AIR MOVEMENT & VENTILATION CONTROL WITHIN BUILDINGS

12th AIVC Conference, Ottawa, Canada
24-27 September, 1991

POSTER 46

MODELLING OF A SUPPLY AIR TERMINAL FOR ROOM AIR FLOW SIMULATION

Jorma Heikkinen

Technical Research Centre of Finland
Laboratory of Heating And Ventilating
Lämpömiehenkuja 3, SF-02150 Espoo, Finland

SYNOPSIS

The paper discusses methods to set boundary conditions at the air supply opening in predictions of room air flows with computational fluid dynamics. The work is a part of the International Energy Agency project "Air Flow Patterns within Buildings", Annex 20.

The air supply terminal in the Annex 20 project is a commercial diffuser which creates a stagnation region and a complicated wall jet below the ceiling.

Fairly well predictions in the wall jet region were obtained replacing the diffuser by a simple opening which has the same momentum flow as in the diffuser. The momentum flow was well known which is not usual for complicated diffusors. In the initial section of the jet, the numerical method causes unintentional and uncontrollable mixing, which resembles the diffusion properties of the actual diffuser. The simple opening case was also measured and the agreement with the diffuser case in the wall jet was found to be satisfactory.

More advanced methods as the momentum method and the prescribed velocity method allow more freedom to modify the jet flow. They should be preferred in practice.

1. INTRODUCTION

Computational fluid dynamics (CFD) is increasingly becoming a practical tool to predict air and contaminant flows in ventilated spaces. A general discussion of the method applied to the room air flows is given in reference [1]. The computational principle is simple: the room is divided into a number of volume cells, say, 20 000 cells, and the balance equations of mass, momentum and heat are solved for those cells. Nevertheless, compromises in practical computations have to be made because of limited computer capacity and incomplete turbulence models. Therefore it is important to compare the effects of different simplifying assumptions with full scale measurements. This kind of work has been done in an international project, IEA Annex 20. This paper is a part of that project and it compares different simplified ways to set boundary conditions for a commercial air diffuser.

The small details of a supply air terminal have an obvious influence on the air flow field in the mixing type of ventilation. These details cannot however be handled in most practical air flow simulations and therefore simplifying assumptions are needed. Different methods to model complicated air terminal devices are described by Nielsen [2]. In this paper various methods are used and compared with measurements.

2. TEST CASE

The test room in the IEA Annex 20 project is a small empty office room shown in figure 1. The air supply terminal is located 0.2 m below the ceiling. It consists of 84 ball nozzles which are all directed 40 degrees upwards, see figure 2. A detailed description of the test room and the diffuser is given in references [3] and [4].

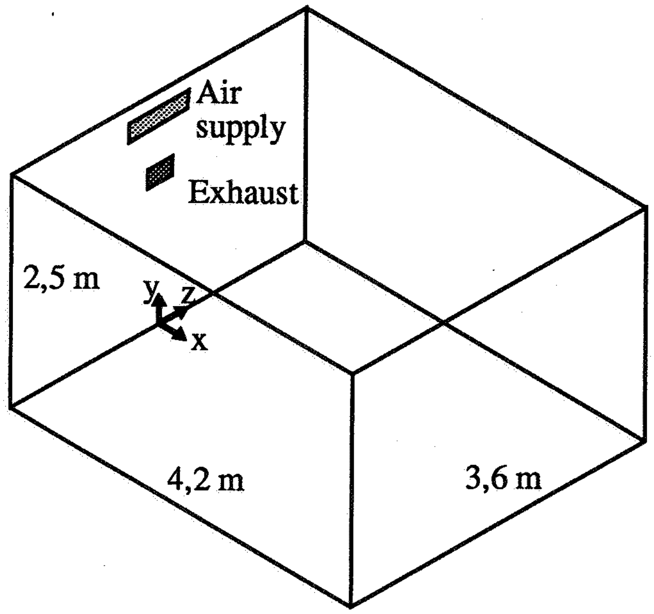

Figure 1. The test room.

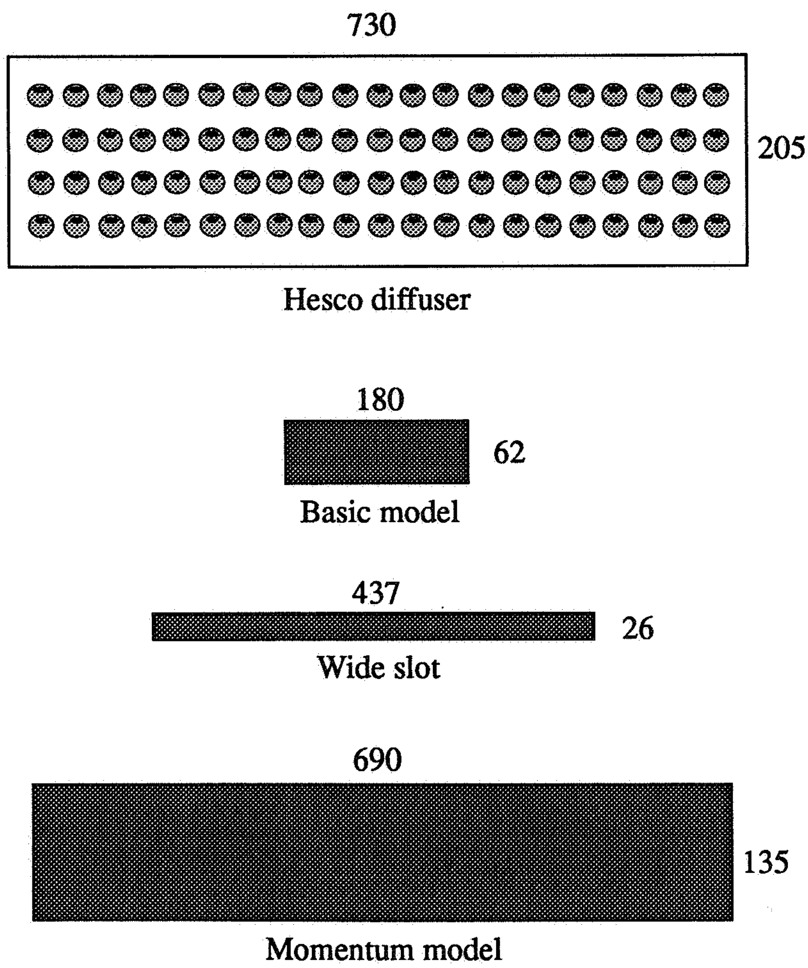

Figure 2. The 84-nozzle supply air diffuser and the methods to replace it by a simple opening. Dimensions are in millimetres.

