Air Distribution in Rooms, (ROOMVENT 2000)
Editor: H.B. Awbi

IMPROVEMENT OF CFD APPLICATION IN VENTILATED ENCLOSURES - A TEST CASE

C. Teodosiu[1], G. Rusaouen[1] and S. Laporthe[1]

[1]Centre de Thermique de Lyon (CETHIL), UPRES A CNRS 5008
Equipe Thermique du Bâtiment, Institut National des Sciences Appliquées
(INSA), Bât. 307, 20 avenue A. Einstein 69621 Villeurbanne Cedex – France

ABSTRACT

The aim of this study is to improve the utilization of CFD approach in the applications of air conditioning technology. More precisely, to establish principles and recommendations to follow in order to design air distribution systems in small enclosures at low room air changes per hour by means of CFD technique. By the use of a commercial code, Fluent, the accuracy and reliability of such a numerical simulation are elucidated in this work for a mixing ventilation system; the air supply terminal is a commercial diffuser which creates a complicated 3D - wall jet below the ceiling. We focus on the factors which have a major impact on the simulations: the description of the computational domain (particular emphasis is put on the supply airflow conditions), the turbulence model and the near wall treatment. Comparisons between predicted and measured values are given in terms of mean air velocity and temperature. The validation of the simulations is completed by an analysis related to analytical expressions for the velocity profile and centerline velocity decay of a three dimensional jet.

KEYWORDS

Numerical simulation, CFD, ventilation, turbulence models, near-wall flow, 3D –wall jet.

INTRODUCTION

At the beginning of the 21st century it is clear that the provision of a good air quality inside the ventilated enclosures as well as an acceptable level of energy consumption for this can not be achieved without modern computational techniques. In conjunction with the progress made in computer hardware, the CFD approach becomes more and more appropriate for the calculations concerning air distribution prediction in rooms, indoor air quality and thermal comfort. However, there is a need to ensure that these computational tools are correctly applied and the purpose of this study is to test the precision of such a numerical simulation by comparing the results obtained with those from experiments. In addition, this paper marked several guidelines that lead to a successful computation. The case taken into account is a mixing ventilation system for small enclosures at low room air changes per hour. We present a brief description of the experimental set-up in the next section.

PHYSICAL MODEL

The physical model is the test room 'MINIBAT' of the Thermal Centre Lyon – 'CETHIL'. Figure 1 illustrates the geometry of the model. The air supply terminal is represented by a commercial diffuser (a grille having an aspect ratio of 12.5) that was placed after a plenum – see figure 1.

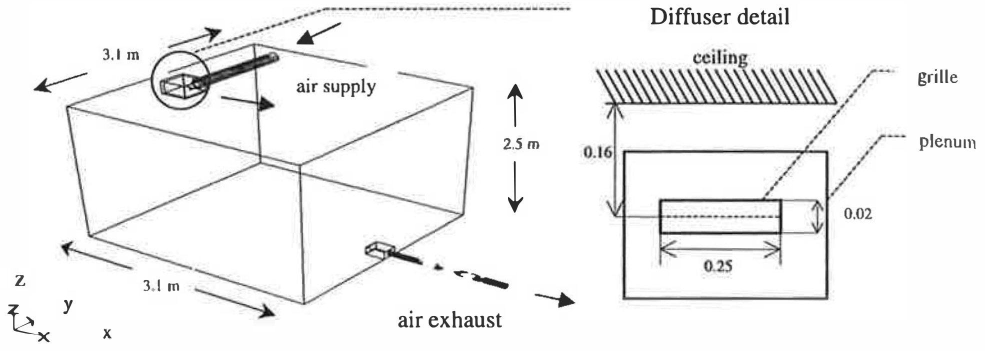

Figure 1: Test room MINIBAT and its ventilation system

Detailed descriptions of the test room and the diffuser are given in Castanet (1998). Measurements have been carried out in order to have a complete description of the boundary conditions (temperature and flow rate of the supply air, surface temperature on the inside walls). On the other hand, the experimentation allowed us to determine the velocity and temperature air distributions inside of the room in a vertical plane normal to the centre line of the air terminal devices. The experiments covered a range of Reynolds number, based on inlet dimensions, between 7000 and 14000 (namely 1-2 h^{-1} air changes rates) and that for non-isothermal conditions, the values of Archimede's number were in the range: 0.0032 – 0.015.

NUMERICAL MODEL

The simulations were carried out by the use of the commercial CFD code Fluent – version 5.0 that is a general purpose, finite volume, Navier-Stokes solver – Fluent (1998).

