Since January 2020 Elsevier has created a COVID-19 resource centre with free information in English and Mandarin on the novel coronavirus COVID-19. The COVID-19 resource centre is hosted on Elsevier Connect, the company's public news and information website.

Elsevier hereby grants permission to make all its COVID-19-related research that is available on the COVID-19 resource centre - including this research content - immediately available in PubMed Central and other publicly funded repositories, such as the WHO COVID database with rights for unrestricted research re-use and analyses in any form or by any means with acknowledgement of the original source. These permissions are granted for free by Elsevier for as long as the COVID-19 resource centre remains active.
Modeling of gasper-induced jet flow and its impact on cabin air quality

Zhu Shi, Jun Chen, Ruoyu You, Chun Chen, Qingyan Chen

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

A R T I C L E   I N F O
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

A B S T R A C T
Gaspers are prevalently installed in aircraft and automobiles to provide supplementary ventilation and improve passengers' thermal comfort. This investigation employed the SST k-ω 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.

© 2016 Elsevier B.V. All rights reserved.

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
Nomenclature

| Symbol | Definition |
|--------|------------|
| A      | Gaster opening area |
| D      | Characteristic diameter of gaster outlet |
| Dc     | Equivalent diameter |
| Gk     | Generation of turbulent kinetic energy due to mean velocity gradient |
| Gω     | Generation of ω |
| K1, K2 | Coefficients |
| k      | Specific kinetic energy |
| M      | Momentum flux |
| p      | Pressure |
| Pst    | Static pressure |
| Qi     | Air flow rate |
| Re     | Reynolds number |
| Sφ     | Source term of scalar φ |
| Uc     | Centerline velocity |
| Umax   | Maximum velocity on centerline |
| U0     | Initial velocity at gaster outlet |
| ui     | Velocity magnitude in direction i |
| uz     | Turbulent fluctuation in z direction |
| xi     | Coordinate in the i direction |
| x0.1, x0.5 and x0.9 | 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 |
| Yk, Yω | Dissipations of k and ω due to turbulence |
| zpeak | Coordinate in the z direction of peak velocity |
| z̃      | Normalized z-direction location |
| Iφ,eff | Coefficient of effective diffusion of scalar φ |
| ε      | Entrainment ratio |
| δ      | Gaster opening size |
| η      | Gasper-induced jet airflow rate ratio |
| λ      | Mixing layer thickness |
| µ      | Dynamic viscosity |
| µt     | Turbulent dynamic viscosity |
| ν      | Kinematic viscosity |
| νt     | Turbulent kinematic viscosity |
| ρ      | Density |
| σk     | Turbulent Prandtl number for k |
| σC     | Schmidt number |
| σc,t   | Turbulent Schmidt number |
| x̃      | Normalized x-direction location |
| ω      | Specific dissipation rate |
| ⟨φ⟩    | Time averaged value of φ |

an adjustable annular air outlet and an internal cone, which makes gaster-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 gaster outlet because the jet will interact with the main flow in the cabin.

To investigate gaster-induced jet flow, both experimental and numerical methods have been used. On the experimental side, for example, Anderson [2] used carbon dioxide (CO2) as a tracer gas to study the effect of a personal gaster on airborne contaminant transport in an aircraft cabin. They found that the gaster 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 gaster. 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 gaster-induced jet flow. For instance, Baker et al. [4] and Gupta et al. [14] used a circular outlet to model gaster outlet flow in an aircraft cabin. By simplifying the gaster 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 [23]. Moore et al. [23] and García-Villalba et al. [31] 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 gaster 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 gaster geometry, to study the flow characteristics of gaster-induced jets, especially in the initial development region of the jet flow.

Furthermore, both the air velocity and opening size of a gaster 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 − ω 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...
The transport equations for momentum, $k$, and $\omega$ can be expressed in a general form:

$$
\rho \frac{\partial \phi}{\partial t} + \rho \bar{u}_i \frac{\partial \phi}{\partial x_i} - \frac{\partial}{\partial x_i} \left[ \Gamma_{\phi, \text{eff}} \frac{\partial \phi}{\partial x_i} \right] = S_{\phi}
$$

where the general variable $\phi$, the source term $S_{\phi}$, and the coefficient of effective diffusion $\Gamma_{\phi, \text{eff}}$ of the $k$ - $\omega$ model are summarized in Table 1. No energy equation was solved because both the experiment and the CFD simulation were conducted under isothermal conditions.

The SIMPLE scheme was used for coupling the pressure and velocity equations. The second-order discretization method was used for the pressure, momentum, turbulent kinetic energy, and specific dissipation rate equations. This investigation assumed that the calculation was converged when the residuals of the continuity, $u_i$, $\phi$, $u_i$, $k$, and $\omega$ in Eq. (1) decreased to $3.1 \times 10^{-8}$, $2.3 \times 10^{-7}$, $2.3 \times 10^{-7}$, $4.2 \times 10^{-7}$, $1.1 \times 10^{-6}$, and $1.6 \times 10^{-5}$, respectively. This investigation used the ANSYS Fluent program V14.0 [3] to perform the numerical calculations.