The air change rate is 3 air changes per hour, which means an air flow 0.0315 m^3/s. Only isothermal flow in discussed in this paper. It is believed that the results can be useful also in non-isothermal simulations if buoyancy effects are small in the early stages of the jet development.

The observed flow field near the diffuser can be seen in figure 3. The velocity in the nozzles is about 3.7 m/s and it decreases quickly when the 84 small jets combine into a single jet. A maximum velocity of 1.5 m/s was measured in the combined jet at a distance of 0.1 m from the wall. The combined jet impinges on the ceiling where the upward momentum force of the jet is balanced by a pressure increase in the stagnation region. From the stagnation region the jet spreads below the ceiling to all directions, also to the left upper corner where a recirculating zone exists.

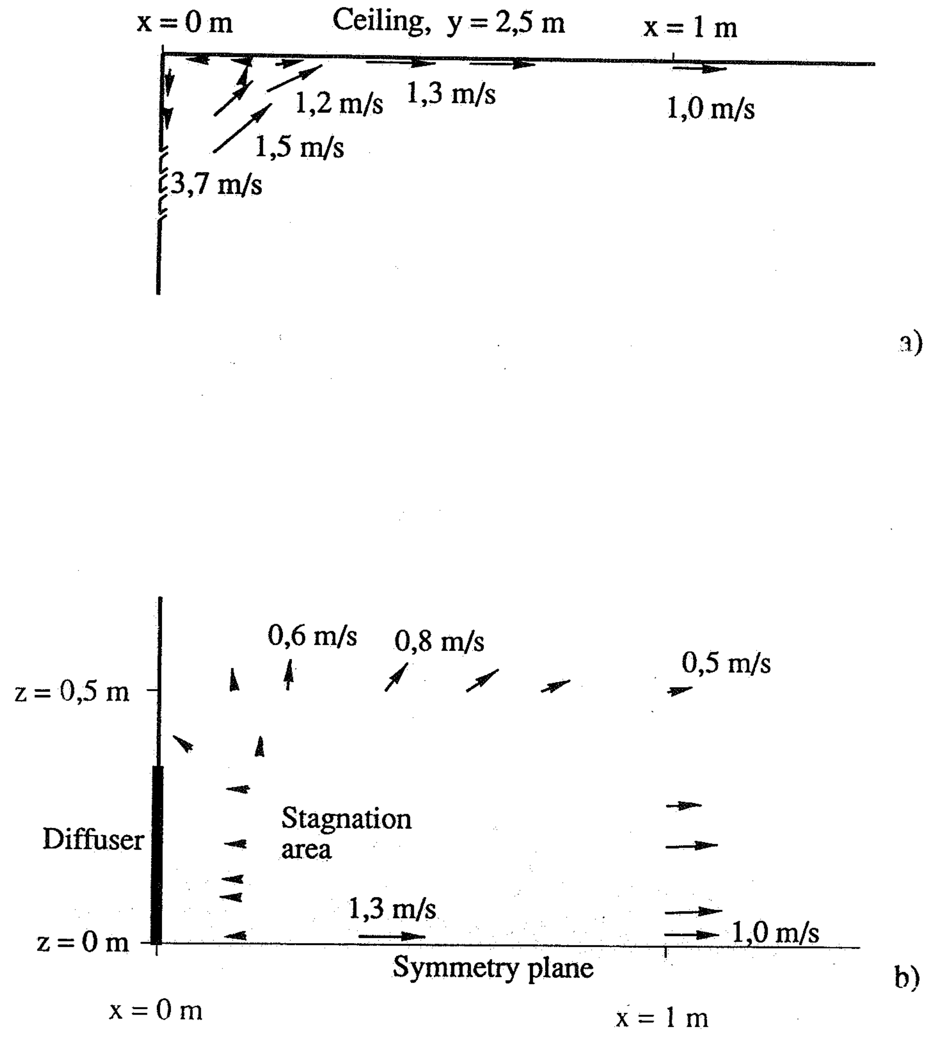

Figure 3. The observed flow field near the diffuser, at the symmetry plane (a), and just below the ceiling (b). Note that only half of the symmetrical room is shown in the figure b.

Air speeds were recorded in 560 locations in the room according to IEA Annex 20 specifications [3]. Velocity profiles in the jet were also measured and these results are mainly used in this report. Detailed measurement results of the same test case have been published already by Skovgaard et al. in reference [5].

Additional measurements were also performed for a case where the diffuser was replaced by a rectangular opening which has the same effective area as the diffuser. This corresponds to the basic model for the supply air terminal discussed in this paper.

The accurate computation of the complete flow field near the diffuser and also near the stagnation area is difficult because a fine grid is needed and standard turbulence models such as the as k-ε model are not perhaps valid. But fortunately the momentum flow of the diffuser is fairly well known and will make the task easier than in most other practical cases. A discussion of the momentum flow is given in appendix 1 after the main text.

3. SIMULATION METHOD

The main features of the simulations are the finite volume method [6], a staggered grid for the velocity components [6], the high Reynolds number k-ε turbulence model and logarithmic wall functions [7]. Fluent code [8] and Wish code [9] have been used. In Fluent, two differencing schemes can be used, namely the "power-law differencing scheme" (PLDS) recommended in reference [6] and the "quadratic upstream interpolation for convective kinematics", which is less prone to false diffusion [10]. Wish code has been used because it enables us to prescribe variables in the flow field and it constitutes an open code for modifications. The present version of the Wish code uses only an upwind differencing scheme, which is comparable with the power-law scheme.

Computations have been carried out on only half of the room because the room is symmetrical. The number of volume cells varied from 6 300 to 22 800 in different simulations. To be able to use wall functions and the high Reynolds number turbulence model, the distance from the first grid point to the wall should be selected properly. Near the ceiling the distance varied between 9 and 25 mm when using different grids.

4. MODELS FOR THE SUPPLY AIR TERMINAL

4.1 Basic model

It is possible to replace the complicated diffuser with a simple opening which has the same effective area as the small nozzles together. One has only to decide the shape of the opening and its location. IEA Annex 20 has agreed to use an opening which has the same aspect ratio as the real diffuser and is located in the middle of the diffuser, see figure 2. The supply air opening width and height are 180 mm and 62 mm respectively and the centre of the opening is 285 mm from the ceiling [11]. The supply velocity is 3.68 m/s and it is directed upwards at an angle of 40°. The turbulence kinetic energy is 0.204 m^2/s^2 and its dissipation 6.65 m^2/s^3. Turbulence energy corresponds to a 10 % turbulence intensity and the dissipation corresponds to developed channel flow. This is called a basic model.

