Available online at www.sciencedirect.com
ScienceDirect
Aerospace Science and Technology 10 (2006) 563–573
Aerospace Science and Technology
www.elsevier.com/locate/aescte

Experimental and numerical simulations of idealized aircraft cabin flows

Gero Günther [a,*], Johannes Bosbach [a], Julien Pennecot [a], Claus Wagner [a], Thomas Lerche [b], Ingo Gores [b]

[a] *German Aerospace Center (DLR), Institute of Aerodynamics and Flow Technology, Bunsenstr. 10, 37073 Göttingen, Germany*
[b] *Airbus Deutschland GmbH, Environmental Control Systems, Kreetslag 10, 21129 Hamburg, Germany*

Received 7 July 2004; received in revised form 24 January 2006; accepted 20 February 2006

Available online 4 August 2006

Abstract

Two different generic cabin configurations, deduced from the passenger cabin and the sleeping bunk of a modern mega liner, have been investigated numerically by means of Reynolds-averaged Navier–Stokes (RANS) computations and experimentally by particle image velocimetry (PIV) as well as thermography. In order to study the flow of fresh air in the vicinity of the luggage compartment, a cabin mock-up which represents the region above the passengers to the luggage compartment has been built. Additionally, a single person sleeping bunk of the crew rest compartments was considered. Comparisons between computations and measurements show a good agreement. They further indicate that higher order low Reynolds number turbulence models are suited best to predict the complex 3D-cabin airflow with separation.

Keywords: Digital human; CFD; PIV; Indoor flow

1. Introduction

Confined mixed convection is of essential importance for a variety of practical applications like e.g. air-supply in offices or residential buildings, the cooling of micro-electronic devices, and the air conditioning of vehicle and aircraft cabins. The numerical prediction of such flows by means of computational fluid dynamics (CFD) allows for an optimization of the cabin design with respect to the thermal comfort, while excessive prototyping can be prevented. One key issue during this task is the selection of the appropriate turbulence model. Most of the available models to account for turbulence in Reynolds-averaged Navier–Stokes simulations (RANS) have been developed and validated for the case of simple shear flows. Systematic comparative studies performed so far reveal large deviations between turbulence models when it comes to quantitative description of mixed convection [1,2]. It is further well known, that the reliable prediction of flow reversal due to premature separation of a cooling wall jet is a major problem for many turbulence models [3]. In [2] it was shown, that the incautious use of turbulence models in computations for cooling applications might prevent even the qualitative prediction of the flow direction. Therefore, significant research is performed in order to improve the predictive capabilities of CFD methods with respect to airflow and temperature distribution inside the aircraft cabin. Mizuno and Warfield [4] for example modified a CFD program to compute thermal effects and contaminants dispersion. They also investigated the influence of obstructions such as passenger seats on the airflow in aircraft cabins. 3D-CFD computations in an aircraft fuselage section were also performed by Singh et al. [5] using a commercial software package. On the experimental side particle image velocimetry of the airflow inside a section of a narrow body Boeing 737 aircraft cabin have been reported by Mo et al. [18]. More recently Lin et al. [16] and [17] performed RANS of the airflow in a passenger cabin of a Boeing 767-300. They also conducted Large-Eddy Simulations to predict the airflow and pathogen transport in a generic cabin model. Their results compare favourable with the presented test data. A similar environment, i.e. a model of the 767 aircraft cabin was chosen by Sun et al. [19] and Zhang et al. [20] for their ex-

* Corresponding author. Tel.: +49 (0) 551 709 2261; fax: +49 (0) 551 2404.
E-mail addresses: gero.guenther@dlr.de (G. Günther), johannes.bosbach@dlr.de (J. Bosbach), julien.pennecot@dlr.de (J. Pennecot), claus.wagner@dlr.de (C. Wagner), thomas.lerche@airbus.com (T. Lerche), ingo.gores@airbus.com (I. Gores).