To accurately model the gasper-induced airflow, it was necessary to implement the detailed geometry of the gasper in the CFD model. Fig. 2(a) is a photograph of the actual gasper installed in the cockpit of an MD-82 commercial airliner. In the present study, we built a CAD model for the gasper with detailed geometry as shown in Fig. 2(b). When the gasper was switched on, conditioned air was discharged through the annular opening (indicated in red in Fig. 2(b)). There was a solid cone inside the rotatable switch.

To model the case measured by Dai et al. [8], this study placed the gasper model (nozzle diameter $D = 12.6$ mm) into a cylindrical calculation domain with a diameter of 500 mm (40D) and a length of 600 mm (48D), as shown in Fig. 3. According to Fan et al. [9], this dimension is large enough for simulating a free jet. Since the geometry in this case was symmetric, one quarter of the geometry was used to construct the computation domain, as shown in Fig. 3(a). The plane $z=0$ was defined on the edge of the gasper’s rotatable switch (Figs. 2b and 3a). The velocity inlet boundary was set at the gasper outlet (the red area in Fig. 2b). The side and bottom boundaries of the calculation domain were defined as the pressure outlet, and standard atmospheric pressure was prescribed. A symmetry (zero-shear, no normal velocities) boundary was used for the two symmetric faces. All other boundaries were defined as no-slip wall boundaries. Independence tests were conducted to determine the size of the calculation domain. It was found that enlarging the domain to a cylinder with a length of 900 mm and diameter of 750 mm did not lead to a noticeable difference in simulation results. Hence, this further verified that the originally proposed size of the

---

Table 1: Definitions of $\phi$, $\Gamma_{\phi, \text{eff}}$, and $S_{\phi}$ in Eq. (1).

| $\phi$ | $\Gamma_{\phi, \text{eff}}$ | $S_{\phi}$ | Constants |
|--------|----------------|-----------|-----------|
| $u_i$  | $\mu + u_i$   | $-\frac{\partial u_i}{\partial x_i} + \frac{\partial}{\partial x_i} \left( \mu + \mu_i \right) \frac{\partial u_i}{\partial x_i}$ | $-\mu_i$ |
| $k$    | $\mu + \mu_k$ | $G_k - Y_k$ | $\sigma_k = \frac{1}{\tau_k}$ |
| $\omega$ | $\mu + \mu_\omega$ | $G_\omega - Y_\omega$ | $\sigma_\omega = \frac{1}{\tau_\omega}$ |

Fig. 1. (a) Schematic of the setup and (b) photo of the gasper and plenum chamber.

Fig. 2. (a) Photograph of the actual gasper (unit: mm) [8] and (b) detailed geometry of the gasper model used in this study.
calculation domain was sufficient to provide accurate results for the gasper-induced jet flow.

Fig. 4 shows the grid structure used in this study, where the grid sizes were non-uniform. The smallest meshes were used around the gasper nozzle (0.12 mm), and the mesh size gradually increased with distance from the gasper. A grid independence test was conducted by comparing the velocity profile along the jet direction with three different grid numbers (444,565, 1,445,920, and 2,914,101). As shown in Fig. 5, the resolution of 1,445,920 was sufficient for obtaining a grid-independent result for the velocity profile.

3. Results

This section first discusses the accuracy of the CFD simulation by comparing the numerical results with the corresponding experimental data for the jet that developed from the gasper. We then
describe the use of the validated CFD model to study the jet flow characteristics.

3.1. Validation with experimental data

Fig. 6 shows the simulated velocity profile at different downstream locations. The velocity profile starts with double peaks, which were caused by the annular gasper outlet. The two peaks gradually merge to one peak. Fig. 7(a) compares the calculated centerline velocity $U_c$ along the jet axis ($z$ direction) with the experimental data, where $U_c$ has been normalized by its maximum value $U_{\text{max}}$ and $z$ by the location of $U_{\text{max}}$ and $D$. Because of the Coanda effect [36], the air from the annular inlet immediately attached to the solid cone. At a certain distance from the inlet, the airflow around the cone merged to a single stream. This explains why $U_{\text{max}}$ did not occur at the inlet ($z = 0$). The comparison shows that the modeling results agreed well with the experimental data, especially after the merging point ($z = 3.2D$) where the flow was more developed. The mixing mechanism of this flow is illustrated in detail in Section 3.2.

This investigation also performed a self-similarity test on the peak velocity profiles in the region downstream from the jet. Pope [28] has discussed self-similar characteristics in turbulent round jets in detail. Fig. 7(b) shows $U_c/U_{\text{max}} - x/x_{1/2}$ profiles, where $x_{1/2}$ is the $x$ location at which the $z$-velocity is half of $U_c$. After normalization, the three profiles at the three different downstream locations collapse into almost a single profile, which strongly indicates self-similarity. The similarity profiles also matched well with the experimental data. Since the numerical model used in this study is able to predict the gasper-induced airflow with reasonably good accuracy, the model is considered to be validated.

