Thermal and Fluid-dynamical Optimisation of Passengers Comfort in a Touring Bus Cabin

Petrone G. (*), Fichera G. (**), Scionti M. (*)

(*) Bus-Engineering S.r.l.
Viale Africa, 44 – 95129 Catania, ITALY
E-mail: info@bus-engineering.it
Web-site: www.bus-engineering.it

(**) Department of Industrial and Mechanical Engineering, University of Catania
Viale Andrea Doria 6 – 95125 Catania, ITALY

Abstract: This study deals with a thermal and fluid-dynamical investigation on passengers comfort in a touring bus cabin. Ventilation and air quality in indoor environment is an issue of very high interest, determining comfortable conditions for occupants and no-contaminated local atmosphere. This topic plays a crucial role in transportation means, where occupants stand inside a cabin for several hours having no large opportunities of moving. In this framework, COMSOL Multiphysics® is exploited as a powerful design and optimisation tool for air distribution devices.

Keywords: HVAC, comfort, cabin, fluid-dynamics, heat transfer.

1. Introduction

Air quality and thermal comfort are very important factors in the design of indoor ambient. Innovations in air-conditioning and other forms of cooling or ventilation can be viewed as technological solutions improving environmental conditions that are beneficial for human health, comfort and productivity [1]. Providing thermal comfort and saving energy are two main goals of heating, ventilation and air conditioning (HVAC) systems. Moreover, the effectiveness of an HVAC system is to provide comfort in any condition. Comfort is vague. ASHRAE defines comfort as ‘‘condition of mind that expresses satisfaction with the thermal environment” [2]. Thermal comfort is influenced by many variables such as, metabolic rate, clothing, air velocity, air temperature, air temperature stratification, radiant temperature, radiant temperature asymmetry, relative humidity, and turbulence intensity in the occupied zone [3]. For the same activity level, clothing type, and indoor geometry, thermal comfort is related to air velocity and temperature, temperature stratification, relative humidity, and turbulence intensity. With different diffusers and different locations of supply inlets and return outlets, the distribution of the thermal comfort parameters is different. Hence, it is necessary to understand quantitatively how different inlet and outlet layouts affect local thermal comfort. Thermal comfort plays a crucial role in public transports, much more that in building applications. In fact passengers, standing inside a cabin for several hours, do not have large opportunities of moving. This is the reason why since several years many studies have been devoted to the air distribution system for passengers cabin. Two main approaches are available for analysing air distribution in a cabin: experimental measurements (particle image velocimetry, ultrasonic anemometers, volumetric particle streak velocimetry) and computer simulations (CFD) [4]. Experimental measurements, which are often considered reliable, can be very difficult and expensive for studying air distributions. Most of the experimental studies did not consider realistic thermo-fluid conditions or geometry. The spatial resolution of air and contaminant distributions is generally low. CFD seems like a good alternative. Several bibliographic studies concern ventilation and thermal comfort conditions in aircraft, car, truck and vessel cabin. However, to the best knowledge of authors, no many studies have focused on air distribution system in touring bus cabin. The air distribution system is an important component of the environmental control system since it is used to distribute conditioned air properly to the cabin, providing a healthy and comfortable cabin environment. Since a cabin has a higher occupant density, more complex geometry and a lower outside air supply rate per person as compared to buildings, it is very challenging to design a comfortable and healthy cabin environment for commercial purposes also [5]. Currently, mixing air distribution systems are used to distribute air in a cabin. Conditioned air is supplied at the ceiling level with a high velocity and then mixes with the air in the cabin. The air temperature in the cabin should be rather uniform and contaminants in the cabin have to be diluted. The present communication deals with an investigation on the most appropriate layout for the ventilation air inlets in a touring bus cabin in order to optimise thermal and fluid-dynamical conditions for passengers.

2. Modelling

The geometry of the system consists in a portion of the bus cabin, containing two rows of seats (Figure 1). Exploiting the longitudinal symmetry of the cabin, just one half of it has been reproduced.