The basic model case was also measured by replacing the diffuser by a rectangular duct which was 2 m long and aligned 40° upwards. The velocity profile at the opening thus corresponds to a developed channel flow. The dimensions of the opening were increased by 4 mm to get the same maximum velocity at the opening as in the nozzles. The measured maximum velocity in the middle of the opening was 4.0 m/s, which corresponds to the maximum velocity found in the nozzles in the middle of the diffuser [16].

4.2 Wide slot

Making the simple opening wider is believed to result in more mixing in the early stages of jet development because the perimeter of the jet is greater than in the basic model and corresponds more closely to reality. A width of 437 mm and a height of 26 mm were selected, see figure 2. The area and the turbulence quantities are the same as in the basic model.

4.3 Momentum model

The area of a simple opening can be freely selected by using a so called momentum model which has been used earlier by Chen et al. [12] in the Phoenics program. This makes it possible to set separate boundary conditions for the continuity equation and the momentum equations. Unfortunately in the Fluent program (version 2.99) and in the Wish program the given inlet velocity determines the boundary source terms for all equations. Therefore the Wish program was modified to allow free setting of boundary source terms. The width of the opening was selected to be 690 mm and the height 135 mm, which is the area occupied by the nozzles. The inlet momentum and the turbulence quantities are the same as in the basic model.

4.4 Box model

This is a model where the boundary conditions are given at the surface of an imaginary box around the diffuser. The box was selected to be 0.4 m high, 1 m long and 1 m wide. The air speeds on all four surfaces of the box were measured with TSI 1640 omnidirectional hot film anemometer. In addition to speed measurements, flow directions also had to be detected using smoke. The anemometer was not suitable for measuring turbulence; consequently turbulence values applicable to a two-dimensional wall jet [13] were given. An example of the measured profiles is shown in figure 4.

4.5 Prescribed velocity method

In this model boundary conditions are given at a simple opening and also in the flow field as in the box model. The idea is to minimize the necessary measurements by giving only the most important variables in the most important locations in the jet and to compute the rest. In this case only the x-direction velocity was specified in a small plane at a distance of 1 m from the supply terminal. The width of the plane was 0.6 m (0.3 m on both sides of the symmetry plane) and the height was 0.13 m where the velocity is about 35 % of the maximum value, see figure 4. The present version of the method means a simplification of the method described in the reference [2], where two velocity components are given on two perpendicular planes. The idea here is to test if the flow field produced by the basic model can be easily revised.

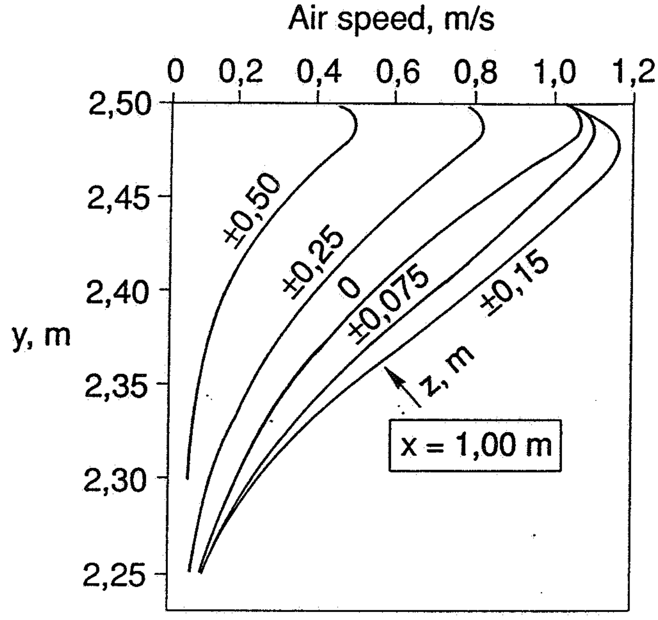

Figure 4. The air speed profiles measured at a distance of 1 m from the diffuser. These have been used in the box model and in the prescribed velocity model.

5.0 RESULTS

The supply air terminal has the most direct influence on the decay of jet velocity and also on the shape of the jet. These should be predicted correctly to be able to predict also nonisothermal cases. The discussion in this paper deals mainly with those properties. A short discussion of the maximum velocity in the occupied zone in different cases is given below before proceeding to a detailed analysis.

5.1 Maximum velocity in the occupied zone

The most interesting property for thermal comfort is the maximum velocity, which is shown in table 1 for various simulations and also for the measurements. The occupied zone has here been defined such that the volume above 1.8 m the floor and the volume closer than 0.6 m to the walls has been excluded.

From a practical viewpoint the simulated maximum velocities are fairly close to each other and also close to the measured velocities. When looking at different models for the supply air terminal it can be concluded that it is not only the diffuser model which has an effect on maximum velocity but also the number of grid points, the differencing scheme, and even the code that has been used has an effect. It is obvious that most results are not grid-independent. It will be shown later that in the early stages of the jet development numerical diffusion plays an important role.

Table 1. Maximum velocity in the occupied zone and its location in different measurements and simulations.

Case	*Velocity m/s*	*x m*	*y m*	*z m*
Measured, diffuser case	*0.19*	*3.0*	*0.03*	*0*
Measured, basic model (simple opening)	*0.20*	*2.2*	*0.05*	*0*
Basic, Fluent code, power-law scheme, 6300 points	*0.22*	*2.5*	*0.05*	*0.2*
Basic, Fluent code, QUICK scheme, 6300 points	*0.27*	*2.5*	*0.05*	*0.3*
Basic, Fluent code, power-law scheme, 22800 points	*0.23*	*2.4*	*0.16*	*0*
Basic, Fluent code, QUICK scheme, 22800 points	*0.24*	*2.6*	*0.03*	*0.6*
Basic, Wish code, 6300 points	*0.19*	*2.8*	*0.05*	*1.2*
Wide slot, Fluent code, QUICK scheme, 22800 points	*0.24*	*2.6*	*0.03*	*0.6*
Momentum model, Wish code, 8580 points	*0.16*	*2.8*	*0.05*	*1.2*
Prescribed velocity model, Wish code, 6300 points	*0.22*	*2.8*	*0.05*	*1.2*

5.2 Basic model

Computed flow field near the diffuser in figure 5 looks similar to the one observed, figure 3. There are however some differences. Near the ceiling the flow field corresponds fairly well to the observed field except near the stagnation area, which is much smaller in the z-direction in the computations because of the much narrower air inlet opening (fig. 5 b). Near the inlet (fig. 5 a), the computed jet seems to spread more in the vertical and horizontal directions than does the measured jet. One reason for this phenomenon is the staggered grid, which sets the inlet sources of x and y momentum in different locations. The other is numerical diffusion, which can be considerable because the velocity vectors are not aligned with the grid lines. Because of jet spreading, the recirculation in the left upper corner is smaller than in real conditions.