In addition to mean velocities, this study also investigated the second-order flow feature in the gasper-induced jet. Fig. 8 is a plot of normalized turbulence intensity $\langle u'^2 \rangle^{1/2} / \langle u'^2 \rangle_{\text{peak}}^{1/2}$, where $\langle u'^2 \rangle^{1/2}$ represents turbulent fluctuation in the $z$ direction and $\langle u'^2 \rangle_{\text{peak}}^{1/2}$ the maximum turbulence intensity at a particular location, against normalized radial distances in the developing region (before the merging point) and fully developed region (after the merging point). The non-dimensional radial distances used were $(x - x_0)/x_0$ for $x_0 = 0.5$ and $x_0 = 0.1$, $x_0 = 0.5$, and $x_0 = 0.9$ were the radial locations where $\langle u'^2 \rangle^{1/2}$ reached 0.1, 0.5, and 0.9, respectively, of the corresponding local maximums. The comparison shows that in the developing region, where a bell-shaped jet dominated and had not merged towards the center, there was a “dent” at the flow center. In the fully developed region, the turbulence intensity profile was similar to that for a fully developed round jet [31]: the “dent” still existed, but it became significantly shallower. Overall, the CFD prediction of turbulence intensity agreed well with the experimental data, which further validated the CFD model for simulating the gasper-induced jet flow on the second-order flow characteristics level. The discrepancies between the CFD results and experimental data can be attributed
to the assumption in the SST $k-\omega$ model that kinetic eddy viscosity is isotropic [15], which is not strictly true.

### 3.2. Mixing mechanism of the gasper-induced jet

The validation of the CFD model enabled further study of the mixing mechanism in the gasper-induced jet through investigation of the streamlines of the mean flow field, mean velocity, and static pressure. **Fig. 9(a)** illustrates the streamlines in the initial developing region of the jet. It can be seen that before the merging point, the flow was still developing and vortices were generated. These vortices indicate a complicated flow pattern. Thus, this region was more difficult to model accurately in Section 3.1. The formation of these vortices can be explained by static pressure distribution, as illustrated in **Fig. 9(b)**; before the peak pressure point, a positive axial pressure gradient maintained the reverse flow, which formed vortices, and the local minimum pressure location coincided with the centers of the vortices. In this investigation, the $z$-stagnation point was defined as the location at which $U_z = 0$. In particular, the two $z$-stagnation points on the jet centerline were defined as $z_{s1}$ and $z_{s2}$. The distance between them, $z_{s2} - z_{s1}$, characterized the size of the vortices that were formed; in this case, the size was 0.3D. The second $z$-stagnation point marked the end of vortices, and also approximately coincided with the peak pressure point, which was in good agreement with what the observations of Rehav et al. [30] and Chigier and Beer [6]. After the second $z$-stagnation point, the static pressure dropped so dramatically that ambient fluid was quickly entrained towards the jet center. This explains the increase in centerline velocity after the second $z$-stagnation point in **Fig. 9(b)**, and the expansion of the $z$-stagnation lines in **Fig. 9(a)**.

To characterize the degree of mixing at different axial locations in the gasper-induced jet, the CFD model was also used to explore mixing layer thickness development in the flow. As recommended by Pope [28], the present study defined the mixing layer thickness as $\lambda = x_{0.1} - x_{0.9}$, where $x_{0.1}$ and $x_{0.9}$ are the radial locations at which the $z$-velocities are 0.1 and 0.9, respectively, of the maximum velocity at a particular $z$ location. As shown in **Fig. 10**, the normalized mixing layer thickness $\lambda/D$ was plotted against $z/D$ for cases 1 through 5 (for which the details are listed in Table 2), in order to determine the influence of gasper inlet velocity and opening size. Good self-similarity can be seen in the mixing layer thickness for different inlet velocities ($U_0$) and opening sizes ($\delta$). The mixing layer thickness increased continuously along the jet axis, which can be explained by the spread of coherent structures in the
3.3. Mathematical modeling of the gasper-induced jet

3.3.1. Development of empirical equations

i. Centerline velocity

As discussed in Section 3.2, various cases were designed using the validated CFD model to explore the influence of inlet velocity and opening size on mixing layer thickness. The same set of cases was used to investigate the impact of inlet velocity and opening size on centerline velocity distribution.

Fig. 11(a) compares the centerline velocity profiles of the gasper-induced jet flow with three different inlet velocities. The general trends of the profiles are very similar. The results also show that, with the same gasper geometry, the peak locations under various initial velocities are very close to each other, which agrees with the experiments in Dai et al. [8]. Furthermore, a plot of normalized centerline velocity $U_c/U_{max}$ versus normalized axial location $(z - z_{peak})/D$ (Fig. 11(b)) shows that these velocity profiles are closely matched with one another, especially as the velocity decayed along the centerline. This indicates that the flow characteristics after the peak location, where jet flow was more developed, tended to follow a more uniform pattern than in the region where flow was still developing.

Fig. 12 compares the centerline velocity distributions and normalized profiles in Cases 1, 4, and 5, where the gasper was adjusted to different opening sizes. As with the uniform pattern in Fig. 11(b), the normalized velocity profiles, $U_c/U_{max}$ versus $(z - z_{peak})/D$, also collapsed into a single curve. The $z_{peak}$ values, however, were found to increase as the opening size increased according to Fig. 12(a). This trend agrees with the results illustrated in Dai et al. [8], and the rea-

---

Table 2

| Case # | Inlet velocity ($U_0$, m/s) | Reynolds # ($Re$) | Opening size ($\delta$, mm) |
|--------|-----------------------------|------------------|--------------------------|
| 1 (Original) | 40                          | 32,102          | 0.9                      |
| 2      | 60                          | 48,153          | 0.9                      |
| 3      | 80                          | 64,204          | 0.9                      |
| 4      | 40                          | 32,102          | 0.7                      |
| 5      | 40                          | 32,102          | 1.1                      |

---

Fig. 9. Flow characteristics near the gasper inlet: (a) air velocity streamlines and (b) pressure and velocity distributions.

