#6333

AIRFLOW IN THE PASSENGER COMPARTMENT OF A BUS

M.C. Silva, D.X. Viegas and L.A. Oliveira
Grupo de Mecânica dos Fluidos
Universidade de Coimbra
Coimbra, Portugal

SUMMARY

In order to evaluate the comfort conditions in the passenger compartment of an intercity bus, a study of the airflow properties was carried out. Both experimental and numerical simulations of the flow inside the bus were considered.

A full size section of the bus 2 m long was used for the experiments. Forced air at ambient temperature was injected in the compartment.

The properties of the mean and turbulent flow for the isothermal case were measured with a conical hot film probe, in the volume between two rows of seats. Particular attention was given to the flow created by the jets placed over the passenger heads.

Numerical predictions based on a finite volume type discretization of the differential equations for mass, momentum and energy conservation were performed for a two-dimensional configuration. The theoretical study, where turbulence effects were modelled through a k-ε type approach, included a systematic analysis of the influence of the relevant parameters on the flow field, thus providing important guidelines for the laboratory work.

AIRFLOW IN THE PASSENGER COMPARTMENT OF A BUS

M.C. Silva, D.X. Viegas and L.A. Oliveira
Grupo de Mecânica dos Fluidos
Universidade de Coimbra
Coimbra, Portugal

INTRODUCTION

Thermal confort in luxury passenger buses is an important requirement for both users and designers of high quality vehicles. Comfort conditions for indoor situations are well established by heating and ventilation standards, namely ISO 7730, ASHRAE 62-89, and have been the object of various studies, namely by Fanger [1], Olesen [2], Olesen & Rosendahl [3].

The concept of equivalent temperature that integrates the influences of air temperature velocity and mean radiant temperature is widely used in practice, in conjunction with the values of relative humidity and of two numerical indexes related to clothing and to physical activity, in order to evaluate the thermal confort. Most of the times, its evaluation is done with an unique sensor that simulates the thermal behaviour of a human body and integrates the influences of all those variables. However, if a good knowledge of the air distribution is required as a step to the development of a new design or in the process of improving an existing one, the velocity field in the studied domain should be well known. Some examples of this kind of studies can be found in Temming & Hucho [4] and in Klemp et al. [5].

In this paper the isothermal airflow produced by the ventilating system of a passenger bus is studied both experimentally in a full scale laboratory test, and also numerically, using a two-dimensional simulation of the turbulent flow in a transverse cross section of the bus.

EXPERIMENTAL SETUP

The measurements were carried out in a full scale sectional module of a passenger compartment of an intercity "Salvador Caetano Delta" bus. The module is 2m long, with two rows of seats and was built with all the constructive details of a real bus (fig. 1).

The air was introduced in the compartment through the nozzles placed in the ducts over the seats. Two centrifugal fans of the type commonly used in the car industry, driven by DC motors, were used to blow the air into the ducts.

The power supply of the DC motors was regulated in order to obtain an air velocity at the outlet of the jets of 10m/s which was the order of the mean value measured in the real bus.

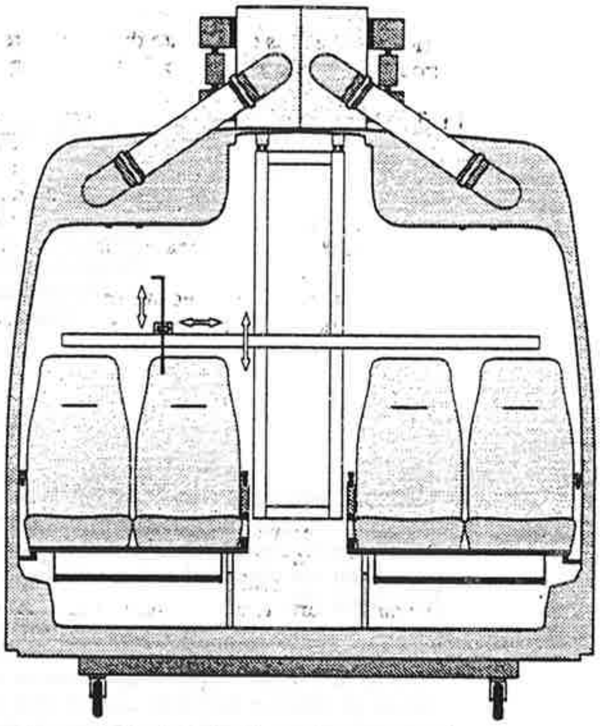

Fig. 1. Bus module scheme.