The spreading of the jet near the diffuser would be smaller if a finer computing grid could be used. Here the grid consists of 38x40x15 = 22 800 cells, which is the finest grid used because the computing time was already too long: about 4 hours on a Cray X-MP computer. The QUICK differencing scheme was used, which took twice as much computing time the power-law scheme does but reduces numerical diffusion. This can be seen in figure 6, where the decay of the jet is shown in simulations with various differencing schemes and different grids. It may be concluded that a very fine grid is needed to achieve grid-independent results, especially near the diffuser. Excessively low maximum velocity predictions were also observed in the reference [14] just beyond the supply opening owing to limited number of grid points. In the wall jet region it seems like the grid has an influence on the velocity decay. Perhaps the first grid point below the ceiling is too close to the surface (13 mm) in the fine grid and causes too high friction.

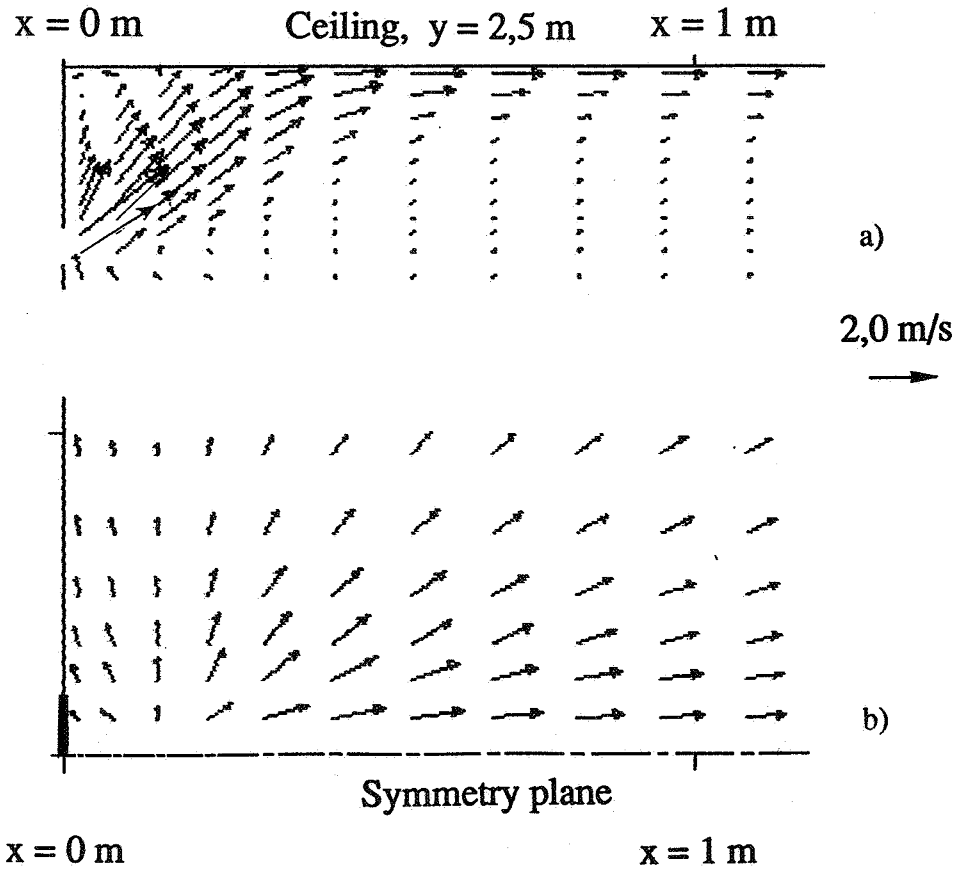

Figure 5. The flow field near the diffuser using the basic model. The symmetry plane (a) and a plane 13 mm below the ceiling (b) are shown.

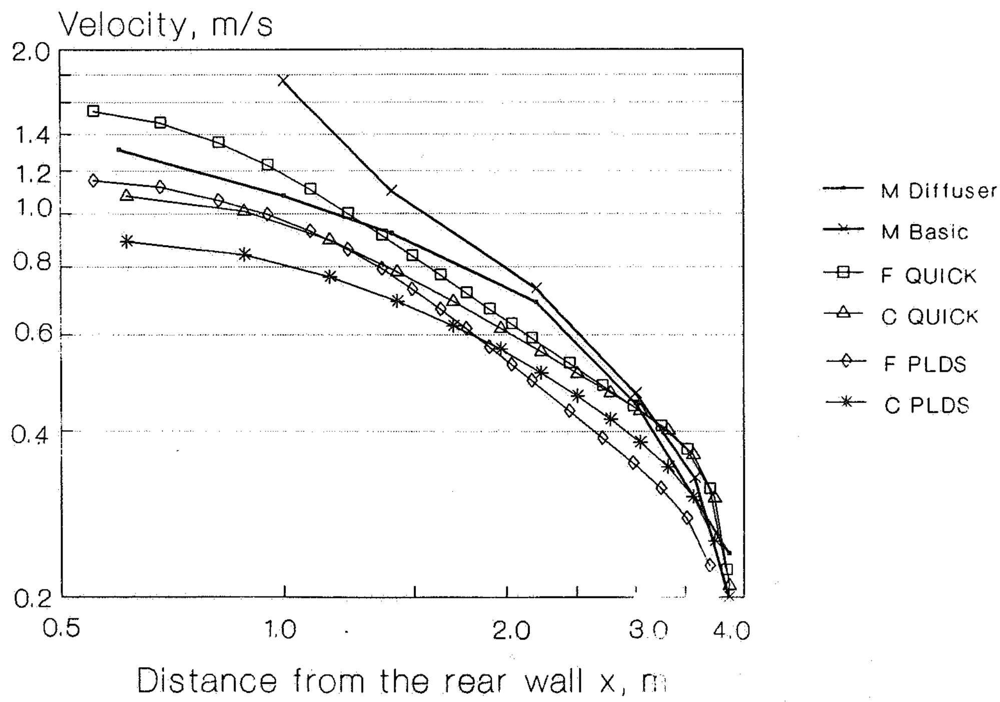

Figure 6. Measured and simulated velocity decay in the symmetry plane using the basic model. In the legend "M" refers to the measurements, "F" to the computations using the fine grid an "C" to the computations using the coarse grid.

