Building and Environment 144 (2018) 268–280

Contents lists available at ScienceDirect

Building and Environment

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

Experimental and numerical study of space station airflow distribution under microgravity condition

Congcong Wang[a], Junjie Liu[a], Wenjin Shang[b], Hejiang Sun[a,*], Jiayu Li[a], Fenghua Fan[c]

[a] Tianjin Key Laboratory of Indoor Air Environmental Quality Control, School of Environmental Science and Engineering, Tianjin University, Tianjin, 300072, China
[b] Astronaut Center of China, Beijing, 100094, China
[c] China Academy of Building Research Tianjin Institute, Tianjin, 300072, China

ARTICLE INFO

Keywords:
Particle image velocimetry
Thermal comfort
Shrinkable ratio method
CFD

ABSTRACT

A space station that operates under microgravity conditions is a closed environment where a reasonable airflow distribution is required to eliminate body heat dissipation, remove contaminants and thus keep the crew comfortable. However, design of a reasonable airflow distribution has remained challenging. In this study, a computational fluid dynamics (CFD) methods with various turbulence models were used to investigate the airflow distribution inside a space station under microgravity conditions. To compare the performance of different models, the shrinkable ratio method was used to set up a mockup station to eliminate the influence of gravity-induced natural convection. The air velocity distribution in the narrow scale model was measured using particle image velocimetry (PIV). Results showed that the performance of the standard k-ε turbulence model was better than the renormalized group (RNG) k-ε turbulence model. The air distribution was optimized by changing the angle of the air supply outlet, suggesting that a three-dimensional air supply can provide better thermal comfort and higher air quality.

1. Introduction

A space station is a closed environment that operates under microgravity conditions. To keep the crew comfortable, a reasonable airflow distribution is required in the space station that creates suitable local air velocity within a specified range, eliminates body heat dissipation and removes contaminants [1]. However, it has remained challenging in designing a reasonable airflow distribution for the space station due to two main aspects. One is that compared with a typical building environment, the thermal load of a space station is large because of the dense occupancy and equipment. Another is that only forced convection is available in a microgravity environment to keep astronauts comfortable, which results in a weaker removal of human heat [2,3].

Two categories of methods are commonly used to study airflow distribution in such enclosures, i.e., experimental measurements and numerical simulations. In a space station during operation, experimental study on airflow is impractical. Instead, ground test of airflow is usually performed. However, it is restricted by many factors such as requiring considerable material resources and being laborious [4]. More importantly, the ground test usually cannot simulate a real environment under microgravity conditions, thus it cannot deliver satisfactory accuracy in analysis. With the improvements in both computer processing speed and advanced simulation methods, computational fluid dynamics (CFD) methods are widely used in designing and optimizing the air environment of closed spaces, such as large aircrafts, submarines and buildings [5]. Underlying such methods, turbulence models are critical in determining simulation accuracy, which need to be validated using experimental data and in turn provides guidance on evaluating and designing airflow distribution in the space station [6].

Various turbulence models have been used. For example, the RNG k-ε turbulence model was used to study velocity characteristics in the cabin X-38 Crew Return Vehicle at the International Space Station (ISS) in Eckhardt's research [7]. The RNG k-ε turbulence model was also used to study the airflow distribution in a space cabin [8]. Smirnov et al. used the standard k-ε turbulence model to characterize the airflow in the Columbus module [9]. Chang et al. studied the airflow distribution and accumulation of carbon dioxide in ISS Node 2 and the service module by using the standard k-ε turbulence model [10–12]. Edward et al. continued to use the standard k-ε turbulence model to simulate air circulation in the ISS based on Chang's results [13]. Darrah et al. found the numerical results of airflow in node 1 of the ISS model by using large eddy simulation (LES), which produced results similar to the

---

[*] Corresponding author.
E-mail address: sunhe@tju.edu.cn (H. Sun).

https://doi.org/10.1016/j.buildenv.2018.08.017
Received 20 June 2018; Received in revised form 2 August 2018; Accepted 9 August 2018

standard k-ε turbulence model [14]. Zheng et al. [15] used the standard k-ε turbulence model to study patterns in upside air-supply and bottom air-return symmetric ventilation in cabins under microgravity conditions. The study suggested that field velocity is at a maximum when the air supply is at 45° angle. Fu and Pei investigated the impact of the inter-module ventilation on a space station by using the realizable k-ε turbulence model [16]. However, the turbulence models and results of the above numerical simulation studies lacked experimental verification. Therefore, the most suitable turbulence model for studying airflow in a space station still needs to be identified.

To validate the CFD method for studying the airflow in a space station, Chang et al. [1,5] analyzed the air distribution characteristics of the ISS using CFD and a ground test, respectively, from 1993 to 1994. They also used the k-ε turbulence model to conduct numerical analysis. The experiment eliminated the effects of gravity on natural convection by conducting airflows under isothermal conditions in a full-scale model. However, their compromised approach did not take a practical microgravity cabin environment into consideration. To verify numerical simulation results, Wei et al. [17] used experimental data from a ground test of a capsule. The results showed that the largest error in air speed value was less than 25% and that the standard k-ε turbulence model was in agreement with the experimental value. In their experiment, only a few velocity points in the airflow field were measured because of the limitations in the instrument. Smirnov et al. [4] used a high Reynolds number k-ε model and LES to study the velocity characteristics of the Columbus module and to compare the numerical results with the measurement data. They concluded that the steady-state Reynolds Averaged Navier–Stokes (RANS) approach is a reliable method for evaluating air ventilation, and that, although the LES method to analyze airflow is time-consuming, it provides a deeper understanding about the behavior of unsteady flows. However, their ground experimental study using full-scale geometric models did not consider the effects of microgravity neither.