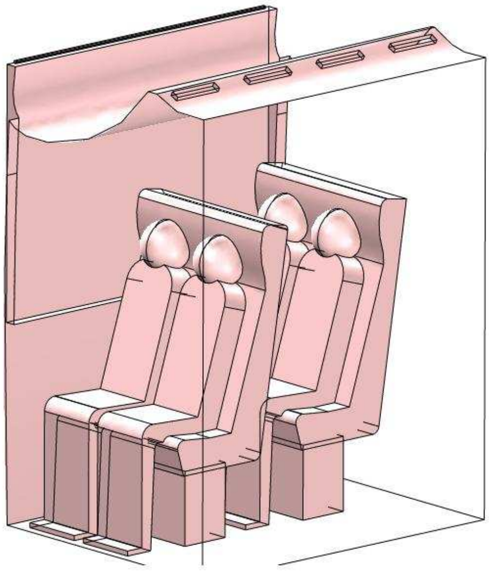

Figure 1. Outline of the studied system: portion of the touring bus cabin.

Two inlet sections for fresh air are considered: the first one, located at the roof and representative of a ceiling mixing air distribution, is made of rectangular slots positioned in the horizontal channel developing along the cabin aisle; the second one, representative of a displacement ventilation system, is located in correspondence of the lateral side of each seat. Contrary to building applications, where furnishing and people position is difficult to predict, so that simulations are very often carried-out for empty rooms, in transport applications the passengers' position is well-known, so that some geometrical elements have been used in this model in order to simulate the human presence. In order to solve dynamical and thermal field in the considered air volume, momentum and energy equations have been solved adopting a k-ε scheme for turbulence modelling. The continuous equations governing motion distribution for steady state are listed below:

$$\rho u \cdot \nabla u = \nabla \cdot \left[-pI + (\eta + \eta_T)\left(\nabla u + (\nabla u)\right)^T \right] + F$$

$$\nabla \cdot u = 0$$

$$\rho u \cdot \nabla k = \nabla \left[\left(\eta + \frac{\eta_T}{\sigma_k} \right) \nabla k \right] + \eta_T P(u) - \rho \varepsilon$$

$$\rho u \cdot \nabla \varepsilon = \nabla \left[\left(\eta + \frac{\eta_T}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{C_{\varepsilon 1} \varepsilon \eta_T P(u)}{k} - \frac{C_{\varepsilon 2} \rho \varepsilon^2}{k}$$

where

$$P(u) = \nabla u : \left(\nabla u + (\nabla u)^T \right) \text{ and } \eta_T = \frac{\rho C_\mu k^2}{\varepsilon}$$

Temperature field has been computed by solving energy conservation equation, reported below:

$$\nabla \cdot (-k \nabla T) = \rho C_P u \cdot \nabla T$$

The above partial differential equations are numerically integrated with the following fluid-dynamical and thermal boundary conditions:

- No-slip condition at solid walls;
- Chosen velocity (u_{in}) for incoming air through the inlet slots of the air distribution system;
- Pressure imposed in correspondence of the recovery grid for air;

- Symmetry conditions for the longitudinal and transversal confinement of the air volume;

- Chosen temperature for incoming air (T_{air});
- Chosen heat flux for external confinement of the air volume (both window and lateral wall);
- Adiabatic condition for symmetry sections, floor and roof of the cabin.

Physical properties adopted for air are reported in Table 1.

Parameter	Unit
$M = 0.0288$	$[kg\ mol^{-1}]$
$\mathrm{R} = 8.134$	$[J\ mole^{-1} kg^{-1}]$
$p_a = 101325$	$[Pa]$
$\rho \equiv p_0 M / (RT)$	$[kg\ m^{-3}]$
$\eta \equiv 6,0 \cdot 10^{-6} + 4,0 \cdot 10^{-8} T$	$[Pa\ s]$
$k \equiv \exp(-3,723 + 0,865 \log T)$	$[W\, m^{-1} K^{-1}]$
$C_P = 1100$	$[J\ kg^{-1} K^{-1}]$

Table 1: Physical properties assumed for fluid.

Differential equations have been spatially discretized on a no-uniform and no-structured computational grid. The used grid for carrying-out computations is represented in Figure 2.