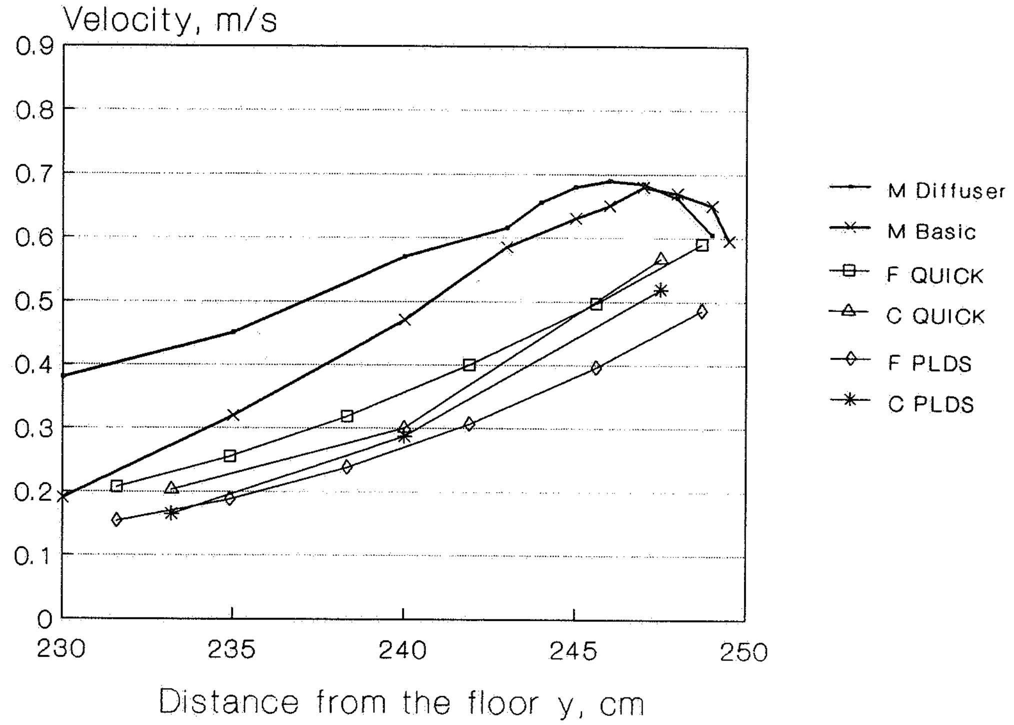

Figure 7. Measured and simulated velocity profiles in the vertical direction at 2.2 m from the diffuser in the symmetry plane using the basic model. In the legend "M" refers to the measurements, "F" to the computations using the fine grid and "C" to the computations using the coarse grid.

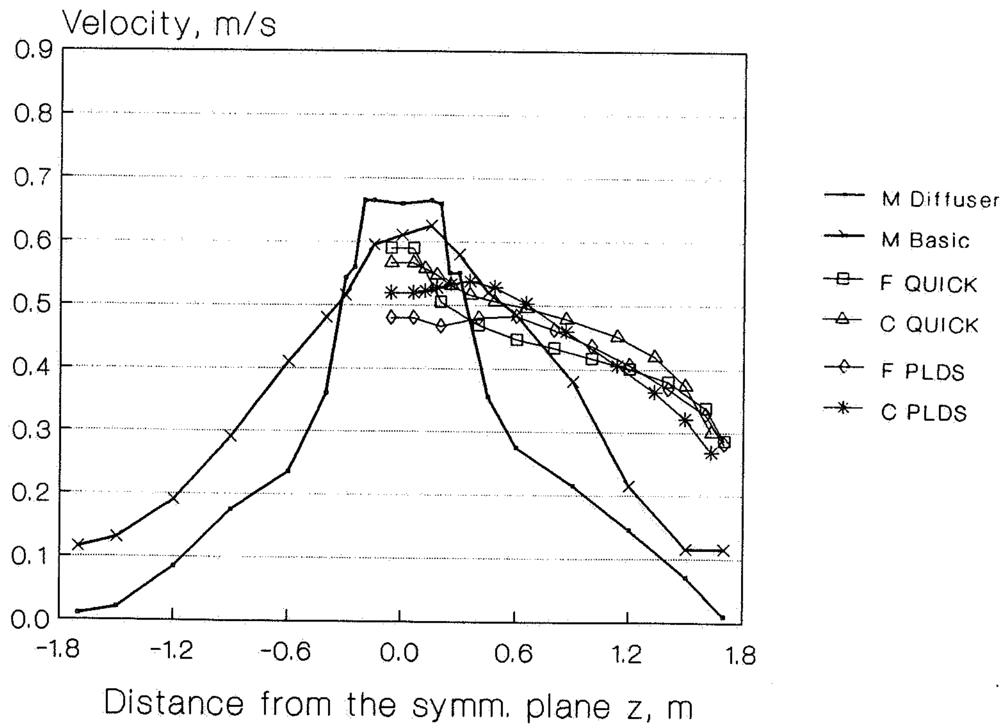

Figure 8. Measured and simulated velocity profiles in the horizontal direction at 2.2 m from the diffuser and near the ceiling using the basic model. In the legend "M" refers to the measurements, "F" to the computations using the fine grid and "C" to the computations using the coarse grid.

The velocity profiles in the wall jet near the ceiling at 2.2 m from the diffuser can be seen in figures 7 and 8 in the vertical and horizontal directions respectively. In the vertical direction the thickness of the wall jet is smaller in the computations than in the measurements. Note that the velocity maximum in the y-direction could not be predicted. In the horizontal direction the computed jet spreads more than the measured jet.

Figures from 6 to 8 also present the measured results for the basic case. The velocity is higher in the early part of the jet than in the diffuser jet. The differences become quite small during the wall jet development. This shows that from a physical viewpoint the basic model is fairly good. Apparently smaller mixing near the opening will be compensated by mixing in the wall jet region. The jet from a simple opening spreads more in the horizontal direction than the diffuser jet resembling the simulations.

5.3 Wide slot

The simulations were performed using the QUICK scheme and nearly the same fine grid as in the basic model. The decay of the jet is shown in figure 9 for various models. Using the wide slot model, at small distances mixing is increased compared with the basic model, as expected. The mixing seems to be even higher than it is in the real conditions. The decay of the jet is very slow because the jet is thick at its start near the stagnation area and therefore the characteristic decay for an axisymmetric or radial jet starts very late, at x-distances greater than 3 m.

