Energy and Buildings 127 (2016) 700–713

Contents lists available at ScienceDirect

Energy and Buildings

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

Modeling of gasper-induced jet flow and its impact on cabin air quality

Zhu Shi[a], Jun Chen[a], Ruoyu You[a], Chun Chen[b], Qingyan Chen[a,c,*]

[a] School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, USA
[b] Department of Mechanical and Automation Engineering, The Chinese University of Hong Kong, Shatin, N.T. 999077, Hong Kong Special Administrative Region
[c] Tianjin Key Laboratory of Indoor Air Environmental Quality Control, School of Environmental Science and Engineering, Tianjin University, Tianjin 300072, China

ARTICLE INFO

Article history:
Received 27 January 2016
Received in revised form 12 May 2016
Accepted 12 June 2016
Available online 15 June 2016

Keywords:
Gasper
Jet flow
Computational fluid dynamics
Mathematical modeling
Entrainment

ABSTRACT

Gaspers are prevalently installed in aircraft and automobiles to provide supplementary ventilation and improve passengers' thermal comfort. This investigation employed the SST $k-\omega$ model to simulate gasper-induced jet flow with the use of detailed gasper geometry, and validated the simulation results using experimental data. The validated CFD results not only revealed the mixing mechanism of a gasper-induced jet with ambient air, but also enabled the development of two mathematical models for characterizing the jet development along the gasper axis and radial velocity profiles in the downstream region. Furthermore, these models enabled the prediction of the entrainment ratio at different locations along a gasper-induced jet, and this ratio was used to evaluate the impact of the jet on air quality in a passenger's breathing zone. The performance of the CFD model and mathematical models was evaluated, and the models were compared on the basis of prediction accuracy and computing time.

1. Introduction

Traveling by air has become increasingly popular among passengers around the world. It was reported that the number of passengers carried by air transport reached 2.98 billion in 2012 [17]. The International Civil Aviation Organization has estimated that this number will reach 6 billion by 2030 [18]. As air travel becomes more frequent, the issue of airborne infectious disease transmission in aircraft cabins has been addressed collectively by researchers from various disciplines. In particular, the spread of severe acute respiratory syndrome (SARS) [24] and H1N1 influenza [5] in airplane cabins has suggested that the transmission of airborne infectious diseases could be associated with the airflow patterns in these spaces [20]. Therefore, it is necessary to investigate the airflow distribution in aircraft cabins in order to improve the design of cabin ventilation systems and reduce the risk of infection.

Current aircraft cabins use mixing ventilation, which supplies fresh air from diffusers at or near ceiling level and returns air through exhaust openings at the floor level on the cabin walls. Such a ventilation system creates large re-circulation patterns in the cabins, so that the air temperature and contaminant concentration distributions are uniform However, it was found that mixing ventilation could easily lead to the transmission of airborne infectious diseases among passengers [39]. In addition to mixing ventilation, researchers have investigated the airflow distribution in an aircraft cabin with other ventilation modes. For example, Gao and Niu [10] studied the distribution of air from a personalized ventilation nozzle and concluded that the personalized system could reduce a passenger's exposure to air pollutants by up to 60%. Zhang and Chen [39] also found that personalized ventilation could provide better cabin air quality than mixing or displacement ventilation without the risk of draft. Other studies [21,40] also reported an increased comfort level when personalized air supplies were incorporated in the seat headrest and seat back. Hence, personalized ventilation has the potential to reduce the risk of airborne infectious disease transmission in aircraft cabins, while at the same time improving passengers' thermal comfort.

Gaspers are the most commonly used personalized ventilation devices in aircraft cabins, and they are installed at ceiling level and directly above passengers' seats. By adjusting the opening area and angle of a gasper, a passenger can change the airflow rate and direction of the jet induced by the gasper. A typical gasper consists of

---

[*] Corresponding author at: School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, USA.
E-mail address: yanchen@purdue.edu (Q. Chen).

