Building and Environment 65 (2013) 118–131

Contents lists available at SciVerse ScienceDirect

Building and Environment

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

Evaluation of various categories of turbulence models for predicting air distribution in an airliner cabin

Wei Liu [a,c], Jizhou Wen [a], Chao-Hsin Lin [b], Junjie Liu [a], Zhengwei Long [a,*], Qingyan Chen [a,c]

[a] School of Environmental Science and Engineering, Tianjin University, Tianjin 300072, China
[b] Environmental Control Systems, Boeing Commercial Airplanes, Everett, WA 98203, USA
[c] School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, USA

A R T I C L E I N F O

Article history:
Received 17 November 2012
Received in revised form
20 March 2013
Accepted 21 March 2013

Keywords:
Airliner cabin
Flow field
Turbulence model
CFD simulation
Validation

A B S T R A C T

Flow fields in commercial airliner cabins are crucial for creating a thermally comfortable and healthy cabin environment. The study of flow fields in cabins could be achieved by numerically solving Navier–Stokes equations with a suitable turbulence model. This investigation evaluated three turbulence models in different categories: the RNG k-ε model, LES, and DES for the steady-state flow in the first-class cabin of a functional MD-82 commercial airliner. The measured flow fields under unoccupied and fully-occupied conditions in the first-class cabin were used for validating the turbulence models. The flow in the unoccupied cabin was isothermally forced convection created by air jets from the diffusers, while the flow in the fully-occupied cabin was mixed convection driven by both the jets and thermal plumes from the thermal manikins used to simulate passengers. This study found that the RNG k-ε model gave acceptable accuracy in predicting the airflow in the unoccupied cabin where the flow was simple, but not for the complicated flow in the fully-occupied cabin. The DES gave acceptable flow fields for both cabins. The LES performed the best and the results agreed well with the experimental data. Comparing the measured flow fields in the two cabin conditions, this study found that the thermal plumes from the heated manikins had a significant influence on the flow fields, but little influence on the turbulence.

1. Introduction

Nowadays, more and more people are traveling by air, and the flying public is becoming more concerned about the cabin environment. Since the cabin environment could be too hot or too cold and air contaminants such as ozone, carbon monoxide, various organic chemicals, and biological agents could exist in cabins [1], the cabin environment needs further improvement. In commercial airliner cabins, the air distribution is used to regulate the air temperature and air velocity to create a thermally comfortable environment and to provide adequate ventilation to reduce gaseous and particulate contaminants for maintaining a safe and healthy environment. Therefore, to improve the cabin environment, the air distribution should be carefully studied.

To study the air distribution in an airliner cabin, Computational Fluid Dynamics (CFD) simulations have become a practical approach. Since Nielsen [2], who was the first one to apply CFD to room airflow prediction, applications of CFD for airflow predictions in enclosed spaces have become popular [3]. Compared with experimental study, CFD simulation is less expensive and more efficient. However, since the turbulence models in CFD used approximations, the simulation results may contain uncertainties. Therefore, the CFD results need to be validated by corresponding experimental data before CFD can be used for further studies.

We reviewed CFD application for airliner cabins [4] and found that the CFD models mainly used were Reynolds Averaged Navier–Stokes equation (RANS) models and Large Eddy Simulation (LES) [5,6]. For example, Lin et al. [7] studied airflow in a section of a twin-aisle aircraft cabin with the Re-Normalization Group (RNG) k-ε model [8]. The simulation substantially under-predicted the turbulence intensity, especially in and around the breathing zone. Zhang et al. [9] also used the RNG k-ε model to study the airflow in a twin-aisle, economy-class section of an airliner cabin. Poor agreement was found between the computed results and the experimental data, and they concluded that the deviation was due to the difficulties in measuring accurate flow boundary conditions from the air supply diffusers. Singh et al. [10] used the RNG k-ε model to simulate the airflow in a cabin mockup without occupants and with occupants. The inlet air velocity in their study was uniform. However, due to the lack of reliable experimental data,

* Corresponding author. Tel.: +86 22 2740 9500.
E-mail address: longzw@tju.edu.cn (Z. Long).

0360-1323/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.buildenv.2013.03.018

Table 1
Heat distribution of the thermal manikin.

Body segments	Heat generated by thermal manikin (W)	Percentage (%)
Head and neck	5.7	7.6
Trunk	20.1	26.8
Left arm	8.2	10.9
Right arm	8.2	10.9
Left leg	16.4	21.9
Right leg	16.4	21.9
Total	75.0	100.0

their study could not make quantitative comparisons between the simulated results and experimental data. Lin et al. [7] conducted a LES to obtain the turbulent flow in a generic cabin mockup. The turbulence level predicted was in fairly good agreement with the experimental data. However, these studies did not compare the performance on prediction of air flow field of different categories of turbulence models. Moreover, since numerous turbulence models have been developed in the past decades, many of them may be used in predicting airflows and turbulence in enclosed environments. Besides the RANS models and LES, Detached Eddy Simulation (DES) [11] has been widely used to predict airflow in indoor environments. Roy et al. [12] compared DES and RANS and Jouvray et al. [13,14] compared DES, LES, and RANS and found that DES appeared to be a promising model, giving good agreement on velocity and Reynolds stresses. It is indispensible to evaluate the generality and robustness of the DES for airflow prediction in aircraft cabins.