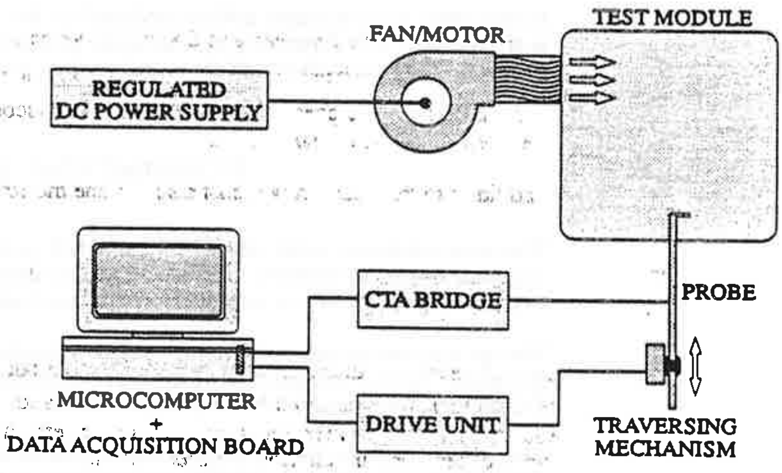

Fig. 2. Block diagram of the experimental setup.

Particular care was taken with the duct system in order to have the same conditions in the various jets and to asume the flow symmetry in the passenger compartment.

In fig. 2 the measuring system is schematically represented. The measurements were performed with a conical hot film sensor (TSI 1231) connected to a constant temperature anemometer bridge Dantec 55M10. The signal of the anemometer was digitized with a Metrabyte DAS 16G1 interface and acquired by the microcomputer. The sensor was calibrated, with the standard proceeding, before each test, using the calibration equipment Dantec 55D90. The linearization of the signal was done by the data acquisition software.

A three axis traversing mechanism was used inside the model. Two axes were manually positioned and the third one was equipped with a Dantec 55E40 motorized traversing unit, controlled by the computer.

NUMERICAL FORMULATION

Even though temperature distributions for non-isothermal situations were also calculated to provide guidelines for future laboratory work, the present paper only concerns the isothermal flow field analysis. For this case the governing transport equations of the two-dimensional turbulent flow were applied in their cartesian co-ordinate form.

Governing equations

The time-averaged transport equations for the conservation of momentum are in compact tensor notation, as follows:

$$\frac{\partial}{\partial x_j}\left[\bar{\rho}\bar{u}_j\bar{u}_i-\mu\left(\frac{\partial\bar{u}_i}{\partial x_j}+\frac{\partial\bar{u}_j}{\partial x_i}-\frac{2}{3}\frac{\partial\bar{u}_k}{\partial x_k}\delta_{ij}\right)\right]+\frac{\partial\bar{p}}{\partial x_i}+\left(\bar{\rho}-\rho_{ref}\right)g_i+\frac{\partial}{\partial x_j}\left(\overline{\rho u'_i u'_j}\right)=0 \quad (1)$$

where $\bar{u}_i$ and u'_i are the mean and fluctuating velocity components in the x_i direction, ρ is density and ρ_{ref} a reference value, g_i is the magnitude of the gravitational acceleration in the x_i direction, $\bar{p}$ is pressure, μ is the laminar viscosity and δ_{ij} is the Kronecker delta. The overbar denotes mean values.

In addition to this relation we must also include the equation of mass continuity:

$$\frac{\partial}{\partial x_i}\left(\bar{\rho}\bar{u}_i\right)=0 \quad (2)$$

In the present isothermal conditions ρ may be taken as a constant equal to ρ_{ref}, thus reducing mass conservation to $\partial\bar{u}_i / \partial x_i = 0$ and allowing further simplification of equation (1). This latter equation is however left unchanged as it is used in the present form for temperature distribution predictions.