In order to predict the airflow within the physical model already presented, we assume that the air is a Newtonian fluid, incompressible that has a constant viscosity and is affected by the gravity force. In the same time, the flow presents the following characteristics: 3D, turbulent, steady and non-isotherm. In our work, we focused on the main elements that have a major influence on the simulations. These key factors are discussed in the next sections.

Computational domain description and supply air flow conditions

This point is a critical one in the prediction of airflow in a ventilated room because it is well known that the flow in an enclosure is mainly governed by its properties at the supply inlet.

We considered several methods in order to reach a better description of the air diffuser and its inlet boundary conditions. First, we have taken into account the simplified methods presented in the studies within the work of the IEA (Annex 20 – 1993), Heikkinen (1991). Unfortunately, the results obtained, by applying these methods, in terms of air velocity in the jet zone were in disagreement with the experimentation. This surely means that these methods are related to a certain type of air terminal device. In fact, after a thorough study, we noticed the basic elements that must be taken into account for a successful simulation in our case. It is possible to replace the complicated diffuser with a simple opening but this opening must have the same aspect ratio as the real diffuser and must be located in the middle of the real diffuser. (The jet is a 3D-wall jet and every variation concerning the aspect ratio and the distance from the ceiling determines an altered behaviour of the jet). Moreover, in order to have a properly formed jet, we changed the real geometry (figure 2): we specified the inlet boundary conditions at a larger distance than the position of the virtual origin of the jet. Also, we imposed a thickness (0.02 m) for the air terminal device in order to have more computational cells in this region which represents now a flow passage.

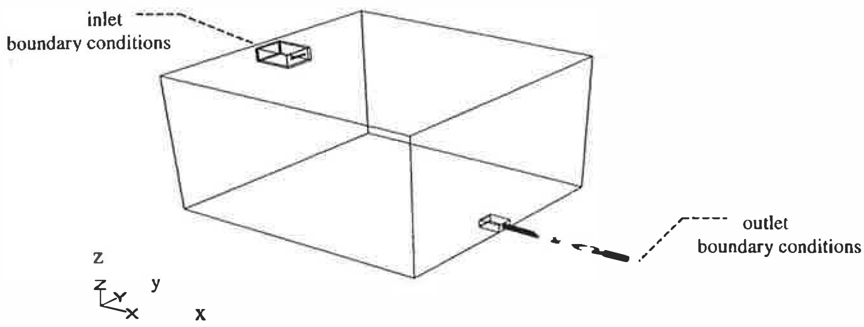

Figure 2: Numerical model geometry

Choice of turbulence model

The reliable representation of turbulence in airflow simulation modelling has proved to be an extremely difficult task and it is obvious that there is no single turbulence model which is able to cover all classes of problems. Besides, there are several parameters which could have an effect on the choice of a turbulence model such as the practice achieved for a specific domain of applications and the available computational resources.

There are four classes of turbulence models in Fluent: one-equation model (Spalart-Allmaras), two-equation models (k-ε), second order closure model (Reynolds stress model) and the large eddy simulation model (LES).

It is known that one-equation models are not always able to give accurate results when the flow changes rapidly from a wall-bounded to a free shear flow. On the other hand, the second order closure models have not found a very large application in simulations concerning ventilation problems since they don't allowed much better results than the k-ε model despite their complexity. If we add the more computer power required in the case of the stress models as well as for LES methods, we can understand why the two equation turbulence models still remain the preferred approaches for numerical simulation of ventilation flows even that the LES models become more and more of interest – Davidson & Nielsen (1996).

In our study, we used a variant of the well-known standard k-ε model – Launder & Spalding (1972), the so-called 'realizable' k-ε model which demand very little more computational effort than the standard k-ε model. This revised k-ε model includes new formulations for the turbulent viscosity and for the dissipation rate transport equation – for details see reference Fluent (1998). The comparisons carried out during our work between the standard k-ε model and its 'realizable' version have shown that this model leads to the best spreading rate predictions of the jet and this was a crucial point in our simulations knowing the particular behaviour of a 3D dimensional wall jet.

Near wall treatment

The correct specification of boundary conditions is a vital part of a numerical calculation of indoor airflow. There are always difficulties regarding the description of the near-wall region where the turbulent flows are affected by the damping influence of the solid walls. There are two techniques to deal with the near-wall region: the classical logarithmic wall functions and the two-layer approach.