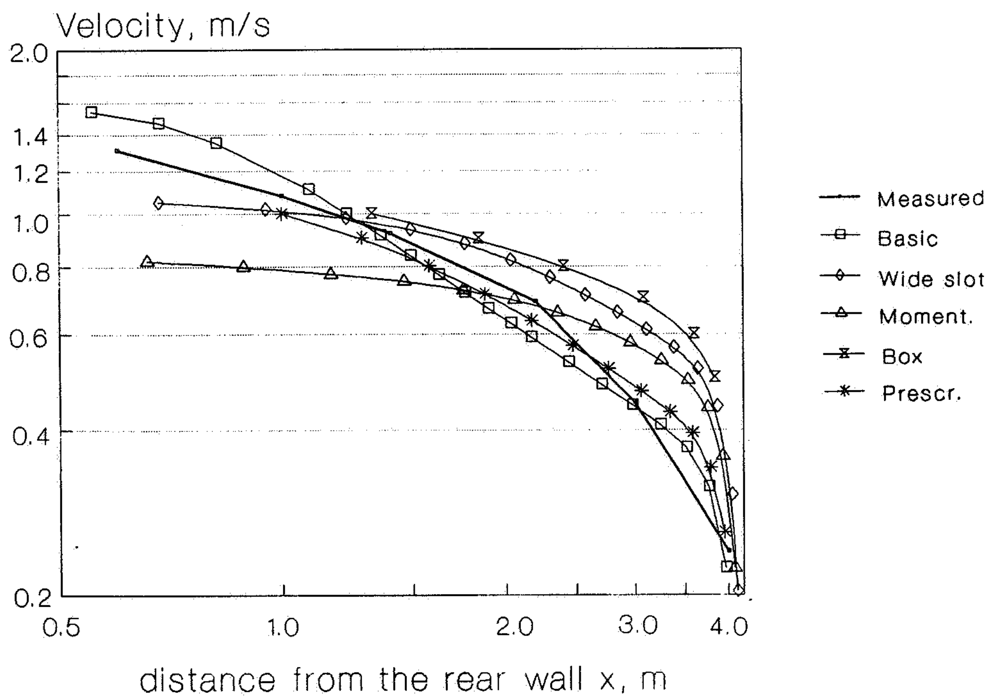

Figure 9. Measured and simulated velocity decay in the symmetry plane using various models for the supply air terminal.

Velocity profiles in the vertical direction (fig. 10) and in the horizontal direction (fig. 11) are fairly well predicted at a distance of 2.2 m from the diffuser.

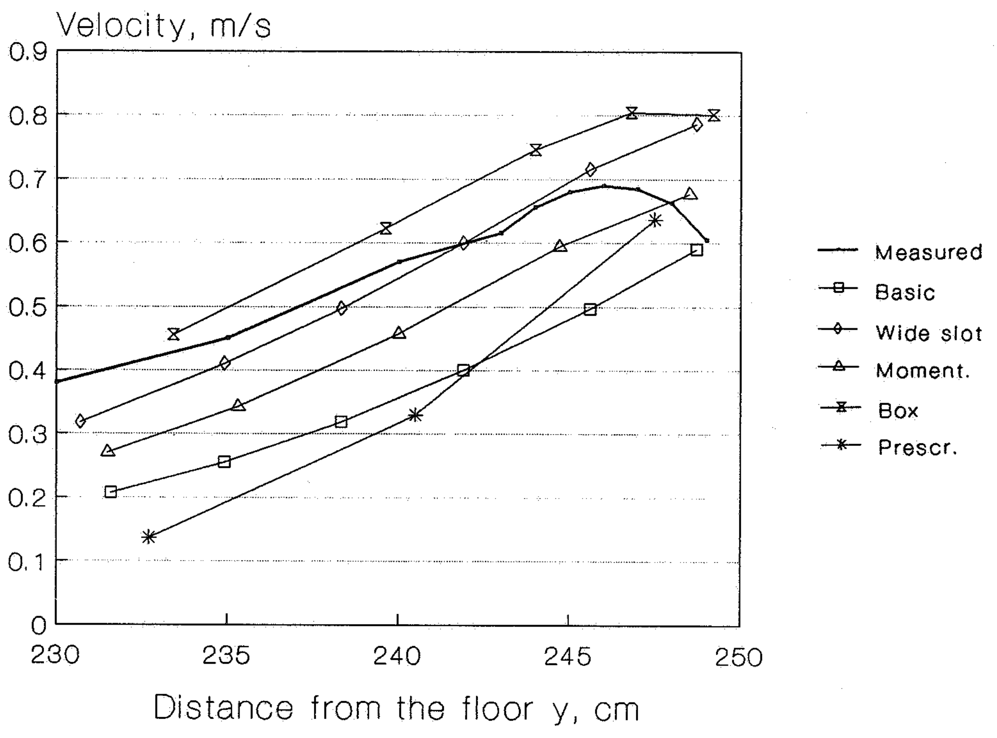

Figure 10. Measured and simulated velocity profile in the vertical direction at 2.2 m from the diffuser in the symmetry plane.

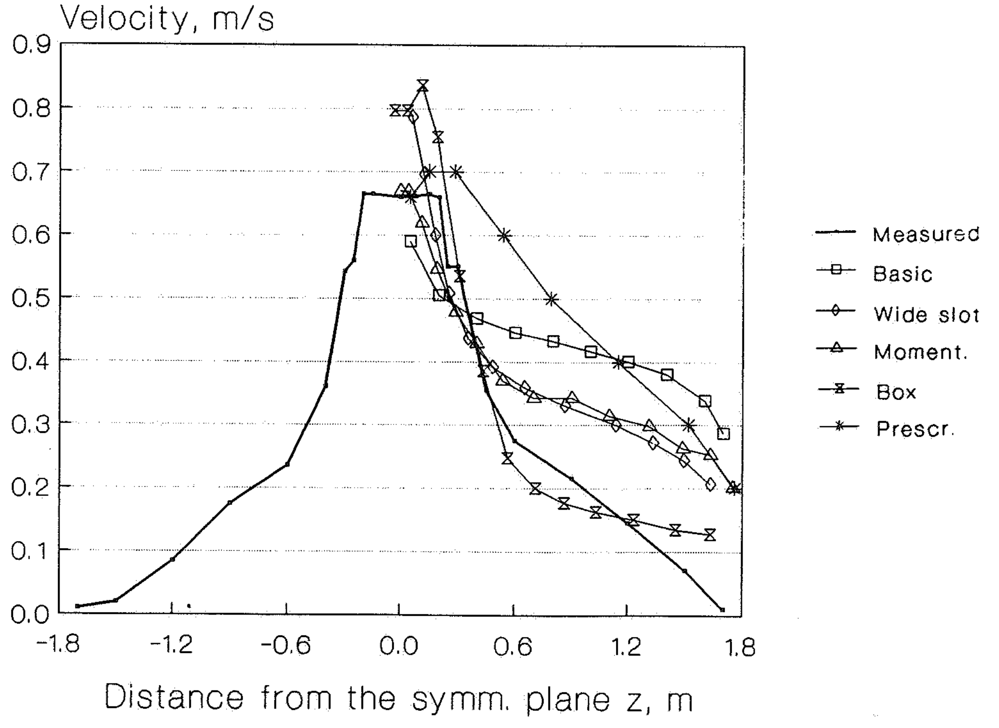

Figure 11. Measured and simulated velocity profiles in the horizontal direction at 2.2 m from the diffuser near the ceiling.