Fig. 10. Mixing layer thickness at locations in the downstream region of the jet.
son might be that under the same initial velocity, a larger opening size led to greater fluid momentum from the gasper inlet, which pushed the merging point forward. Meanwhile, due to greater momentum at larger opening, maximum axial velocity was also found to be larger as opening increased.

The analysis above indicates that the normalized centerline velocity profile is independent of the inlet velocity and inlet opening. Thus, this study developed an empirical equation for the centerline velocity. As mentioned in Section 2.4, the flow developments before and after the merging point are completely different, and thus the centerline velocity before and after the peak velocity location can be modeled separately as a piecewise function. Before the peak velocity location, a cubic function was used to describe the velocity profile, which increases dramatically at first, and then gradually becomes slower as the location becomes closer to the peak location. Experimental data [8] was used for curve fitting, which yielded the normalized velocity profile

\[
\frac{U_c}{U_{\text{max}}} = g(\bar{z}) = 1.49\bar{z}^3 + 2.31\bar{z}^2 + 1.19\bar{z} + 1.18, \quad \bar{z} < 0
\]

\[
\frac{U_c}{U_{\text{max}}} = g(\bar{z}) = 0.42\bar{z}^2 + 1, \quad \bar{z} > 0
\]

Thus, the present study has proposed a rational function to describe the velocity profile when \( \bar{z} > 0 \). Fitting the proposed rational function with experimental data yields

\[
g(\bar{z}) = \frac{0.42\bar{z}^2 + 1}{0.12\bar{z}^2 + 0.37\bar{z} + 1}, \quad \bar{z} > 0
\]

In summary, the new gasper centerline velocity formula (GCVF) is

\[
\frac{U_c}{U_{\text{max}}} = g(\bar{z}) = \begin{cases} 
1.49\bar{z}^3 + 2.31\bar{z}^2 + 1.19\bar{z} + 1.18, & \bar{z} < 0 \\
0.42\bar{z}^2 + 1, & \bar{z} > 0
\end{cases}
\]

Fig. 13 compares the centerline velocity predicted by the empirical equation, the simulation results, and the experimental data. It can be seen that the proposed empirical equation is able to predict the centerline velocity profile with good accuracy.

ii Radial velocity profile

After the merging point, the radial velocity profiles from the CFD calculations exhibited self-similarity characteristics, as shown
in Fig. 14. This study further developed an empirical equation for calculating the radial velocity after the merging point. Papanicolau and List [26], and Corrsin and Uberoi [7] reported a Gaussian velocity profile at a downstream location in a turbulent round jet. Assuming that the lateral velocity profile after the merging point follows the same pattern, the present study used a Gaussian equation to describe the relationship between the normalized z velocity and normalized lateral location. Experimental data was utilized to determine the parameters in the Gaussian equation. This yielded the gasper downstream radial velocity profile (GDRP):

$$\frac{U_z}{U_c} = e^{-\frac{x}{\bar{z}}} x^2. \tag{3}$$

where normalized x location, $x = x/x_{1/2}$ and $U_z$ is the z-direction mean velocity at a given point.

As shown in Fig. 14, the normalized lateral velocity profiles calculated by the empirical equation were in good agreement with the CFD calculation and the experimental data.

For application of Eq. (3), $x_{1/2}$ needed to be pre-determined. Kwon and Seo [19] reported a linear relationship between $x_{1/2}$ and the axial location $z$ in a free round jet. Fig. 15 is a plot of the normalized jet half width $x_{1/2}/D$ versus $\bar{z}$ from CFD results for the current gasper-induced jet flow. In the downstream region, a strong linear relationship can be observed between $x_{1/2}/D$ and $\bar{z}$, which is similar to that in a round jet. However, in the vicinity of the gasper inlet, the linear relationship does not exist. This investigation used a quadratic equation to describe the development of $x_{1/2}/D$ near the inlet region.

From these observations, this study developed an empirical equation to describe $x_{1/2}$:

$$x_{1/2} = \frac{h(\bar{z})}{D} = \begin{cases} 0.021\bar{z}^2 - 0.0087\bar{z} + 0.478, & \bar{z} \leq 3 \\ 0.116\bar{z}^2 + 0.270, & \bar{z} > 3 \end{cases}. \tag{4}$$

The empirical formulas (Eqs. (2) through (4)), when utilized together, can provide useful information about velocity in the gasper-induced flow field, and thus can be helpful for the analysis of gasper-induced jets and the design of gaspers. The empirical parameter $U_{\text{max}}$ and $z_{\text{peak}}$ are dependent on two key parameters, the peak centerline velocity $U_{\text{max}}$ and its location $z_{\text{peak}}$, which change from case to case. In order to make the mathematical models complete, this study further explored the relationships between these parameters and two gasper boundary conditions that can be easily measured: initial air flow rate $Q_0$ and opening area $A$.