Turbulence model

The "two-equation" turbulence model of Launder and Spalding [6] in which equations for the kinetic energy of turbulence, K, and its dissipation rate, ε, are solved, was used for the present calculations. The correlations in equation (1) are expressed, in analogy with the laminar flow, as:

$$-\bar{\rho}\overline{u'_i u'_j} = \mu_t\left(\frac{\partial \bar{u}_i}{\partial x_j}+\frac{\partial \bar{u}_j}{\partial x_i}\right)-\frac{2}{3}\bar{\rho}k\delta_{ij} \tag{3}$$

where μ_t is a "turbulent" viscosity that may be related to k and ε by dimensional arguments:

$$\mu_t = c_\mu \rho k^2 / \varepsilon \tag{4}$$

where c_μ is a constant of the model. The turbulent exchange coefficient $\Gamma_{\phi,t}$ for any variable ϕ may be expressed as:

$$\Gamma_{\phi,t} = \mu_t / \sigma_{\phi,t} \tag{5}$$

where $\sigma_{\phi,t}$ is a Prandtl number of order unity.

The transport equations for k and ε that were solved for the present purposes are:

$$\frac{\partial}{\partial x_j}\left(\bar{\rho}\bar{u}_j k-\Gamma_{k,t}\frac{\partial k}{\partial x_j}\right)-\mu_t\frac{\partial \bar{u}_i}{\partial x_j}\left(\frac{\partial \bar{u}_i}{\partial x_j}+\frac{\partial \bar{u}_j}{\partial x_i}\right)+\bar{\rho}\varepsilon-S_k=0 \tag{6}$$

and

$$\frac{\partial}{\partial x_j}\left(\bar{\rho}\bar{u}_j\varepsilon-\Gamma_{\varepsilon,t}\frac{\partial \varepsilon}{\partial x_j}\right)-C_1\frac{\varepsilon}{k}\mu_t\frac{\partial \bar{u}_i}{\partial x_j}\left(\frac{\partial \bar{u}_i}{\partial x_j}+\frac{\partial \bar{u}_j}{\partial x_i}\right)+C_2\bar{\rho}\frac{\varepsilon^2}{k}-S_\varepsilon=0 \tag{7}$$

where C_1 and C_2 are further constants and $\Gamma_{\phi,t}$ and $\Gamma_{\phi,t}$ are determined via equation (5). The standard constants of the model were used [6].

Numerical approach

The partial differential equations were discretized using the finite volume method [7] The approximation of the convective fluxes applied at each control volume face was performed using the hybrid central/upwind difference scheme [7].

The velocities and pressures were calculated by the SIMPLEC algorithm described in [8] The solution of the individual equation sets was obtained by a Gauss-Seidel line-by-line iteration, where horizontal sweeps were alternated with vertical ones.

Solid wall and outflow boundaries were represented by no-slip and zero-gradient conditions, respectively.

The domain considered was mapped by a non-uniform mesh having small size cells in regions with pronounced gradients. A grid of 62x62 nodes was used, though negligible differences were observed between the solutions obtained with this grid and one of 52x52. Convergence was attained when the normalised residuals for the two momentum equations and mass conservation were less than 5×10^{-3}.

RESULTS AND DISCUSSION

It is found that velocity fluctuations in a certain frequency range can produce discomfort. Melikov [9], refering results from Fanger and Pederson and from Assakay and Sakay, indicates that the frequency range of 0.3 to 0.5 Hz is quite critical.

In order to investigate if the airflow produced by the overhead nozzles contains velocity fluctuations in this range, a set of experiments was performed. A preliminary study was carried out using various combinations of acquisition frequencies and sampling periods in order to obtain a good reproducibility and a good time resolution of the signal during the tests. A sampling period of ten seconds with a rate of fifty points per second was adopted as being the best compromise.

In figure 3 the instantaneous axial flow velocity record for a period of ten seconds, as well as its respective frequency spectrum are shown. These measurements were made in the axis of one of the jets at a distance of 100, 300 and 500m from the exit, respectivelly. Looking at the different signals it can be concluded that the high frequency fluctuations are atenuated with increasing distance from the nozzle. The power spectrum of the fluctuations at the level of the passenger's head has a dominant peak at 0.2 Hz and some secondary peaks at 1.2, 1.4, 1.6 and 3.0 Hz. From this we conclude that the incoming flow does not contain velocity fluctuations in the range that is considered more discomfortable.

The mean velocity and turbulence intensity data were obtained in two planes. The first one was a vertical plane containing the axes of two jets, just below the nozzle exits, as is represented in fig. 4. A square measuring grid of 31x24 points, with an interval of 20mm, was used in this plane. When computing the turbulence intensity at each point, the mean value of the initial velocity of the imping jets was used instead of the value of the mean local velocity. Lines of constant velocity and constant turbulence intensity for the considered plane are presented in the figure 5.