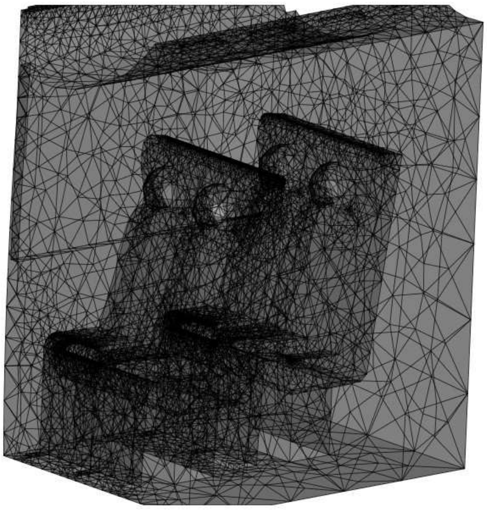

Figure 2. Numerical grid adopted for computations.

That adopted computational grid determinates 360.000 degrees of freedom for the system. Steady solution of governing equations has been reached by applying a Newton-Raphson method. Algebraic systems coming from differential operators discretization have been solved by using a direct Unsymmetrical Multi-Frontal method based on the LU decomposition. Computations were carried-out on a 64 bit calculator disposing of 16 GB of RAM.

3. Results

Results have been carried-out for the two inlet layouts of the conditioning air previously introduced. Firstly a pure ceiling system for air inlet is studied. This scheme represents a standard air distribution system, based on the mixing air principle for indoor conditioning and ventilation. Then a second scheme has been considered: inlet sections for conditioning air are also arranged in correspondence of the lateral side of each passenger seat. This layout is representative of a combination between the standard ceiling system and a displacement air distribution system. Both configurations are studied for winter conditions, referring to an external temperature of 5°C. Figure 2 presents isosurfaces of velocity for the first analysed configuration. Ventilation air is provided by four rectangular slots located at the roof in correspondence of the aisle of the bus cabin.

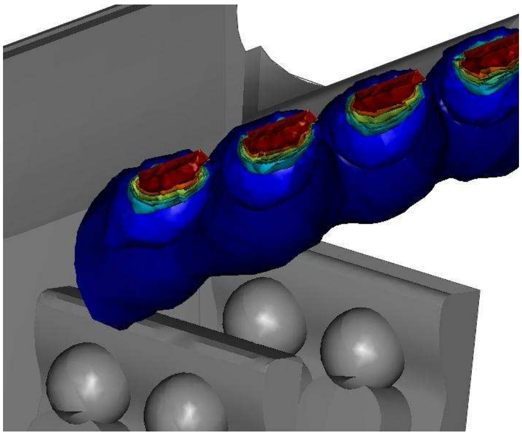

Figure 3. Isovalue surfaces of velocity close to the inlet slots for the air inlet (ceiling system).

The interest was firstly focused on verifying passengers comfort referring to the motion field of ventilation air. In fact it is recognized that values of air velocity exceeding 0.8 m/s could be responsible of discomfort condition for human occupants of a ventilated ambient.

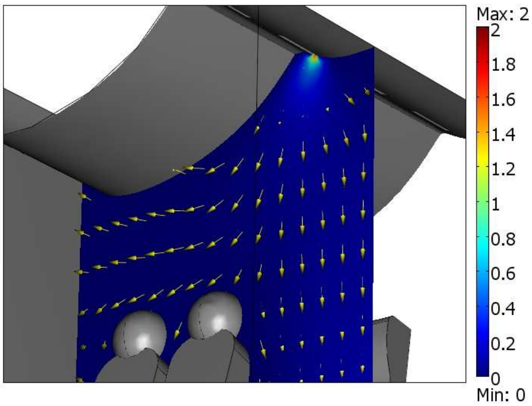

Figure 4. Motion field of air in a transversal vertical section of the cabin.

Figure 4 shows distribution of air velocity and velocity vectors plotted in a transversal section of the cabin chosen as indicative for verifying comfort of passenger (the surface crosses passengers head). From analysis of Figure 4 it can be verified as the ceiling system for air distribution does not involve in discomfort for passengers. The inlet velocity for incoming air was chosen at value 2.5 m/s, that well fits with reasonable values usually adopted for this kind of ventilation system .