The studies above included some experimental data for validating the CFD results. Through a more extended literature search, this investigation found that most of the experimental measurements were conducted in cabin mockups with only a few rows of seats or no seats at all [9,15–17]. The cabin mockups were different from real aircraft cabins, especially in the air supply system and internal furnishings. The influence of these differences on the airflow is mostly unknown. Mazumdar et al. [18] concluded that the flow and contaminant transport obtained using the small-scale water cabin model may not be the same as in a full-scale aircraft cabin because it is difficult to achieve flow similarity. Therefore, it is necessary to use a real aircraft cabin to obtain reliable and high quality experimental data. Moreover, most of the former experimental studies [19–24] applied optical measuring techniques such as Particle Tracking Velocimetry (PTV), Particle Streak Velocimetry (PSV), and Particle Image Velocimetry (PIV). These optical anemometries could only measure in the spaces where a laser light sheet can penetrate. When they were used in an aircraft cabin, passengers (typically manikins) and seats would block the laser light sheet, so no airflow could be measured in the lower part of the cabin. Zhang et al. [9] applied Ultrasonic Anemometers (UA), which can provide three-dimensional, point-by-point airflow information, to measure the flow field in a cabin mockup. The measured data had low resolution because the UA sensor was very expensive, so they used only two UAs in their experiment. Our previous paper [25] reported our effort to obtain accurate cabin geometry, boundary conditions of diffusers, and high-resolution flow fields with nine UAs in an unoccupied, first-class cabin of a functional MD-82 commercial airliner. The experimental data in unoccupied conditions could be used to evaluate the turbulence models, but it is not sufficient since the occupied conditions are more important for passengers and crew. Therefore, it is necessary to obtain a high-resolution flow field in occupied condition for validating turbulence models.

Therefore, this study measured further the boundary conditions and flow fields in fully-occupied conditions with heated manikins in the first-class cabin of an MD-82 commercial airliner. The measured data together with that of the unoccupied cabin [25] was used here to compare it with the corresponding numerical simulation results by three turbulence models in different categories: RNG k-ε model, LES, and DES. This effort would be able to identify a suitable model for further studies of airflow in airliner cabins and provide engineers a good sense on the model performance and computing costs.

2. Research method

The main objective of our study was to evaluate different CFD turbulence models by using high quality flow data measured in a functional MD-82 airliner cabin. Since the experimental data from our previous experiment [25] was insufficient for the validation, additional measurements for the cabin under fully-occupied conditions were conducted. The following describes the method used in the experimental measurements and CFD modeling.

2.1. Experimental measurements

This investigation used the same MD-82 aircraft [25] for additional experimental measurements. Readers may refer to that

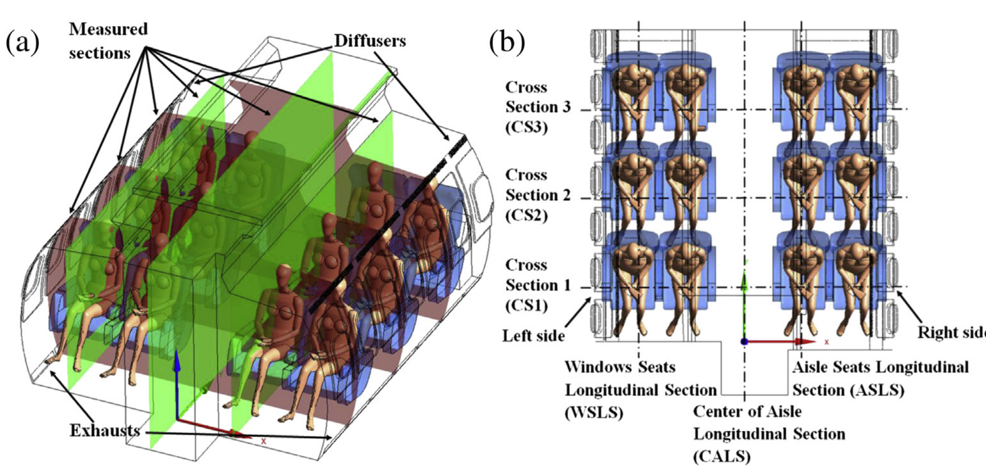

Fig. 1. Schematic of the fully-occupied, first-class cabin and measured sections: (a) perspective view and (b) plane view.