Section 3.2 demonstrated that the peak in the centerline velocity profile was due to rapid entrainment of ambient fluid after $z$-stagnation point $z_{1/2}$. Moreover, Ricou and Spalding [16] have shown that in a round jet, the entrainment amount is proportional to $\sqrt{M_0}$, where $M_0$ is the initial momentum flux at the jet nozzle. Thus, this research has assumed that $U_{\text{max}}$ is correlated with $\sqrt{M_0}$. With the use of dimensional analysis, the relationship can be expressed as

$$\rho D^2 U_{\text{max}}^2 = K_1 M_0 \tag{5}$$

where $\rho$ is air density and $K_1$ is a dimensionless value.
Since $M_0 = \rho U_0^2 A$ and $Q_0 = U_0 A$, the above equation can be further expressed as

$$U_{\text{max}} = \sqrt{\frac{Q_0^2}{AD^2}}$$ (6)

**Fig. 15.** (a) Sketch of lateral velocity profile and (b) relationship between $x_{1/2}/D$ and $z$.

**Fig. 16a** is a plot of $U_{\text{max}}$ versus $\frac{Q_0^2}{AD^2}$, at opening sizes of 0.7 mm, 0.9 mm, 1.1 mm, and 1.3 mm. For each opening size, five $U_0$ values were tested: 40 m/s, 50 m/s, 60 m/s, 70 m/s and 80 m/s. These 20 sample points were well fitted to the power equation with $R^2 = 0.95$:

$$U_{\text{max}} = 0.93 \sqrt{\frac{Q_0^2}{AD^2}}$$ (7)

or

$$\frac{U_{\text{max}}}{Q_0/A} = 0.93 \sqrt{\frac{A}{D^2}}$$ (8)

On the other hand, according to Figs. 11 and 12, $z_{\text{peak}}$ was correlated with gasper opening size but not initial velocity. In Fig. 16b, $z_{\text{peak}}$ is plotted against $\delta$ at four gasper opening sizes. The results show that $z_{\text{peak}}$ increased almost linearly with $\delta$. The correlation between peak velocity location and opening size can be fitted by a straight line:

$$\frac{z_{\text{peak}}}{D} = 18.0 \frac{\delta}{D} + 1.4$$ (9)

which yields

$$\frac{z_{\text{peak}}}{D} = 18.0 \frac{A}{\pi D^2} + 1.4$$ (10)

Formulas (8) and (10) supplement Eqs. (2) through (4) and thus bring to completion the development of empirical equations for gasper-induced jet velocity.

3.3.2. Virtual origin model

Previous investigations have reported that because of self-preserving properties in the developed flow region, $U_0/U_c$ is proportional to $(z - z_0)/D$ in the downstream regions of a round jet [16,12] and an annular jet [6], where $z_0$ is defined as the virtual origin. In particular, $z_0$ represents the $z$-intercept of the straight line, $U_0/U_c$ versus $z/D$, and it represents a point source of momentum of the jet. In analogy, the present study has also explored the relationship between the inverse of centerline velocity and axial distance in a gasper-induced jet for cases demonstrated in Table 2, by plotting $U_0/U_c$ against $z/D$ as shown in Fig. 17. The results indicate a strong linear relationship between $U_0/U_c$ and $z/D$ after $z = 7D$, and the straight lines share a common $z$-intercept: 0.2D. The relationship is independent of initial velocity, but the slope of the straight line decreases as gasper opening size increases.

**Fig. 17** also shows the relationship between slope and opening size, and a fitting linear equation can be used to describe the relationship (an additional case with $\delta = 1.3\text{mm}$ was included for the linear fitting). On the basis of the above relationships, the virtual origin model for gasper-induced jet flow can be defined as:

$$\frac{U_0}{U_c} = (0.74 - 3.53 \frac{\delta}{D}) \frac{z - 0.2D}{D} = (0.74 - 3.53 \frac{A}{\pi D^2}) \frac{z - 0.2D}{D}$$ (11)

This simple model is able to predict centerline velocity with good accuracy in the downstream region of a gasper-induced jet. The axial velocities as a function of radial distance can be determined from Eq. (3) after the centerline velocity has been identified.

In comparison with the empirical equations method described in Section 3.3.1, the virtual origin model is a much simpler method that is able to predict the airflow velocity in the developed region of a gasper-induced jet. This model was created with the assumption that the jet originates from a point source of momentum spread linearly in the developed region, and it has also been used to predict the entrainment ratio, as discussed in the following section.

3.4. Entrainment analysis and impact of gasper-induced jet flow on cabin air quality

An important flow phenomenon in a gasper-induced jet is entrainment. In a previous study [29], the entrainment ratio was calculated as $\varepsilon = \frac{\dot{m}}{m_0}$, where $m_0$ is the mass flow rate at the jet nozzle, and $\dot{m}$ is the mass flow rate at a certain point downstream from the jet’s main stream. In the current study, the fresh air ratio is defined as $\eta = \frac{1}{\varepsilon} = \frac{m_0}{\dot{m}}$, in order to quantify the percentage of fresh air (the air that is discharged from the gasper nozzle) at a location in the downstream region of the jet. The fresh air ratio can be pre-
dicted by use of the validated CFD model, the empirical equations, and the developed virtual origin model, as follows.