5.4 Momentum model

Velocity decay (fig. 9) is very similar to the wide slot model but there is even more mixing in the early stages of jet development. The jet is thick in the vertical direction already after the stagnation area and the thickness (defined as a distance from the ceiling where velocity reaches half the maximum) remains nearly constant in the wall jet. Velocity profiles at an x-distance of 2.2 m are also similar to the wide slot model but the maximum velocity is somewhat lower.

It seems that mixing should be reduced near the opening. This could be achieved using a smaller diffuser area in the simulation.

Part of the momentum flow of the supply air stream is lost because underpressure is formed in the area between the small nozzles. This phenomenon is well known for perforated plates as a diffuser, see the reference [15]. The underpressure was measured to be around 0.18 Pa which means about a 14 % loss of jet momentum (see appendix 1 for details) and thereby about 7 % decrease in the velocities in the room. The predicted underpressure was only slightly lower; a 10 % loss of momentum was predicted. Apparently the predicted momentum loss will depend on the grid spacing near the diffuser. In this case the length of the first volume cell in x-direction was 57 mm, which corresponds roughly to the length of the mixing zone of the small jets.

5.5 Box model

Velocity decay is again slow, corresponding to the decay of a two-dimensional wall jet. This may be due to erroneous velocity directions given as a boundary condition. Perhaps the detections of velocity directions using smoke was inaccurate. It is inconvenient to use the box model especially if the temperature or concentration profiles also have to be given as boundary conditions as explained in the reference [2]. That is why the prescribed velocity model is favoured over the box model.

5.6 Prescribed velocity model

Velocity decay is close to the measured decay. The profile in the vertical and horizontal directions is not very good: the jet does not spread enough in the vertical direction and spreads too much in the horizontal direction. A peak in the velocity profile on both sides of the symmetry plane can be clearly seen. This is partly due to the prescribed velocity profile, which has a minimum in the symmetry plane.

The way in which the method adapts the velocity field near the ceiling can be seen by comparing figure 12, where the velocity field using the basic model can be seen, and figure 13 where the velocity has been additionally prescribed. It can be clearly seen, how the method gives a kick in the x-direction at a distance of 1 m from the left wall.

The prescribed velocity method seems to be the most promising one to be used in practice when the momentum of the jet is not well known. It requires minimum amount of measured information; in this case only one velocity component was prescribed at 8 computing points. Perhaps the best combination would be to use the momentum method and the prescribed velocity method together.

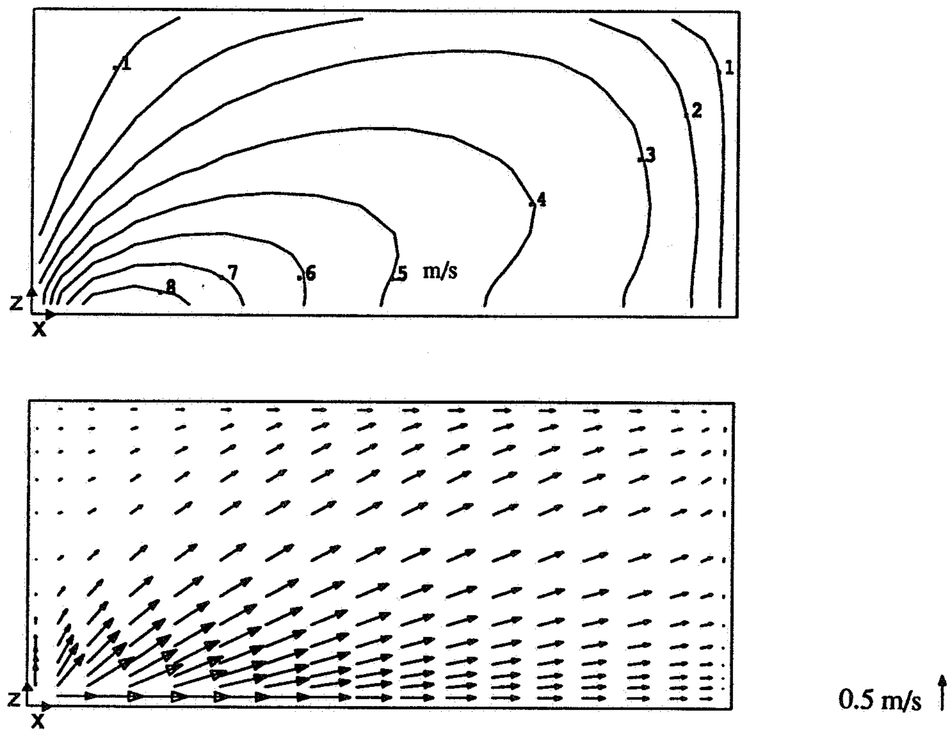

Figure 12. Velocity vectors and air speed contours using the basic model and the Wish code. Plane 25 mm below the ceiling.

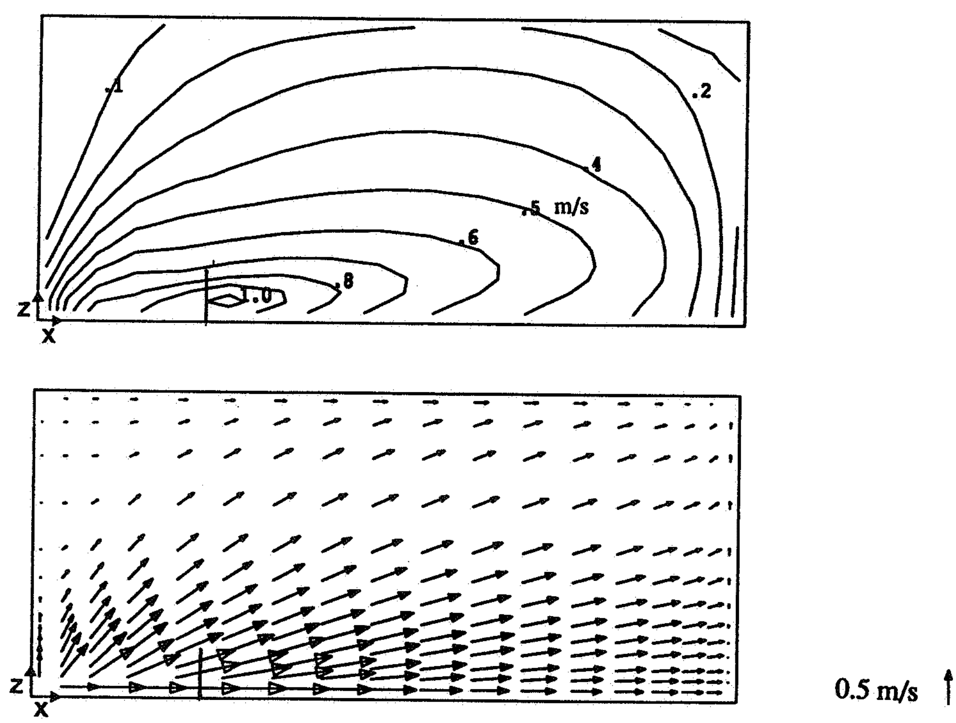

Figure 13. Velocity vectors and air speed contours using the prescribed velocity model and the Wish code. Plane 25 mm below the ceiling. The plane at x = 1 m is shown where the x-direction velocity is prescribed.

