International Journal of Heat and Mass Transfer 93 (2016) 584–594

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

# Characteristics of heat transfer for tube banks in crossflow and its relation with that in shell-and-tube heat exchangers

Yongqing Wang, Xin Gu, Zunlong Jin *, Ke Wang

*Key Laboratory of Process Heat Transfer and Energy Saving of Henan Province, Zhengzhou University, Zhengzhou 450002, China*

ARTICLE INFO

*Article history:*
Received 19 January 2015
Received in revised form 6 October 2015
Accepted 8 October 2015

*Keywords:*
Heat exchanger
Tube bank
Fluid flow
Heat transfer enhancement
Velocity components

ABSTRACT

The thermodynamics performances for tube banks in crossflow and for the shell sides of shell-and-tube heat exchangers were investigated, and the relation of fluid flow and heat transfer between them were analyzed. The results indicate that the incline degree of tube does not lead to obvious change on characteristics of fluid flow and heat transfer for fluid flowing across tube banks. Under different incline degrees of tubes, the characteristics of fluid flowing across tube banks are similar concerning fluid velocity components and local heat transfer along across angles. The ratios of vertical velocity component to parallel velocity component remain about 4.2 in the banks. Decreasing of impacting angle in the model reduces average fluid velocity crossing tubes, which weakens heat transfer. Characteristics for fluid flowing across tube bundles in shell sides distinguish from that for tube banks in crossflow with different impacting angles. At the same mass rate, the values of parallel velocity components in different shell sides are equivalent, but the vertical velocity components vary greatly. The ratios of vertical velocity component to parallel velocity component change from about 0.33–0.92 in the three types of shell sides. Many different leakage paths and bypass streams leads complicated fluid flow in shell sides of heat exchangers. The complicated flow pattern is not a simple combination of fluid flowing across tube banks with different impacting angles. The thermodynamics performance of shell side depends greatly on the vertical velocity component of fluid. The achievement in the paper can be a reference for study and development of heat exchanger.



## 1. Introduction

Heat exchangers play an important role in product quality, energy utilization, and systemic economy efficiency. In many types of heat exchangers, the tubes are arranged as a bundle or a bank in an in-line or staggered manner. Typically, one fluid flows over the tube banks, while the other fluid with a different temperature passes through the tubes [1,2]. With the fluid outside tubes flowing cross over tube bundle or bank, the transverse velocity component of fluid that impinging on tubes vertically is augmented greatly. A nonuniformity of the flow distribution accompanying the secondary flow comes into been, which enhances the turbulence and augments the heat transfer on the tube surfaces, and the characteristics is very different from that of fluid flowing parallel to tubes [3].

The heat transfer characteristics for tube banks in crossflow are of important practical interest [4], which also represents an idealization of many other industrially important processes, such as flow in filtration, biological systems, fibrous media as encountered in polymer processing and in insulation materials [5]. Therefore, it is of importance to study characteristics of fluid flow and heat transfer for tube banks in crossflow, which would contribute to predict the heat transfer and to design or develop heat exchanger appropriately [3].

The convective heat transfer characteristics for tube bundles or banks in crossflow have been developed over the ranges of geometric characteristics covering many practical needs [6]. Study on the heat transfer characteristics of crossflow over tube banks has been extensively conducted from the early of 20th century. Colburn [7] proposed a correlation of heat transfer for flow across tube banks with staggered tubes. Zukauskas [8] reported extensive experimental data for heat transfer and fluid flowing across tube banks, and proposed a correlation of the heat transfer coefficient as the function of the fluid Reynolds numbers, Prandtl numbers of fluid and tube surface. Hausen [9] presented an improving empirical correlation incorporating the tubes arrangement factor *Fa*. Pis'mennyi [10] analyzed experimental data and universal methods of calculating convective heat transfer of tube bundles

* Corresponding author.
*E-mail address:* zljin@zzu.edu.cn (Z. Jin).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.10.018

in crossflow, and generalized the dependence of the Reynolds number exponent on tube pitch characteristics in a tube bundle.

A number of studies have been carried out by using the numerical method to analyze characteristics of fluid flow and heat transfer for tube banks in crossflow with different geometries. By applying a calculation procedure for two-dimensional elliptic flow and using an implicit finite volume procedure, Wilson [11] predicted the pressure drop and heat transfer characteristics of laminar and turbulent flow for air flowing across tube banks. Mandhani [5] investigated numerically the heat transfer characteristics for an incompressible, steady crossflow over a tube bundle, and addressed the prediction of the Nusselt numbers in terms of the Reynolds and Prandtl numbers for a range of voidages of tube banks. Khan [2] investigated heat transfer for tube banks in crossflow under isothermal boundary condition. In the study, analytical models were developed for the heat transfer from tube banks, and the flow was assumed as steady, laminar, and fully developed. Kim [3] carried out an analytical investigation by using the CFD code FLUENT to evaluate the flow and heat transfer characteristics of in-line tube banks, and proposed a correction factor for the Zukauskas' correlation considering the effect of the longitudinal pitch. Li [12] presented a simulation of the fluid flow and heat transfer in a wall bounded in line tube bundle, and analyzed the local and average flow and heat transfer characteristics.

