Applied Thermal Engineering 103 (2016) 428–433

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

Investigation of flow and heat transfer characteristics on different heat exchangers of air conditioner

Ni Liu *, Xiao-Ling Lai, Kai Yan, Hua Zhang

College of Energy and Power Engineering, University of Shanghai for Science and Technology, Shanghai 200093, China

HIGHLIGHTS

- A 3D thin-section model was established for two types of heat exchangers.
- The numerical calculations were validated by experimental results.
- Flow and heat transfer characteristics were investigated.

ARTICLE INFO

Article history:
Received 22 October 2015
Revised 5 April 2016
Accepted 18 April 2016
Available online 19 April 2016

Keywords:
Split air conditioner
Heat exchanger
Numerical simulation
Cross-flow fan

ABSTRACT

The characteristics of flow and heat transfer of fin and tube heat exchanger for the three-stage and semi-arc structure considering different geometric shell were investigated in this study. A three-dimensional representative thin-section model was established for modeling split air conditioner indoor unit by employing CFX 14.0 in ANSYS. The model was validated by comparing the differences of outlet air volume obtained numerically and experimentally within acceptable limits. The effects of geometric parameters for heat exchanger were investigated through numerical analysis on the outlet air volume and heat transfer performance. It was found that the semi-arc heat exchanger showed more uniform flow field distribution with fewer vortexes. And also it shows better heat transfer performance compared to the three-stage heat exchanger. The heat transfer capacity of the semi-arc model is 7.02% higher than that of three-stage arrangements. On the other hand, the optimized geometric shell structure showed higher performance, resulting in removal of stagnant flow region with more effective heat transfer at the same air volume.

1. Introduction

The heat transfer efficiency of heat exchanger has become one of the main researches dealing with how to improve the performance of split air conditioner (SAC) indoor units. Air enters from the rooftop into the inside part, driven by cross-flow fan (CFF), flowing through the fins of the heat exchanger, blew away from the lower front of the device. The volume and thermal characteristics of air is important to improving thermal performance, since the air needs to bear a great responsibility for heat transfer resistance. Therefore, different airflow structures are designed to get better heat transfer performance.

A number of related studies have been undertaken by using a variety of methods to optimize the airflow characteristics. As a result of the complicated hydrodynamic characteristics of internal flow, the higher cost and longer period of the experimental method, the use of numerical method has become increasingly common. However, the validation of numerical results still needs to be carried out by the experiments.

Since CFF has a great impact on the airflow structure by forming the complex flow field, it is important to study the whole flow field. There have been a number of numerical studies carried out to investigate CFF. Shih et al. [1] and Toffolo [2] simulated the internal flow field of the CFF by a commercial CFD code and discussed the relationship between the geometrical configuration and fan performance in a theoretical perspective. Other than the geometry of the housing and the orientation of the CFF, Dang and Bushnell [3] discussed the position and the magnitude of the eccentric vortex formed by the rotation of the fan. Unlike axial or centrifugal fans, there was still no long-established theory to design the CFF used in a split air conditioner due to the existence of an eccentric vortex. Many studies of eccentric vortex in CFF were investigated. Sun et al. [4] proposed a new design of the inlet guide vanes to reduce

* Corresponding author.
E-mail address: ni-liu@hotmail.com (N. Liu).

http://dx.doi.org/10.1016/j.applthermaleng.2016.04.084
1359-4311/

the size of the eccentric vortex. The eccentricity and the strength of the vortexes were investigated by Toffolo et al. [5]. Gabi and Klemm [6] studied the aerodynamics of the flow and vortex regions inside the CFF. Most of these studies were carried out experimentally and numerically. Current studies of CFFs followed other more developed experimental methods. Hirata et al. [7] studied short span CFFs without casings (rear wall and vortex wall) using the PIV method for flow visualization.

Except the studies on CFF alone, there are also a number of studies on heat exchangers, (the main part of split air conditioner indoor units). Alizadehdakhel et al. [8] studied the simultaneous evaporation and condensation phenomena in a thermosyphon. Different types of heat exchanger, plain and wavy fin-tube, four-stage and arc-shaped have been used to enhance heat transfer performance. The effects of different geometrical parameter of heat exchanger on the heat transfer and pressure drop characteristics have been investigated by Bhuiyan et al. [9–11]. Deng and Meng [12] designed a kind of four-stage heat exchanger, to avoid the uneven internal flow field. Hu et al. [13] compared the outlet flow volume and air noise of four kinds of the heat exchangers with different shapes. As a result of the significantly increased outlet air volume and effectively reduced vortex (which improves the overall transfer performance), the use of arc-shaped heat exchanger was proposed, although four-stage heat exchanger is still a commonly used in air conditioner. The experimental data of different shape heat exchangers have been obtained though performance test in a wind tunnel by Tuztas and Egrican [14]. Wu et al. [15] studied the inlet flow field by comparing the impact of three stage and arc-shape heat exchangers as well as different front panel of the indoor unit. Again, some different experimental methods were used to investigate the heat transfer performance by changing the heat exchanger shape. Karadeniz et al. [16] showed the temperature distribution at the outflow section of the SAC indoor unit by stereo particle image velocimetry (SPIV) method and a novel application of infrared thermography called meshed infrared thermography (MIT). In the study by Kumlutas et al. [17], SPIV measurements at the outer section of the arc-shape heat exchanger model were conducted to compare the velocity distribution.

