks can be considered. To enhance the heat transfer capacity of the heat exchanger.

![Graph showing heat transfer coefficient change along the direction of flow](image)

Figure 8. Heat transfer coefficient change along the direction of flow

It can be seen from the above analysis that the presence of the bending will greatly change the velocity distribution of the flow field, but this change will also increase the flow resistance of the heat exchanger while enhancing the heat exchange capacity, the flow resistance coefficient of the fluid being 3.66 under this operating condition, The resistance coefficient in the straight tube heat exchanger is only 2.25, which increases 62.6%, which will have a great negative effect on the flow, and the requirement of the heat exchanger to the intake air flow can also limit the design size of the tube row curvature, so that the optimal curvature needs to be taken into consideration.

### 4.2. Effect of tube discharge curvature on heat transfer and flow performance

In order to explore the influence of curvature on heat transfer and flow, pipe rows with curvature of 0, 4.2, 8.4 and 12.6 are selected to model. The flow field model is shown in Fig. 9, and the heat transfer area of each model is the same. The heat transfer characteristics under different Re numbers are calculated by numerical simulation.
The relationship between Nu number and Re number under different curvature is shown in Fig. 10. Under each curvature, the Nussel number increases with the increase of Reynolds number, and the growth trend is close to linear. The Nu increases with the increase of curvature, but the curvature has little effect on the Nu number when the curvature is small, such as the curvature of 0 and 4.2 Nu is almost equal, but when the curvature is 8.4 and 12.6, the Nunnber increases by 10.96% and 15.82%, respectively.

Fig. 11 shows the relationship between the number of resistance coefficients and the number of Re under different curvature. It can be seen that the flow resistance decreases with the increase of Reynolds number and increases with the increase of curvature. When the curvature is small, the resistance increases obviously, such as 4.2 and 8.4, which is 30.8% and 60.3% higher than that of the straight tube, respectively, but decreases with the further increase of curvature, such as 3.75 and 3.81 when curvature is 8.4 and 12.6, which increases only 1.6%.

The existence of curvature can also increase the heat transfer surface area per unit volume, make the structure more compact, and then increase the heat transfer power per unit volume of heat exchanger, which is one of the reasons why the pre-cooler used in aero-engine is designed as a ring. Fig. 12 shows the variation of heat transfer power per unit volume of tube row with curvature under different Reynolds numbers. it can be seen that the heat transfer power increases with the increase of tube discharge curvature at each Reynolds number, and the increase amplitude decreases with the increase of curvature at first.
4.3. Design and application of bending tube heat exchanger

Based on the above analysis, an efficient compact pre-cooler is designed. The heat exchanger adopts an arc tube row with curvature of 8.4, the number of pipe rows is 10, the cross section diagram is as follows: outer diameter 260mm, inner diameter 210mm, heat exchanger diameter 1mm, transposition equilateral triangular fork arrangement, pitch 2.5mm. According to the periodic symmetry, the 12 ° partial model unit of the pre-cooler is selected as shown in Fig. 14 for simulation calculation.

As shown in Fig. 15, the pre-cooler can cool the air flow at 1350K to 532K in 0.78 milliseconds, and the heat transfer power per unit volume can be further calculated to be 409.2 MW. In the future design, the heat transfer efficiency can be further improved by increasing the number of pipe rows and reducing the diameter of the pipe to meet the pre-cooling requirements of the engine. The model has certain reference value for the design and research of the future pre-cooler.
5. Conclusion

As one of the key components of the combined engine, the flow heat transfer characteristics of the precooler will significantly affect the overall performance of the engine. It is of great significance to study the internal heat transfer mechanism of the combined engine in China. In this paper, the flow heat transfer characteristics of gas passing through different curvature tubes are calculated by numerical simulation. The numerical simulation of tube rows with curvature of 0, 4.2, 8.4 and 12.6 is carried out in the range of $8000 < Re < 25000$. The following conclusions are drawn:

The main results are as follows: (1) when the fluid passes through the tube with different curvature, the Nussel number increases with the increase of Re, the friction coefficient decreases with the increase of Re, and the heat transfer power per unit volume increases with the increase of Re.

(2) The curvature has little influence on the heat exchange efficiency, but the increase of the flow resistance is obvious; after the curvature is increased to a certain range, the number of Nu increases with the curvature, and the resistance decreases with the increase of the curvature; and the increase of the curvature can increase the heat exchange power of the unit volume.

(3) the design of a pre-cooler with a unit volume heat transfer power of 409.2 MW can reduce the 1350K flow within 0.78mm by 532K, which provides a certain reference value for the design and application of high efficiency compact pre-cooler in the future.

References

[1] TONG Chuanchen, LOU Decang. Application of Pre-Cooling Technology on the TBCC Propulsion System [J]. Gas Turbine Experiment and Research, 2013, 26 (6):21-26.

[2] GUAN Wayout, LI Lifu, LIN Yongman. The study on heat transfer characteristic of coil's internal-melt ice storage tank for elliptical tube [J]. International Journal of Engineering and Industries, 2012, 3 (2):46-52.

[3] Myeong-Gie Kang. Effect of tube pitch on pool boiling heat transfer of vertical tube bundle [J]. Journal of Heat & Mass Transfer, 2016, 13 (4):485-496.

[4] Bacellar D, Aute V, Huang Z W. Airside friction and heat transfer characteristics for staggered tube bundle in crossflow configuration with diameters from 0.5 mm to 2.0 mm [J]. International Journal of Heat and Mass Transfer, 2016, 98:448-454.

[5] Kong Y Q, Yang L J, Du X Z, et al. Air-side flow and heat transfer characteristics of flat and slotted finned tube bundles with various tube pitches [J]. International Journal of Heat and Mass Transfer, 2017, 99:357-371.

[6] DONG Qiwu, OU Yangke, Liuminshan et al. Numerical Investigation of Fluid Flow Across Tube Bundles in Zukauskas Experimental Model [J]. Pressure Vessel Technology, 2010, 27 (01):21-26.

[7] Ahn J, Sparrow E M, Gorman J M et al. Turbulence intensity effects on heat transfer and fluid-flow for a circular cylinder in crossflow [J]. INTERNATIONAL JOURNAL OF HEAT AND MASS TRANSFER, 2017, 113:613-621.

[8] Wang Y Q, Gu X, Jin Z L et al. Characteristics of heat transfer for tube banks in crossflow and its relation with that in shell-and-tube heat exchangers [J]. INTERNATIONAL JOURNAL OF HEAT AND MASS TRANSFER, 2016, 93:584-594.
[9] LI Chenpei, WANG Yueshe, WANG Haijun et al. Numerical Analysis of Heat Transfer in Precooler for Hybrid Airbreathing Rocket Engines [J]. JOURNAL OF ENGINEERING THERMOPHYSICS, 2017, 38 (04):811-816.

[10] Li X, Wu X, He S. Numerical Investigation of the turbulent Cross flow and Heat Transfer in a wall Bounded tube Bundle [J]. International Journal of Thermal Sciences, 2014, 75:127-139.

[11] LIU Minshan, YANG Fan, DONG Qiwu et al. Effect of Wall Function on Simulation of Flowing Crossing Pipe Bundle [J]. JOURNAL OF ENGINEERING FOR THERMALENERGY AND POWER, 2010, 25 (05):497-500.

[12] GRIMSON E D. Correlation and utinization of new data on flow resistance and heat transfer for crossflow of gases over tube banks [J]. Trans ASME, 1937, 59 (3):583-594.

[13] Zukauskas. Convection heat transfer in heat exchanger[M] Beijing; Science Press, 1986.