At the level of the passenger head (400-500mm from the nozzle exit) the mean velocity in the center of the jet is about 1.5-2.5m/s, and the turbulence intensity varies between 3% and 4%. A slight dissimetry in the behaviour of the two jets is apparent, with a more pronounced enlargement of the left jet, probably as a consequence of the presence of the lateral wall.

To describe in detail the flow at the head level some additional measurements in a horizontal plane, 400mm distant from the nozzle exits, were done in a 46x16 measuring grid with an interval of 20mm. This plane is represented by the bold line in the figure 4. The mean velocity and turbulence intensity profiles relative to those measurements are represented in figure 6. It can be seen that the jets have radial symmetry and the area in which the flow velocity is larger than 0.1m/s is restricted to a circle of about 20cm in diameter.

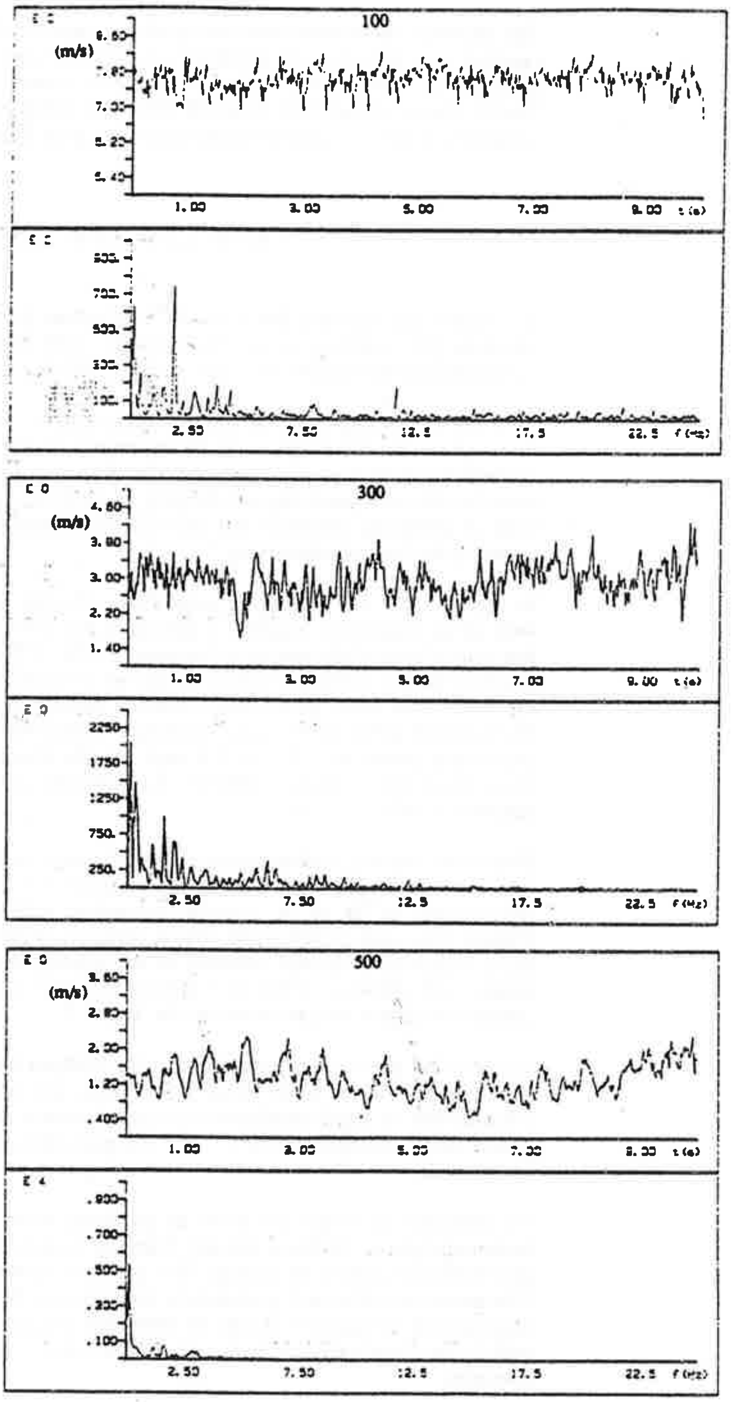

Fig. 3. Time dependent velocity signals and respective power spectra at distances of 100, 300 and 500mm from the nozzle exit.