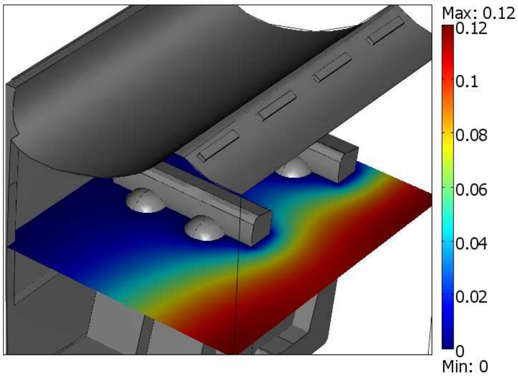

Figure 5. Velocity distribution in a horizontal transversal section of the cabin.

The above discussed consideration is also confirmed by results reported in Figure 5. That Figure reports the velocity field in a horizontal transversal section of the cabin. Maximum value of air velocity is reached in a zone not occupied, such as the cabin aisle. From a fluid-dynamical point of view, the second element that can determinate disagreeable condition for occupants is the turbulence level. In Figure 6 streamlines of the vorticity function for the obtained air flow have been reported. From analysis of that finding it is possible to deduce as high turbulence levels only occur in the top portion of the cabin space, so that passengers do not result affected by this potential cause of discomfort.

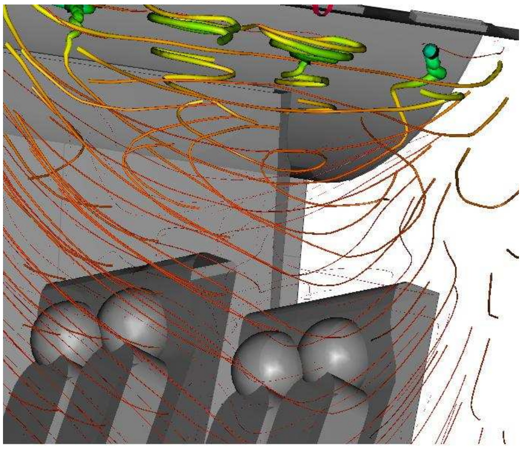

Figure 6. Streamlines of the vorticity function.

Let now examine the ceiling air distribution system from a thermal point of view. Heating air is introduced from the lateral side of the passengers seat close to the cabin aisle. On the other hand, the higher level of heat dissipation occurs on the opposite side, close to lateral wall of the bus cabin. This wall obviously confines with the external ambient. Moreover, it is partially made of the window glass that, as known, is characterised by a high thermal conductivity. These assumptions let guess to results obtained from thermal analysis of this inlet layout for heating air. In Figure 7 temperature distribution has been plotted on contours of solid elements of the numerical model. Results reported in that figure well show the ineffectiveness of the ceiling air distribution system from a thermal point of view. As previously mentioned, thermal comfort highly depends on uniform distribution of the air temperature surrounding the human body. From

Figure 7 is clearly observable as passengers seated close to the window are submitted to strong temperature gradient in horizontal direction.

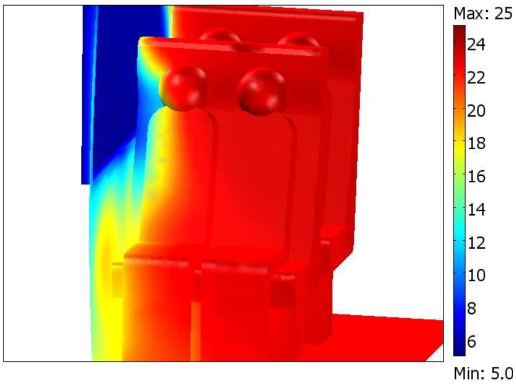

Figure 7. Temperature field at solid-fluid interfaces.

Thermal distribution is also reported in Figure 8, where the difference in temperature value between the right and the left side of the passenger body seated close to the window is much more observable.