1270-9638/$ – see front matter
doi:10.1016/j.ast.2006.02.003

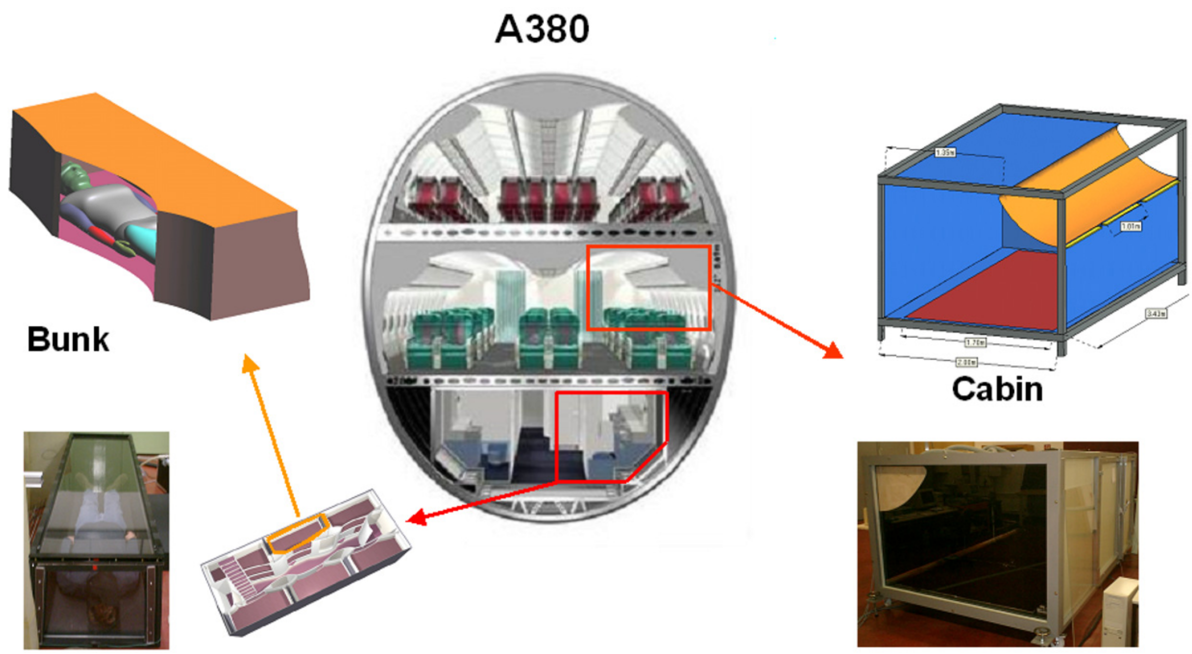

Fig. 1. Placement of bunks and cabins in a modern mega liner and investigated cabin configurations.

perimental investigations using the particle streak velocimetry (VPSV) technique.

The intention of our study is to investigate the reliability of industrially conducted cabin aerodynamic and thermal comfort predictions and to identify necessary improvements. Further, we want to improve the understanding of the physical mechanisms in the airflow of different cabin configurations and their effect on the thermal comfort (see [14] and [15]). To reach this goal we conducted numerical simulations of flow and heat transport in a generic aircraft passenger cabin and a sleeping bunk and compared the obtained results to those of measurements using particle image velocimetry (PIV) and thermography. In Fig. 1 a centre cross section of a modern megaliner with passenger cabins at two decks and crew rest compartments at the lower deck is presented. On the left hand side a CAD representation of an idealized sleeping bunk (left, top) and it's mock up (left, bottom) is shown, while the right hand reflects the idealized passenger cabin. The airflow and temperature distribution in the latter two configurations has been studied.

2. Idealized sleeping bunk

2.1. Configuration

The crew rest compartments consist of up to 12 bunks, out of which a single person sleeping bunk was considered. For CFD the curved designer geometry was used, while the experiment was conducted in a geometrically simplified box of 0.60 m to 0.68 m height, 1.98 m length and 0.48 m to 0.55 m width. The size of the inflow slot was 1.59 m × 15 mm and that of the outflow slot 1 m × 80 mm. The air entered the mock-up horizontally as indicated in Fig. 2.

2.2. Numerical simulation

The RANS computations were performed with the commercial CFD code StarCD using algebraic multi grid acceleration on a solution adapted hybrid grid of tetras and prisms containing 381614 points and 11954505 volume elements, as presented in Figs. 2 and 3. The smallest wall distance was chosen such that a dimensionless wall distance $y+ < 1$ was realized. Be-

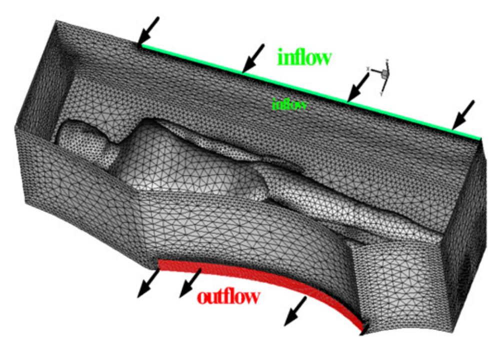

Fig. 2. Computational surface mesh of bunk.

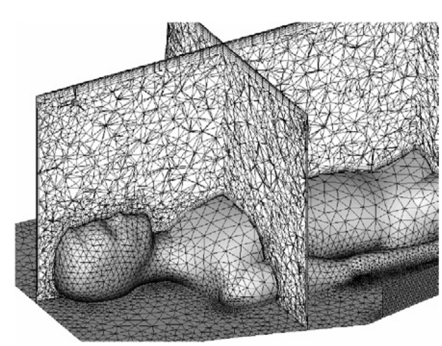

Fig. 3. Computational hybrid mesh of bunk in detail.