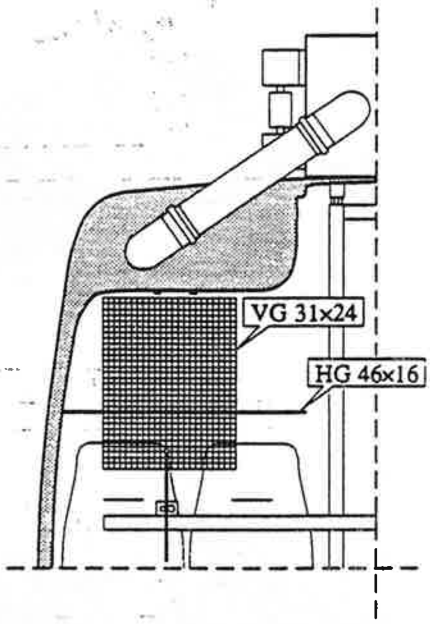

Fig. 4. Measuring planes used in mean velocity and turbulence intensity measurements.

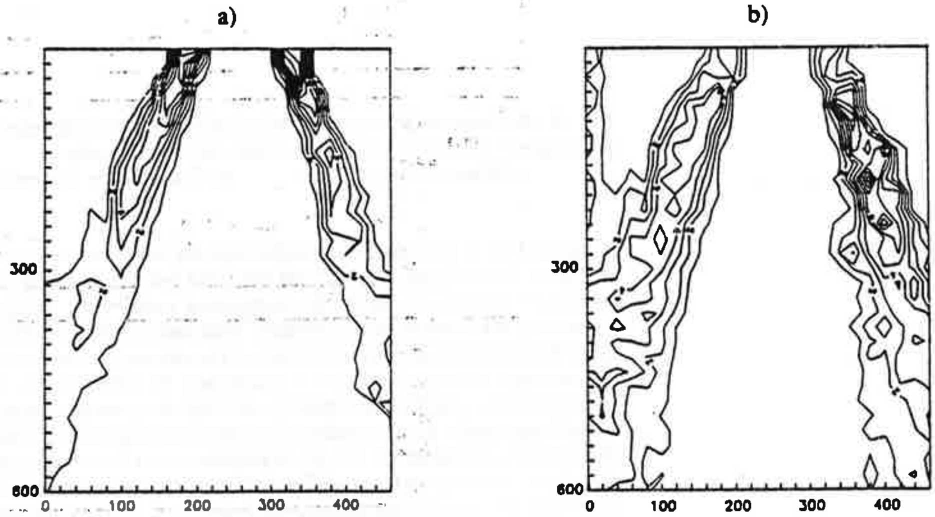

Fig. 5. Isolines in a vertical plane, over the seats and containing the axes of the jets.
a) Mean velocity (m/s) b) Turbulence intensity (%).