Thus, there has been a great need for more advanced experiments to validate the numerical research under microgravity on airflow. For the ground experiments of space station environment, the primary issue is to reduce the effects of gravity on the ground experiment. Numerous investigations have been conducted by using the ground experiment method to simulate a space station. Several main methods have been summarized including isothermal method and analogy method based on similarity theory. The analogy method based on the similarity theory includes a stress relief method [18–20], temperature-material-Nu number comprehensive hold down method [21–23], and a shrinkage ratio method [24]. The studies showed that the isothermal method could eliminate the influence of natural convection and restore forced convection movement. But this method cannot simulate the experimental study under non-isothermal conditions. The stress-relief method could achieve a heat flow transfer effect similar to the microgravity condition in a ground module, but the cabin pressure must drop to 0.3 atm, which was difficult to achieve. The comprehensive hold-down method using temperature material and Nu number involved, for example: depressurization, size changes, and changes in gas composition. It also required data correction. The shrinkage ratio method guaranteed a thermal-fluid rule number by adjusting geometry and boundary conditions. The narrow proportion needed a minimum ratio of 1:5. Because the size is narrowed, the cabin was easy to set up. Therefore, the shrinkage ratio method is more suitable for the study of microgravity airflow distribution on the ground. The instruments commonly used to measure the airflow in shrinkage scale space cabin model are hot sphere anemometer and hot wire anemometer. These sensors are embedded in the airflow field, which will increase the error of measurement velocity. Nowadays, the matured particle image velocimetry (PIV) technology can accurately measure the velocity vector without affecting the flow of cabin.

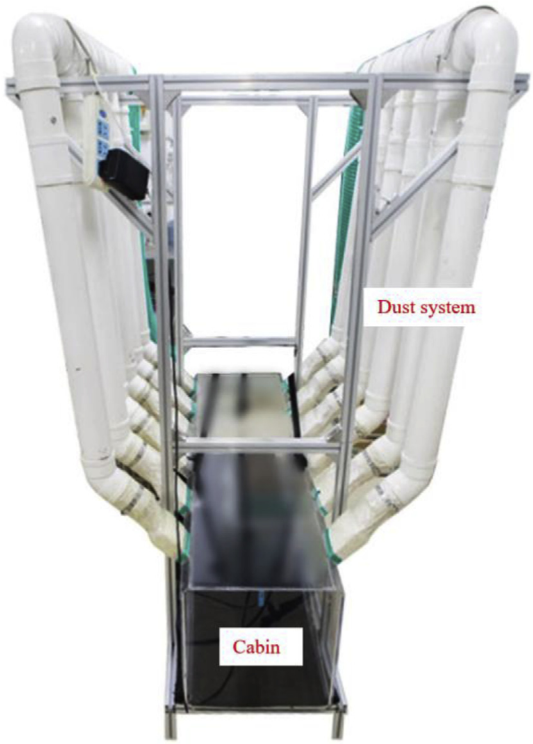

Fig. 1. The scale ratio of 1:5 for the test rig (narrow scale model) of the space station.

Table 1
Static pressure of the branch duct after leveling.

Air supply inlet (No.)	1	2	3	4	5	6
Static pressure (Pa)	318	316	319	318	314	315
Air supply inlet (No.)	7	8	9	10	11	12
Static pressure (Pa)	315	314	313	316	317	312

As the numerical method used to study the airflow distribution in microgravity environment lacked accurate and systematic experimental verification, this paper uses shrinkage ratio method to study the airflow distribution under microgravity condition by PIV. The performance of turbulence models is compared with the PIV experiment data to determine the optimal model for CFD simulation in the microgravity environment. Additionally, the selected turbulence model was used to design the most reasonable airflow distribution in the space station.

2. Research method

2.1. Experimental measurements

Natural convection in a space station is weak due to the microgravity conditions. However, natural convection on the ground is greatly enhanced because of gravity. The influence of gravity on natural convection depends on its relative importance to forced convection as shown by the non-dimensional equations of continuity, momentum, and energy [25]:

$$\nabla \cdot U = 0 \tag{2–1}$$

$$U \cdot \nabla U = \frac{1}{\mathrm{Re}}\nabla^2 U - \frac{1}{\mathrm{Re}^2}\nabla P - \frac{Gr}{\mathrm{Re}^2}Te_g \tag{2-2}$$

$$U \cdot \nabla T = \frac{1}{\mathrm{PrRe}}\nabla^2 T \tag{2–3}$$

$$Gr = \frac{g\beta(T_s - T_\infty)L^3}{\nu^2} \tag{2–4}$$

Where U refers to velocity, T refers to temperature, and P refers to