(1) CFD method
With the validated CFD model, $\dot{m}$ can be simply evaluated by reporting the flow rate at different downstream locations inside the envelope formed by $z$-stagnation lines, which were defined in Section 3.4. The fresh air ratio $\eta$ can then be calculated by dividing the gasper inlet mass flow rate $\dot{m}_0$ by $\dot{m}$ at each downstream location.

(2) Empirical equations
On the basis of the empirical equations for the centerline and radial velocity profiles, the current study calculated the airflow rate at a certain cross-section as follows.

Fig. 18 demonstrates a 3D schematic of the Gaussian shape profile of $U_z$ at a downstream location. The air flow rate at a cross section can be calculated by integrating $U_z$ over the whole area of the cross section. When the GCVF and GDRP formulas are combined, the integration can be performed as follows:

\[
\dot{Q} = \int \int U_z dA = \int_{0}^{\infty} U_z \cdot 2\pi x dx = \int_{0}^{\infty} \exp \left(-\frac{5}{8} \frac{x^2}{x_{1/2}} \right) U_c \cdot 2\pi dx
\]

\[
x dx = \int_{0}^{\infty} \exp \left(-\frac{5}{8} \frac{x^2}{x_{1/2}} \right) U_c \cdot 2\pi dx
\]

Thus, the fresh air ratio can be calculated as:

\[
\eta = \frac{\dot{Q}(z=0)}{\dot{Q}(z)} = \frac{\dot{Q}(z=0)}{\dot{Q}(\bar{z}=0)} = \frac{\dot{Q}(z=0)}{\dot{Q}(\bar{z}=0)}
\]

\[
\int_{0}^{\infty} \exp \left(-\frac{5}{8} \frac{x^2}{x_{1/2}} \right) g(z) U_{\text{max}} \cdot 2\pi x dx
\]

\[
\int_{0}^{\infty} \exp \left(-\frac{5}{8} \frac{x^2}{x_{1/2}} \right) g(\bar{z}) U_{\text{max}} \cdot 2\pi x dx
\]
Although it is not possible to express the integral analytically, the definite integral can still be calculated using Newton–Cotes formulae [1].

(3) Virtual origin model

George et al. [12] illustrated that in a fully developed jet, the entrainment ratio $\varepsilon = \frac{m}{m_0}$ had a linear relationship with jet axial distance, and the straight line intersected with the abscissa at the virtual origin. Moreover, Ricou and Spalding [29] used experimental results to conclude that the slope of the straight line was $0.32(\frac{2D_e}{\rho_1\rho_0})^2$, where $\rho_1$ was fresh fluid density and $\rho_0$ was the density of ambient fluid. Given that the current gasper-induced flow is homogenous, and by use of the virtual origin model developed in Section 3.3.2, this relationship can be expressed as

$$\varepsilon = \frac{m}{m_0} = 0.32\frac{z - 0.2D_e}{D_e}$$

where the equivalent diameter $D_e$ is defined so that the mass flow rate discharged through the current gasper nozzle is equal to the mass flow rate discharged from a round nozzle with diameter $D_e$. When the continuity equation for mass is employed,

$$\dot{m}_0 = \pi D_0^2 U_0 = \pi \left( \frac{D_e^2}{2} \right) U_0$$

This expression leads to $D_e = \sqrt{4\dot{m}_0/D_0}$. Thus, the fresh air ratio can be calculated by use of the virtual origin model as:

$$\eta = \frac{1}{\varepsilon} = \frac{1}{0.32(\frac{z}{\sqrt{4\dot{m}_0/D_0}} - 0.2)}$$

FIG. 18. Z-direction velocity profile at a location in the downstream region of the gasper-induced jet flow.

(Fig. 19) show that the gasper-induced airflow ratio was about 5% in the breathing zone of the passenger. This means that, when the gasper was turned on, about 5% of the air in the breathing zone was from the gasper, while 95% of the air was from the cabin. This result indicates that the jet flow induced by a gasper has a very minor influence on the air quality in a passenger’s breathing zone.

4. Discussion

4.1. Comparison of the three modeling methods

In the present study, three modeling methods (CFD, empirical equations, and the virtual origin model) were used to investigate the flow characteristics of a gasper-induced jet. The predicting ability and prediction time of the three methods are summarized and evaluated in Table 3, where stars have been used to rate prediction accuracy and solution time. Overall, the CFD method yielded the most accurate results, and it was able to predict additional flow features such as pressure and turbulence intensity. However, this method required a significant amount of time for setup and numerical computation. The empirical equations also provided good predictions of velocity, as well as predicting the entrainment ratio in the developed region, although this method was not able to predict other flow features. The virtual origin model was the simplest and thus the least time-consuming method, and it was capable of predicting velocity and entrainment ratio in most parts of the gasper-induced jet region. Generally speaking, the CFD method is recommended when other flow features are needed in addition to velocity, whereas the virtual origin model is recommended when only velocity or entrainment information in the downstream region of a gasper-induced jet is required.