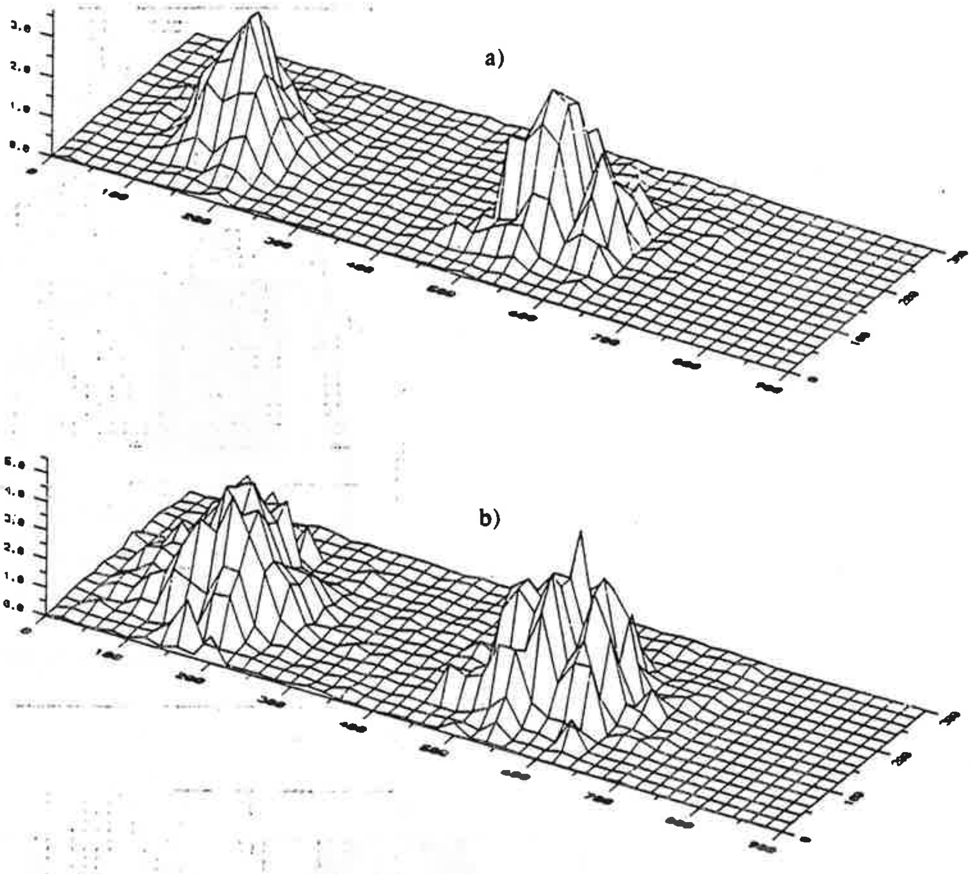

Fig. 6. Axonometric representation of the measured values in a horizontal plane at the passenger's head level (400mm from the nozzle exits).
a) Mean velocity (m/s) b) Turbulence intensity (%).

A sample of 2-D numerical predictions for the mean velocity field distribution is depicted in figure 7. In figure 7.a, 90% of the fluid fed through the jets is symmetrically readmitted into the ventilation ducts, the remaining part being exhausted through a fictitious hole located at the bottom of the section. This latter exit is an attempt to simulate that part of the flow that actually leaves the plane of the section at the level of the seats. In figure 7.b, the percentage of recycled fluid is decreased to 10%. None of these situations corresponds strictly to the conditions actually used for the present laboratory measurements. The first would approach an air conditioning recirculating circuit. The latter fails to correctly predict the acual trajectories of the jet centerlines, as a consequence of the actual 3-D character of the flow, particularly at the level of the seats. However, the decay of the mean velocity from the jet exits to the passengers head level is in agreement with the measured one. This allows one to expect that important gudelines for laboratory work can be numerically obtained for conditions that are closer to a two-dimensional behaviour. This is the case in figure 7.a or in the mixed convection flow generated by hot air injection through longitudinal fences placed on the floor, used in practice for heating purposes. These two situations will integrate the continuation of the present study.