The application of wall functions fails in our case as separation occurs – especially at the ceiling. Therefore we abandoned this approach despite its advantage (savings in computational effort) and we employed the two-layer near-wall model. This approach is based on the partition of the domain into a viscosity affected region and a fully turbulent region. The separation of the two regions is established by the turbulent Reynolds number (Re_y) – based on the normal distance from the wall at the cell. In the viscosity near-wall region (where $Re_y < 200$), a one-equation model is employed. Hence, in this case, the rate of dissipation of the turbulent kinetic energy (ε) is obtained algebraically by the use of length scales. The equations that allow to compute the turbulent viscosity and ε, as well as the formulas employed for the length scales can be found out in Fluent (1998).

Boundary conditions

- flow inlet boundaries:

velocity: fixed value across the opening located at the entree of the plenum (figure 2). The velocity component perpendicular to the opening was determined as the ratio of the measured airflow rate to the area of the opening.

temperature: uniform value using the experimental data

turbulence quantities: uniform specification, defining two parameters: the turbulence intensity – I, based on an empirical correlation for pipe flows, assuming a fully developed duct flow upstream, $I = 0.16(Re_{D_H})^{-1/8}$ (%) and the hydraulic diameter - D_H (m). The relationships used in order to compute the turbulent kinetic energy (k) and its rate of dissipation (ε) based on the I and D_H are presented in reference Fluent (1998). A brief parametric study was completed in order to see the influence of the inlet boundary conditions on the jet development. We noticed that the values specified for the turbulent parameters at the entree of the domain do not notably influence the flow pattern. At the contrary, the impact of the dynamic inlet boundary conditions on the flow within the enclosure is important. These two remarks are in accordance with the conclusions of other studies – Joubert (1996).

- *wall boundary conditions:* the thermal boundary conditions were imposed as fixed values of temperature at the wall internal surface using the measured values.

- *outflow boundaries:* a zero diffusion flux for all flow variables (except pressure). This condition presumes a fully developed flow and we located the outflow boundary away from the real exhaust device in order to fulfil this supposition – see figure 2.

Numerical scheme

The discretisation of the computational domain was realised by means of tetrahedral mesh elements and the main characteristics of our numerical computations were (the solver uses a co-located scheme, consequently the pressure and the velocity are both stored at cell centres):

- implicit-integration version of the solver
- diffusion terms are second-order central-differenced
- second-order upwind scheme for convective terms in order to reduce the numerical diffusion
- SIMPLE algorithm for the velocity-pressure coupling method

RESULTS

The experimental and numerical results were obtained for hot, cold as well as for isothermal jets. In our paper we developed only the comparisons for a heated jet (two room air changes per hour), the values of the Reynolds and Archimede's numbers being 13566 and 0.0032, respectively. It is worthwhile to note that the conclusions regarding this case are pertinent too for the rest of the configurations tested.

The comparisons between predicted and measured values are shown in figure 3 in terms of air mean velocity and temperature profiles in a median vertical plane for three sections located at different distances from the coordinate system presented in the figure 1.