http://dx.doi.org/10.1016/j.enbuild.2016.06.038
0378-7788/

Nomenclature

A	Gasper opening area
D	Characteristic diameter of gasper outlet
D_e	Equivalent diameter
G_k	Generation of turbulent kinetic energy due to mean velocity gradient
G_ω	Generation of ω
K_1, K_2	Coefficients
k	Specific kinetic energy
M	Momentum flux
p	Pressure
P_{st}	Static pressure
$\dot{Q}$	Air flow rate
Re	Reynolds number
S_ϕ	Source term of scalar ϕ
U_c	Centerline velocity
$U_{\max}$	Maximum velocity on centerline
U_0	Initial velocity at gasper outlet
u_i	Velocity magnitude in direction i
u'_z	Turbulent fluctuation in z direction
x_i	Coordinate in the i direction
$x_{0.1}$, $x_{0.5}$ and $x_{0.9}$	Radial locations where z-velocities are 0.1, 0.5, and 0.9 respectively, of local maximum
$x'_{0.1}$, $x'_{0.5}$ and $x'_{0.9}$	Radial locations where turbulence intensities are 0.1, 0.5, and 0.9, respectively, of local maximum
Y_k, Y_w	Dissipations of k and ω due to turbulence
z_{peak}	Coordinate in the z direction of peak velocity
$\bar{z}$	Normalized z-direction location
$\Gamma_{\phi,eff}$	Coefficient of effective diffusion of scalar ϕ
ε	Entrainment ratio
δ	Gasper opening size
η	Gasper-induced jet airflow rate ratio
λ	Mixing layer thickness
μ	Dynamic viscosity
μ_t	Turbulent dynamic viscosity
v	Kinematic viscosity
v_t	Turbulent kinematic viscosity
ρ	Density
σ_k	Turbulent Prandtl number for k
σ_C	Schmidt number
$\sigma_{C,t}$	Turbulent Schmidt number
χ	Normalized x-direction location
ω	Specific dissipation rate
$\langle\phi\rangle$	Time averaged value of ϕ

an adjustable annular air outlet and an internal cone, which makes gasper-induced jet flow much more complicated than the jet from a simple round hole. It is important to understand how the jet develops after the air is discharged from a gasper outlet because the jet will interact with the main flow in the cabin.

To investigate gasper-induced jet flow, both experimental and numerical methods have been used. On the experimental side, for example, Anderson [2] used carbon dioxide (CO_2) as a tracer gas to study the effect of a personal gasper on airborne contaminant transport in an aircraft cabin. They found that the gasper played an important role in overall contaminant transmission. Guo et al. [13] used an anemometer to obtain high-resolution measurements of turbulence parameters in a jet airflow field induced by a gasper. These measurements provided reliable airflow data, but the method was time-consuming. On the other hand, computational fluid dynamics (CFD) has made it possible to predict the airflow distribution of gasper-induced jet flow. For instance, Baker et al. [4] and Gupta et al. [14] used a circular outlet to model gasper outlet flow in an aircraft cabin. By simplifying the gasper geometry as a round nozzle, they significantly reduced computational effort required for their simulations. However, such a simplification may cause errors, since the features of round jets are different from those of annular jets, particularly in jet developing region [33]. Moore et al. [23] and García-Villalba et al. [11] have used annular nozzles, but the geometry of the nozzles was not the same as the gasper geometry in aircraft cabins. You et al. [38] used detailed gasper geometry in simulating airflow in a mockup of a partial aircraft cabin with gasper on, but their study was more focused on the major flow in cabin. Therefore, it is needed to develop a CFD model that incorporates realistic gasper geometry, to study the flow characteristics of gasper-induced jets, especially in the initial development region of the jet flow.

Furthermore, both the air velocity and opening size of a gasper can vary. The jet velocity can be controlled by the environmental control system (ECS) [32,37], and the opening size is controlled by the passenger. Understanding how gasper-induced jet flow develops under various air velocities and opening sizes is essential for better characterization of the gasper-induced jet flow field and optimization of gasper geometry.

The present study built a CFD model for studying the jet flow from a gasper with detailed gasper geometry. The CFD simulation results were first compared with experimental results for the purpose of model validation. The CFD model was then used to explore how the second-order flow feature in the gasper-induced jet was influenced by initial air velocity and gasper opening size. Two mathematical models, based on jet initial velocity and gasper opening size, were developed, so that the airflow pattern and air quality in a gasper-induced jet could be effectively studied and efficiently predicted. The performance of the CFD model and mathematical models was evaluated in terms of prediction accuracy and calculation time, and on the basis of the evaluation, this study has provided suggestions for model selection.

2. Research method

As indicated by the review in the previous section, a CFD model can be an effective tool for studying the jet flow from a gasper with complex geometry. However, the CFD model should be validated by experimental data, as such a model normally uses a large number of approximations and assumptions.

2.1. Experimental setup

To validate the CFD model, this investigation used experimental data from Dai et al. [8]. Fig. 1 shows the experimental rig that was used. It consisted of a gasper, a plenum chamber made of Plexiglas, and an air supply pipe. An air compressor and an air storage tank were used to provide a clean, dry, and stable airflow to the gasper nozzle. Hot-wire anemometers were used to measure the air velocity in the downstream region. By employing an electronically controlled 3D translation stage, the measurements yielded data with a spatial resolution of 0.01 mm. The measurements were taken in a calm, isothermal environment.

2.2. Numerical method

To calculate the gasper-induced jet flow, this study used the shear stress transport (SST) $k-\omega$ model [34,22], because it has the best performance among the commonly used turbulence models in modeling a stratified jet [31]. According to Patankar [27] and White