4.2. Limitations

This study modeled a gasper-induced free jet. Although such a jet usually has a much higher velocity than the ambient airflow, it is still possible that in a real case the jet flow features would be influenced by other flows, such as the flow from the main diffuser in an aircraft cabin or the breathing of passengers. In this study, it was assumed that ambient flow had only a minor impact on the jet.

In addition, this study assumed that the gasper-induced jet was an isothermal flow. In reality, however, ambient air, passengers’ heads, and the aircraft cabin walls may have different temperatures than that of the jet. When making conclusions about the impact of gasper-induced jet flow on the air quality in the breathing zone, this investigation assumed that temperature variations did not have a great influence on the flow in the jet mainstream.

5. Conclusions

This study has led to the following conclusions:

(1) The CFD model with the SST k-ω model can provide good predictions of gasper-induced jet flows. The velocity profiles in the transverse direction began as “double-peak” curves and were transformed to a “single-peak” curve after the merging point. The velocity profiles in the developing region were very different from those for free round jet flow.

(2) At the onset of a gasper-induced jet, the pressure gradient maintained reverse flow and generated vortices. Subsequently, the pressure dropped dramatically, which caused the centerline velocity to continuously increase until it reached the peak value. The mixing layer thickness continued to increase in the direction of the gasper-induced jet.

(3) The empirical equations and virtual origin model developed in this study were able to predict gasper-induced jet velocities with
good accuracy, and they were much less time consuming than the prediction with the CFD method. In comparison with the empirical equations, the virtual origin model was simpler in expression, whereas the empirical equations provided better centerline velocity prediction in the developing region. However, the CFD method was able to provide more comprehensive flow characteristics such as pressure and turbulence intensity in the flow field, which neither the empirical equations nor the virtual origin model were able to predict.

(4) Entrainment ratios predicted by the three modeling methods in the developed region agreed well with one another. The results showed that when the gasper was turned on, more than 90% of the air in the passenger’s breathing zone was entrained from the surroundings. Thus, a gasper-induced jet has only a very minor impact on the air quality in a passenger’s breathing zone.

Acknowledgement

The research presented in this paper was partially supported by the National Basic Research Program of China (the 973 Program) through Grant No. 2012CB720100.

References

[1] M. Abramowitz, I.A. Stegun (Eds.), Handbook of Mathematical Functions: With Formulas, Graphs, and Mathematical Tables, No. 55, Courier Corporation, Mineola, NY, 1964.
[2] M.D. Anderson, Effect of gases on airflow patterns and the transmission of airborne contaminants within an aircraft cabin environment, in: Dissertation, Kansas State University, 2012.
[3] ANSYS Inc., ANSYS Fluent Theory Guide, Release 14.0 [online], 2011, Available from http://www.ansys.com [accessed 8.10.13].
[4] A.J. Baker, S.C. Ericson, J.A. Orzechowski, K.L. Wong, R.P. Garner, Validation for CFD prediction of mass transport in an aircraft passenger cabin, Federal Aviation Administration Oklahoma City OK Civil Aeromedical Inst, 2006.
[5] M.G. Baker, C.N. Thornley, C. Mills, S. Roberts, S. Perera, J. Peters, A. Kelso, I. Barr, N. Wilson, Transmission of pandemic A/H1N1 2009 influenza on passenger aircraft: retrospective cohort study, BMJ (2010) 340.
[6] N.A. Chugier, J.M. Beer, The flow region near the nozzle in double concentric jets, J. Fluids Eng. 86 (4) (1964) 797–804.
[7] S. Corrin, M.S. Uberoi, Spectra and diffusion in a round turbulent jet, J. Propul. Power 16 (4) (1991) 676–688.
[8] S. Dai, H. Sun, W. Liu, Y. Gao, N. Jiang, J. Liu, Experimental study on characteristics of the jet flow from an aircraft gasper, Build. Environ. 93 (2015) 278–284.
[9] Q. Fan, L. Wang, F. Wang, 3D simulation of the plasma jet in thermal plasma spraying, J. Mater. Process. Technol. 166 (2) (2005) 224–229.
[10] N.P. Gao, J.L. Niu, Personalized ventilation for commercial aircraft cabins, J. Aircr. 45 (2) (2008) 508–512.
[12] M. García-Villalba, J. Frohlich, W. Rodi, Identification and analysis of coherent structures in the near field of a turbulent unconfined annular swirling jet using large eddy simulation, Phys. Fluids (1994-present) 18 (5) (2006) 055103.
[13] W.K. George, P.D. Beuther, A. Shabhr, Polynomial calibrations for hot wires in thermally varying flows, Exp. Therm. Fluid Sci. 2 (2) (1989) 230–235.
[14] Y. Guo, N. Jiang, S. Yao, S. Dai, J. Liu, Turbulence measurements of a personal airflow outlet jet in aircraft cabin, Build. Environ. 82 (2014) 608–617.
[15] J.K. Gupta, C.-H. Lin, Q. Chen, Can gaspers provide protection from airborne contaminants to the occupants in an airliner cabin? in: Proceedings of the 12th International Conference on Indoor Air Quality and Climate (Indoor Air 2011), Austin, Texas, 2011.
[15] J.O. Hinze, Turbulence, McGraw-Hill, New York, 1975.
[16] H.J. Hussein, S.P. Capp, W.K. George, Velocity measurements in a high-Reynolds-number momentum-conserving, axisymmetric, turbulent jet Fluid Mech. 258 (1994) 31–75.
[17] International Air Transport Association (IATA), Airlines Expect 31% Rise in Passenger Demand by 2017 [online], 2013, Available from http://www.iata.org/pressroom/pr/pages/2013-12-10-01.aspx [accessed January 2015].
[18] International Civil Aviation Organization (ICAO), Annual Passenger Total Approaches 3 Billion According to ICAO 2012 Air Transport Results [Online], 2012, Available from http://www.icao.int/Newsroom/Pages/annual-passenger-total-approaches-3-billion-according-to-ICAO-2012-air-transport-results.aspx [accessed January 2015].
[19] S.J. Kwon, J.W. Lee, Reynolds number effects on the behavior of a non-buoyant round jet, Exp. Fluids 38 (6) (2005) 801–812.
[20] N. Li, G.M. Leung, J.W. Tang, X. Yang, C. Chao, J.H. Lin, J.W. Lu, P.V. Nielsen, J.L. Niu, H. Qian, A.C. Sleigh, H.J. Su, J. Sundell, T.W. Wong, P.L. Yuen, Role of
ventilation in airborne transmission of infectious agents in the built environment—a multidisciplinary systematic review, Indoor Air 17 (1) (2007) 2–18.