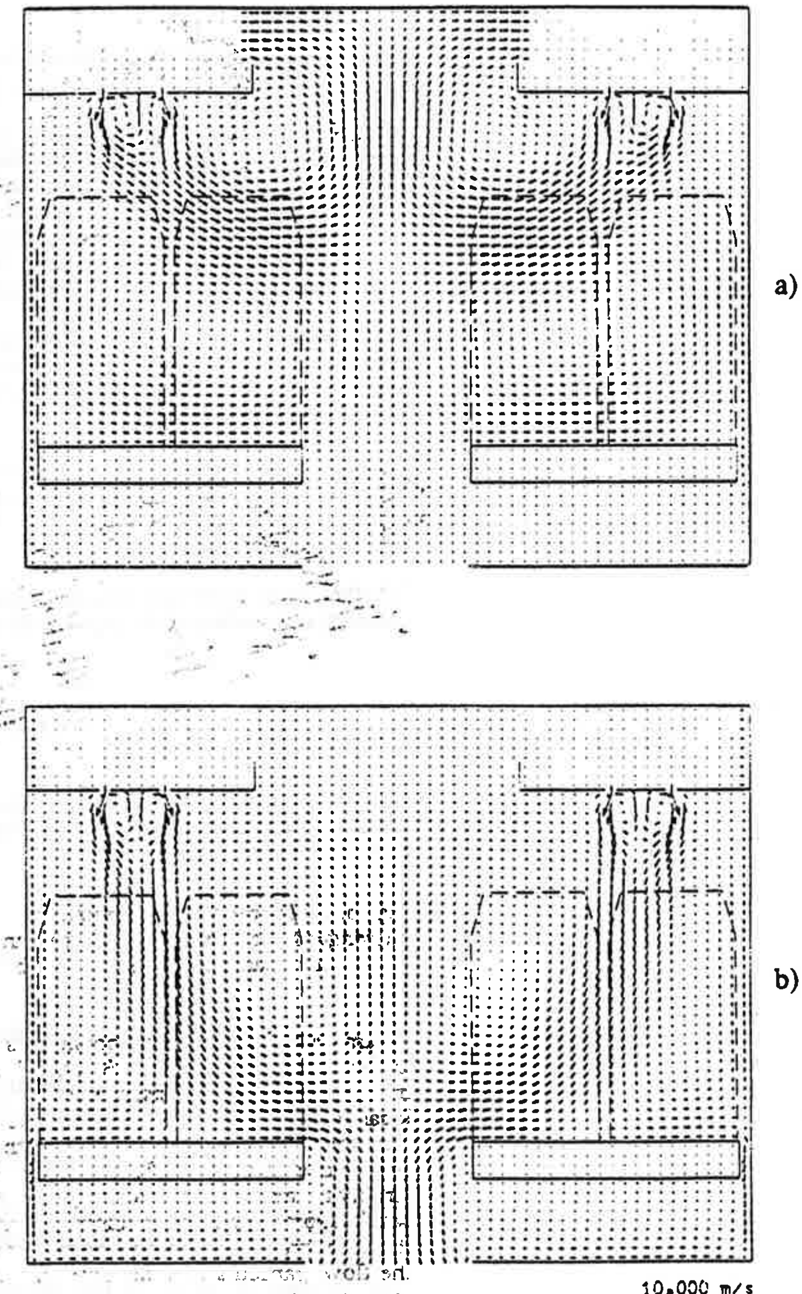

Fig. 7. Vectorial representation of the average velocity field in a vertical cross section of the passenger compartment: (a) 90% of recirculation, (b) 10% of recirculation.

CONCLUSION

The mean and turbulent isothermal flow in the passenger compartment of an intercity bus was analysed both by experimental and numerical methods.

The measured and computed values of the air velocity at the level of the heads of seated passengers were found to be very high, if compared with the recommended value of 0.25m/s, in summer conditions, by the heating and ventilation standards for indoor climates. However, taking into account the high heat load that the passenger compartment receives in summer due to solar radiation and the large glazed surface, a higher limiting value could presumably be tolerated. In this context the value of 0.5 m/s recommended by the Union of the International Transport of Passengers seems to be a reasonable target.

The cross section of the jet flow at the level of the passenger heads is very small compared with the size of a passenger, so a very localized action is obtained. The use of a larger area nozzle can give some improvement to the flow quality, with lower velocities and a more envolving flow.

Even though the actual flow is three-dimensional in character, it was concluded that interesting guidelines for future laboratory work may be obtained from preliminary two-dimensional theoretical predictions. This is namely the case of air cooling or heating operation conditions.

Further work in the bus module is planned with non isothermal conditions, using both heating and cooling of the ambient air as well as the simulation of solar radiation.

ACKNOWLEDGEMENT

The present work is included in a research project named "Design of a new type of Intercity Bus" supported by the EC Programme PEDIP and by Salvador Caetano IMVT, SA.

REFERENCES

[1] Fanger, P.O. "Thermal Comfort". Danish Technical Press, 1972.

[2] Olesen, B.W. "Measurement of Thermal Comfort in Vehicles". Brüel & Kjær application notes, Nærum Denmark, 1988.

[3] Olesen, B.W. and Rosendahl, J. "Thermal Comfort in Trucks" 1990.

[4] Temming, J. and Hucho, W.H. "Passenger-Car Ventilation for Thermal Comfort". SAE Technical Paper 790398, 1979.

[5] Klemp, K. et al. "LDA Measurements in a Model of a Passenger-Car-Cabin". Dantec Information nº 10, Skovlunde, Denmark, 1991.

[6] Launder, B.E. and Spalding, D.B."The Numerical Computation of Turbulent Flows". Comp. Meths. Appl. Mech. Engrng., 1974, 3, pp. 269-289.

[7] Patankar, S.V. "Numerical Heat Transfer and Fluid Flow". Hemisphere, Washington, D.C.,1980.

[8] Jang, D.S., Jetli, R. and Acharya, S. "Comparison of the PISO, SIMPLER, and SIMPLEC Algorithms for the Treatment of the Pressure-velocity Coupling in Steady Flow Problems". Num. Heat Transfer, 1986, 10, pp.209-228.

[9] Melikov, A.K. "Quantifying Draught Risk". Brüel & Kjær Technical Review nº 2, Nærum Denmark, 1988.