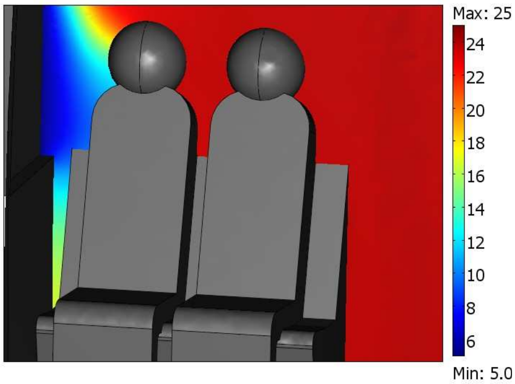

Figure 8. Thermal field in a transversal section of the cabin.

Certainly the obtained thermal field does not assure thermal comfort for passengers. Because of that reason it was simulated a second scheme for heating air incoming. As yet mentioned, this second outline of the numerical model takes its inspiration from the displacement air conditioning systems. This system is based on the opportunity of providing conditioning air from the bottom side of the ambient and very close to the occupants. Velocity values for inlet air have to be chosen very low in order to assure agreeable conditions from a dynamical point of view. For heating application the buoyancy effect generated by metabolic heat dissipation close to human bodies is also exploited for propulsion of air incoming from the bottom. This involves in formation of thermal plumes in correspondence of human occupants during sedentary activities. In order to test this layout of inlet section location, heating air was considered to enter the computational domain trough out the lateral bottom side of each passengers seat. Figure 9 presents the obtained results.

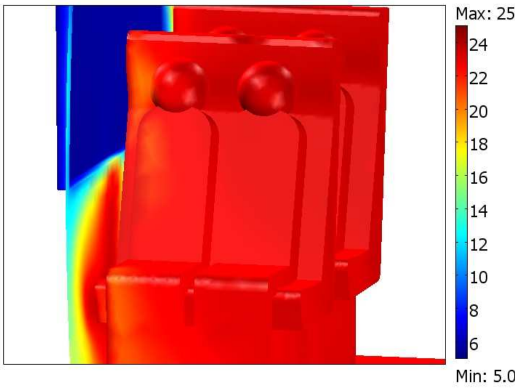

Figure 11. Thermal field (summer conditions).

As observable thermal field at solid-fluid interfaces of the numerical model show an almost uniform distribution of temperature in correspondence of the human occupants. That clearly means comfort conditions for passengers.

4. Conclusions

This study allowed investigating the conditioning air distribution in a touring-bus cabin. Special attention has been paid to optimise air terminal layout and configuration of inlet section for ventilation air in order to research the best operational scheme assuring high level of comfort for passengers. Results have shown as combining a standard ceiling system with a displacement scheme for heating air inlet represents the best compromise guaranteeing thermal comfort for passengers. This research study also underlines the effectiveness of the applied numerical tool in simulating thermal and fluid-dynamical systems of technological interest.

5. Acknowledgement

Authors are grateful to Eng. Francesco Cambria for his precious support in CAD generation.

6. References

1. Kavgic M., Mumovic D., Stevanovic Z., Young A., Analysis of thermal comfort and indoor air quality in a mechanically ventilated theatre, *Energy and Building*, Vol. 40 (2008), pp. 1334-1343.

2. Farzaneh Y., Tootoonchi A. A., Controlling automobile thermal comfort using optimized fuzzy controller, Applied Thermal Energy, Vol. 28 (2008), 1906-1907.

3. Lin Z., Chow T.T., Tsang C.F., Fong K.F., Chan L.S., CFD study on effect of the air supply location on the performance of the displacement ventilation system, *Energy and Building*, Vol. 40 (2005), pp. 1051-1067.

4. Aijun W., Yuanhui Z., Yigang S., Xinlei W., Experimental study of ventilation effectiveness and air velocity distribution in an aircraft cabin mockup, *Building and Environment*, Vol. 43 (2008), pp. 337-343.

5. Tengfei Z., Qingyan C., Novel air distribution systems for commercial aircraft cabins, *Building and Environment*, Vol. 42 (2007), pp. 1675-1684.