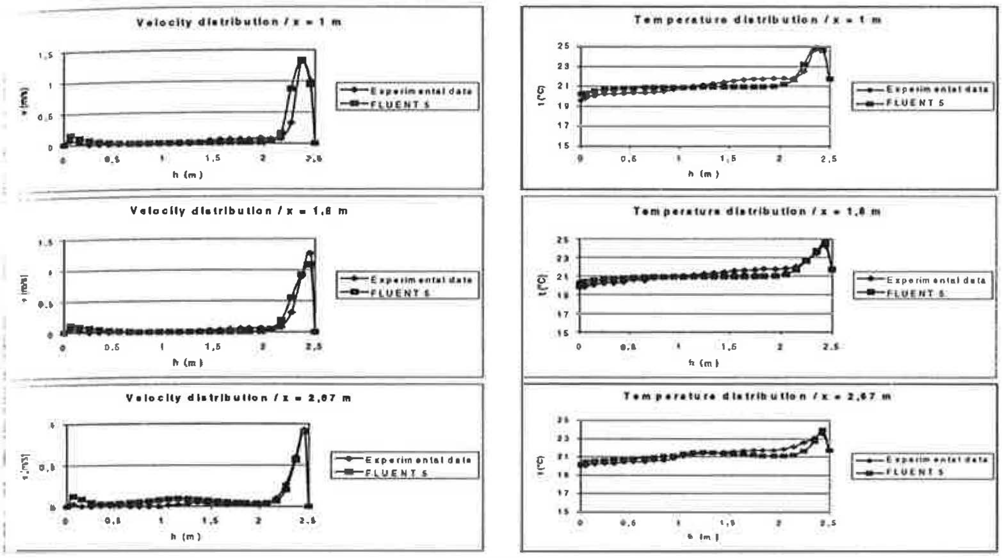

Figure 3: Vertical mean velocity and temperature profiles

They are related to the characteristic and axisymmetric decay region of the 3D jet. In fact, the first one (x = 1 m) is placed at the beginning of the characteristic region, the next one (at x = 1.8 m) is situated at the end of the same characteristic region and the last one (at x = 2.67 m) represents the axisymmetric zone of the jet. The length estimation of these regions was based on the study Trentacoste (1967). We completed our comparisons by a velocity profile jet obtained using empirical formulae. First, we determined the rise of the hot jet (the trajectory) applying the equation obtained by Frean and Billington presented in Awbi (1995). Afterwards, we employed the Sfeir's velocity decay equation – Sfeir (1976) - with a value of 6.25 for the throw constant, K_V. Finally, using the expression proposed by Sforza (1977), we obtained the velocity profile, taking into account the following relationship for the distance from the centre line of the jet where the velocity is equal to half the maximum velocity: 0.1 x – where x represents the distance from the opening. Figure 4 illustrates a comparison between numerical, experimental and empirical velocity profile of the jet at 1.4 meters from the air supply, in fact in the fully developed flow region of the three-dimensional jet. Based on the results shown in the figure 3, it can be demonstrated that the model agrees well to the experimental data, especially when compared with the thermal field (3,5 % maximal difference between simulated and measured values). Regarding the dynamic field, it can be concluded that the jet region is quite correctly predicted. Moreover, the penetration length is accurate performed and that means that the effect of the opposite wall is well taking into account. This fact provides an exact flow pattern in the room. However, the detailed comparison presented in the figure 4 reveals a larger spread of the numerical jet in comparison

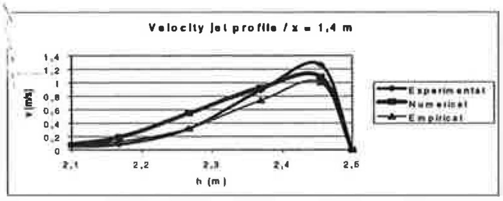

Figure 4: Velocity jet profile

with the experiment and empirical formulations. In our opinion, this is due to the turbulence model. On the other hand, in the region close to the ceiling where complex flows take place, the jet profile is properly numerical represented. This demonstrates that the simulations carried out in terms of near wall treatment and mesh generation in that particular region were correctly performed.

CONCLUSIONS

The test case analysed in this paper demonstrates that the representation of a complicated commercial diffuser and the prediction of a complex flow in small ventilated enclosure can be properly reached by the means of CFD simulation. There is a good agreement between numerical and experimental data. Therefore, the simulated solutions can be used to complete the measurements in works dealing with comfort studies in the occupied zone, ventilation efficiency, as well as prediction of pollutant diffusion in rooms. On the other hand, this study proves once more that the CFD approach is an interesting tool in the design of ventilation systems. The purpose of the guidelines presented here (the simplified description of the inlet conditions, the choice of an adequate turbulence model, the near wall treatment) is to improve the application of this technique in the design of air distribution systems for small ventilated rooms.

REFERENCES

Awbi H.B. (1995). *Ventilation of Buildings*, E & FN Spon, London, UK

Castanet S. (1998). *Contribution à l'étude de la ventilation et de la qualité de l'air interieur des locaux.* Ph.D. thesis. Insa de Lyon, France

Davidson L. and Nielsen P. V. (1996). *Large Eddy Simulations of the Flow in a Three-Dimensional Ventilated Room.* ROOMVENT '96, 161-168.

Fluent User's Guide, Version 5.0. (1998). Fluent Inc., Lebanon – NH, USA

Heikkinen J. (1991). *Modelling of a Supply Air Terminal for Room Air Flow Simulation.* 12th AIVC Conference, 213-230.

Joubert P.; Sandu A.; Béghein C.; Allard F. (1996). *Numerical Study of the Influence of Inlet Boundary Conditions on the Air Movement in a Ventilated Enclosure.* ROOMVENT '96, 235-242.

Launder B.E. and Spalding D.B. (1972). *Lectures in Mathematical Models of Turbulence*, Academic Press, London, England

Sfeir A.A. (1976). *The Velocity and Temperature Fields of Rectangular Jets.* Int. J. Heat Mass Transfer, Vol. 19, 1289-1297.

Sforza P. (1977). *Three-dimensional Free Jets and Wall jets: Applications to Heating and Ventilation.* Int. Seminar of the Int. Center of Heat and mass Transfer – Dubrovnik, 283-295.

Trentacoste N. and Sforza P. (1967). *Further Experimental Results for Three-dimensional free Jets.* AIAA Journal, Vol. 5, 885-891.