[21] A. Melikov, T. Ivanova, G. Stefanova, Seat headrest–incorporated personalized ventilation: thermal comfort and inhaled air quality, Build. Environ. 47 (2012) 100–108.

[22] F.R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, AIAA J. 32 (8) (1994) 1598–1605.

[23] E.M. Moore, R.L. Shambaugh, D.V. Papavassiliou, Analysis of isothermal annular jets: comparison of computational fluid dynamics and experimental data, J. Appl. Polym. Sci. 94 (3) (2004) 909–922.

[24] S.J. Olsen, H.L. Chang, T.Y.Y. Cheung, A.F.Y. Tang, T.L. Fisk, S.P.L. Ooi, H.W. Kuo, D.D.S. Jiang, K.T. Chen, J. Lando, K.H. Hsu, T.J. Chen, S.F. Dowell, Transmission of the severe acute respiratory syndrome on aircraft, New Engl. J. Med. 349 (25) (2003) 2416–2422.

[25] N.R. Panchapakesan, J.L. Lumley, Turbulence measurements in axisymmetric jets of air and helium, Part I. Air jet, J. Fluid Mech. 246 (1993) 197–223.

[26] P.N. Papanicolaou, E.J. List, Investigations of round vertical turbulent buoyant jets, J. Fluid Mech. 195 (1988) 341–391.

[27] S. Patankar, Numerical Heat Transfer and Fluid Flow, CRC Press, Boca Raton, FL, 1980.

[28] S.B. Pope, Turbulent Flows, Cambridge University Press, Cambridge, 2000.

[29] F.P. Ricou, D.B. Spalding, Measurements of entrainment by axisymmetrical turbulent jets, J. Fluid Mech. 11 (01) (1961) 21–32.

[30] H. Rehag, E. Villenoux, E.J. Hopfinger, Flow regimes of large-velocity-ratio coaxial jets, J. Fluid Mech. 345 (1997) 357–381.

[31] Z. Shi, J. Chen, Q. Chen, On the turbulence models and turbulent Schmidt number in simulating stratified flows, J. Build. Perform. Simul. (2015) 1–15.

[32] A. Wang, Y. Zhang, Y. Sun, X. Wang, Experimental study of ventilation effectiveness and air velocity distribution in an aircraft cabin mockup, Build. Environ. 43 (3) (2008) 337–343.

[33] H.A. Warda, S.Z. Kassab, K.A. Elshorbagy, E.A. Elsaddawy, An experimental investigation of the near-field region of free turbulent round central and annular jets, Flow Meas. Instrum. 10 (1) (1999) 1–14.

[34] D.C. Wilcox, Turbulence modeling for CFD, Turbulence Modeling for CFD, vol. 2, DCW industries, La Canada, CA, 1998.

[35] F.M. White, L. Corfield, Viscous Fluid Flow, vol. 3, MacGraw-Hill, New York, 1991.

[36] R. Wille, H. Fernholz, Report on the first European Mechanics Colloquium, on the Coanda effect, J. Fluid Mech. 23 (04) (1995) 801–819.

[37] C. Wu, N.A. Ahmed, Numerical study of transient aircraft cabin flowfield with unsteady air supply, J. Aircr. 48 (6) (2011) 1994–2001.

[38] B. You, J. Chen, Z. Shi, W. Liu, C.H. Lin, D. Wei, Q. Chen, Experimental and numerical study of airflow distribution in an aircraft cabin mock-up with a gasper on, J. Build. Perform. Simul. (2016) 1–12.

[39] T. Zhang, Q. Chen, Novel air distribution systems for commercial aircraft cabin, Build. Environ. 42 (4) (2007) 1675–1684.

[40] P. Zitek, T. Vyhildal, G. Simeunovic, L. Novakova, J. Čiček, Novel personalized and humidified air supply for airliner passengers, Build. Environ. 45 (11) (2010) 2345–2353.