In this study, a three-dimensional thin-section model was established for modeling a SAC indoor unit by employing CFX 14.0 in ANSYS. Results are presented in the form of streamlines patterns, velocity vectors, temperature and pressure distribution, heat transfer coefficient and heat flux distribution. The characteristics of flow and heat transfer performance of the three-stage and semi-arc shape heat exchanger considering different geometric shell were analyzed to provide a reference for the air conditioner optimization.

2. Physical model

The size of SAC indoor unit cabinet is 850 mm × 290 mm × 190 mm. The copper tube outer diameter of the heat exchanger is 7 mm, with tube spacing of 21 mm, row spacing of 12.7 mm, fin spacing of 1.5 mm and fin thickness of 0.095 mm. The specification of wind wheel is 102 mm × 665 mm (see Fig. 1).

In order to reduce the amount of computer calculation process, only the representative thin section was modeled. Considering that the intermediate fins were affected by both sides of the fin, three fins were modeled as the research object, shown as Fig. 2c. The model of SAC indoor unit was embedded into a semicircular area to make better boundary conditions consistent with the actual situation. This area represented the air at the outside of the device. The diameter of the outside air region was determined to be twenty times of the fan diameter, as shown in Fig. 2b.

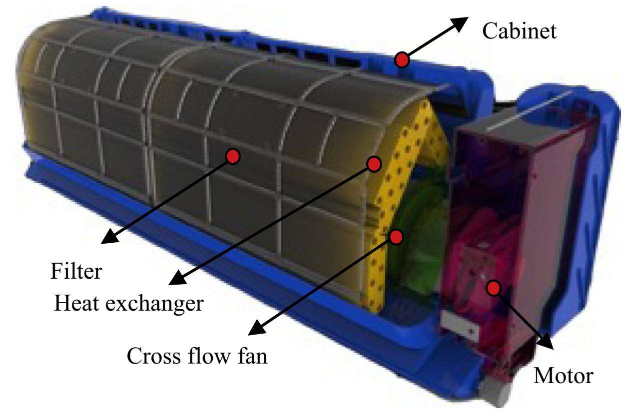

Fig. 1. Air conditioner indoor unit assembly.

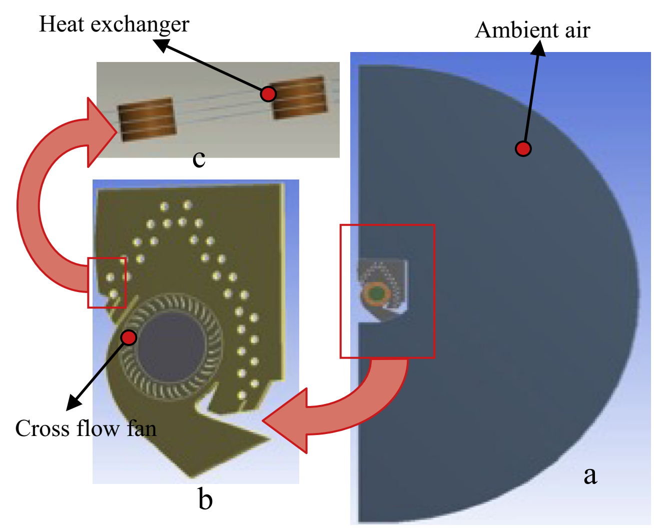

Fig. 2. 3D numerical model (a) indoor unit model, (b) regions of 3D model, (c) heat exchanger part.

3. Numerical setup

To generate a proper numerical grid, unstructured tetrahedral meshes were used in this model. The region around the fan impeller, import, export and especially fins need to choose a smaller mesh. The details of the numerical grid, which including 124,435 nodes, 525,836 meshes are given in Fig. 3.

Considering the complicated airflow characteristics, it was assumed that the air temperature was constant, incompressible, and the numerical parameters were modeled for standard cooling conditions defined in the TS EN 14511:2007 and ISO 5151 standards. The fins and tube surface temperature were set constant at 280 K. The inlet and outlet boundary conditions of the numerical model were kept as the outside temperature of 300 K and relative pressure was assumed to 0 Pa. The standard k–ε turbulence model has been used in this paper due to its stable, numerically robust and wide capability as it was recommend by several previous literature [17,18].

4. Grid sensitivity check

As there is a strong correlation between the mesh resolutions and the accurateness of the numerical results, a number of trial