The shell-and-tube heat exchanger (STHX) is a typical device with fluid flow and heat transfer for tube bundles in crossflow, and is the most commonly used equipment in industrial heat transfer fields [13]. In the shell side of STHX, baffle is a key component. Besides supporting the tube bundle, baffles lead fluid flowing across a bank of tubes with some flow regime. The crossflow regime affects greatly the characteristics of fluid flow and heat transfer in shell side of STHX [14]. Characteristics of tube bundles in crossflow are of particular importance in the design or development of heat exchangers.

In STHXs, the characteristics of fluid flow and heat transfer in shell side is a typical application of tube bundle in crossflow. Although STHXs are applied satisfactorily in engineering fields for lots of years, studying on thermodynamic characteristics, structure development and optimization, applications of heat transfer enhancement techniques in shell sides of STHXs are gotten attentions continually. With development of heat transfer enhancement and equipment manufacture techniques, many novel tube bundle support structures emerged to enhance heat transfer and reduce pumping power [15], such as innovative segmental baffle, rod baffle and helical baffle. Vukic [16] performed experiments of STHX with segmental baffle, and found that heat transfer strongly depends on the shell side geometry, including baffle cut size, baffle distance, and baffle pitch. Li [17] studied the response of the pressure drop and local heat transfer in the shell side of STHXs to a change in the leakage between baffles and shell in a fully developed regime. You [18] experimentally investigated thermo-hydraulic performance in shell side of a STHX with trefoil-hole baffles in the turbulent flow regime. Zhang [19] studied experimentally characteristics in shell side of shell-and-tube oil coolers with overlapped helical baffles and segmental baffles, and compared the performances. Basing on the minimization at economic view point, Mariani [20] presented a new quantum particle swarm optimization approach combining with Zaslavskii chaotic map sequences for STHX optimization. Taher [14] studied numerically the influence of baffle spaces on fluid flow and heat transfer on the shell side of heat exchangers with non-continuous helical baffles by using periodic boundaries. Based on the periodic model, Yang [21] carried out numerical simulations to investigate the effects of number and width of the sealing strip on flow and heat transfer in the shell side of a STHX with helical baffles. Nanofluid is a new engineering fluid to improve the performance of heat exchanger. Elias [22] studied the effect of different particle shapes on the overall heat transfer coefficient, heat transfer and entropy generation of STHXs.

In the paper, the characteristics of fluid flow and heat transfer for tube banks in crossflow with different impacting angles were investigated by adopting an idealized flow model. The flow velocity component magnitudes in the models were analyzed quantitatively. Three types of STHXs with different types of baffles were selected to investigate the thermo-hydraulic performances for tube bundles in crossflow in the shell sides. By comparing heat transfer performance and fluid velocity components, the relation of characteristics of fluid flow and heat transfer for tube banks in crossflow and for tube bundles in shell sides of STHXs was analyzed. The analysis and results provide reference for studying performance of tube bank or tube bundle in crossflow and developing STHXs with new structure in shell side.

## 2. Thermo-hydraulic performance for tube banks in crossflow

### 2.1. Numerical model

In heat exchangers, the fluid does not always flow cross tube bundles or banks vertically. Fluid flows pass tubes with different angles in various structures. Referring to the experimental device of Zukauskas [8], idealized flow models with different fluid impacting angles were built to study the overall thermo-hydraulic performance of tube bank and detailed characteristics of fluid flow and heat transfer around tubes. The geometric model is shown as Fig. 1. In the figure, $\beta$ is the impacting angle of fluid flow.

In the model, tube is selected as the common one used in tubular heat exchangers, with outside diameter of 25 mm. The tubes are arranged as a staggered layout with tube pitch of 32 mm. There are 10 rows of tubes arranged along fluid flowing, and each row contains 10 tubes. The whole width of model, $w$, is 320 mm, the thickness, $b$, is 200 mm, and the height, $h$, is 600 mm. The tubes are located in the middle of model at flow direction, and the lengths of inlet and outlet zones are equivalent. Five impacting angles are selected, 30°,45°, 60°, 75° and 90°.

In calculations, fluid is selected as liquid water with constant physical properties. Fluid flows along the direction of $x$ axis. The lower and upper faces vertical to the $x$ axis are inlet and outlet respectively. The inlet is set as velocity inlet condition, where the bulk temperature of fluid is 293.15 K. The outlet is set as pressure outlet condition, where the gauge pressure is 0 Pa. The left and right faces vertical to the $z$ axis are set as the symmetry conditions, and the front and back faces vertical to the $y$ axis are set as the adiabatic and stand non-slip wall conditions. The tube surfaces are set as stand non-slip wall conditions with a constant temperature of 283.15 K.

The computational grids around tubes in the model are shown in Fig. 2. The values of skewness and Jacobian ratio were used to check the element quality, and the check results indicate that the cell quality is acceptable for computation. To get grid independent solution, a model with refined grids was built for the model with the impacting angle of 30°, which contains the tubes with the biggest angle of inclination. When the number of girds were refined to 8,356,500 from the 2,455,200, the relative differences of Nusselt number and pressure drop are both less than 1% for Reynolds number of 20,000, and it can be concluded that the grid independent solution can be achieved.

The CFD code FLUENT is based on the finite volume method on a collocated grid and uses general purpose software to analyze problems in fluid dynamics. The FLUENT was used to solve the equations for the fluid flow and heat transfer in current calculations.