CONCLUSIONS

The jet flow including the oblique impingement on the ceiling is a complicated flow to be measured and computed.

The most important property of the supply air opening in the mixing type of ventilation is the momentum flow of the jet. In this particular case the momentum flow was fairly well known and therefore predictions of the decay of the isothermal wall jet was satisfactory with all methods. The wall jet spreads usually too little in the vertical direction and too much in the horizontal direction. The prediction of the maximum velocity in the occupied zone was satisfactory.

Modelling the diffuser by means of a simple opening seems to be a fairly good approximation according also to the measurements if the initial stage of the jet is excluded. In simulations the size of the opening has to be small compared with the room dimensions and this increases the number of grid points and the computing cost.

The numerical method causes mixing in the initial section of the jet, especially in the region where the jet flows diagonally from the inlet towards the ceiling. This unintentional diffusion resembles the diffusion properties of the real diffuser and helps to make better predictions. The problem with the unintentional diffusion is that it depends on the numerical grid and the numerical method, and is therefore not easy to control. Because of this mixing, it may in fact be more difficult to predict a flow from a simple opening where mixing is low to be predicted. This shows that the details of the supply air terminal cannot be described together with room air flow computation. New computational methods such as local grid refinement could be a solution to this problem.

The momentum method for the supply air terminal makes it possible to select the size of the simple opening and can be regarded as a generalization of the simple opening model. In this particular example there was too much mixing in the initial section of the jet. The momentum loss due to underpressure was well predicted in this particular case. It is my opinion that the momentum method is a method that could be used more generally. The method should be studied more systematically.

The prescribed velocity method has the possibility to make the best predictions, perhaps in combination with the momentum method.

Momentum flow or momentum force of the supply air stream is

$$F = \int \rho U^2 \, dA \qquad (1)$$

where

F is the momentum flow (N)
U is the velocity at the nozzle exit in the direction of flow (m/s)
ρ is air density 1.2 kg/m³.

Integration is performed over the supply air perpendicularly to the flow. Thus we should measure the velocity profiles at the opening, or at least in one nozzle, to get the momentum flow. An approximation can be however found making use of the known mass flow

$$q_m = \int \rho U \, dA = \rho U_m A_m = 0.0378 \text{ kg/s} \qquad (2)$$

where

A_m is the area of the all 84 nozzles = 0.0092 m² (diameter 11.8 mm)
U_m is the mean velocity = 3.42 m/s.

We can define the effective velocity U_{eff} in such a way that following equation holds

$$F = U_{eff} \int \rho U \, dA \qquad (3)$$

The lower approximation for the momentum flow will be obtained assuming constant velocity in the opening, which means that the effective velocity is same as the mean velocity. The upper approximation for the momentum, representing the plug flow, will be found using the maximum velocity as an effective velocity. The mean value of maximum velocities in the nozzles was found to be 3.68 m/s in reference [5] (corresponding to an area 0.00855 m²) and lately about 3.85 m/s, which is perhaps more accurate, in reference [16]. The effective velocity 3.68 m/s has been used in the simulations.

Momentum force in the free jet after the combination of small jets is smaller because pressure on the diffuser surface is lower than the ambient pressure. The pressure difference was measured by pressing a crown-formed small tube against the diffuser surface. Its mean value was about 0.18 Pa which means a loss of about 0,028 N in x-direction momentum and about 14 % loss of total momentum. If we take the momentum loss into account and use the maximum velocity 3.85 m/s, the effective velocity should be about 3.3 m/s. The direction of the supply velocity vector also changes from 40° to 48°. This is supported by visual observations.

REFERENCES

1. NIELSEN, P.V.
"Airflow Simulation Techniques - Progress and Trends"
10th AIVC Conference, Espoo, 1989.

2. NIELSEN, P.V
"Representation of Boundary Conditions At Supply Openings"
IEA Annex 20, Internal Report, University of Aalborg,
ISSN 0902-7513 R8902, 1989.

3. LEMAIRE, A.D.
"Testrooms, Identical testrooms"
IEA Annex 20 Internal Report, TNO - Institute of Applied Physics, 1989.

4. NIELSEN. P.V.
"Selection of air terminal device"
IEA Annex 20 Internal Report", University of Aalborg,
ISSN 0902-7513 R8838, 1988.

5. SKOVGAARD, M., HYLDGAARD, C.E., and NIELSEN, P.V
"High and Low Reynolds Number Measurements in a Room with an Impinging Isothermal Jet"
ROOMVENT'90, International Conference on Engineering Aero- and Thermodynamics of Ventilated Rooms, Oslo, 1990.

6. PATANKAR, S.V.
"Numerical Heat Transfer and Fluid Flow."
McGraw-Hill, New York, 1980.

7. LAUNDER, B.E and SPALDING, D.B.
"The numerical computation of turbulent flows."
Computer methods in mechanics and engineering, 3, p. 269 -289, 1974.

8. CREARE INC.
"Fluent, a general purpose computer program for modeling fluid flow, heat transfer and combustion."
Hanover NH, seminar notes, 1987.

9. LEMAIRE, A.D.
"User manual WISH."
TNO - Institute of Applied Physics, Delft, 1989.

10. HUANG, P.G, LAUNDER, B.E and LESCHZINER M.A.
"Discretization of nonlinear convection processes: a broad range comparison of four schemes."
Computer methods in applied mechanics and engineering 48 (1985), North Holland, p. 1 - 24, 1985

11. SKOVGAARD, M. and LEMAIRE, A.D.
"Representation of boundary conditions at supply opening".
Technical note on Research Item no. 1.11, IEA Annex 20 Internal Report, 1990.

12. CHEN, Q., MOSER, A., and SUTER P.,
"Indoor Air Quality and Thermal Comfort under Six Kinds of Air Diffusion."
To appear in ASHRAE Transactions, Vol. 97, part 2, 1991.

13. LAUNDER, B.E and RODI, W.
"The turbulent wall jet"
Prog. Aerospace Sci., vol. 19, pp. 81 - 128, 1981.

14. MURAKAMI, S., KATO, S. and NAKAGAVA,H.
"Numerical Prediction of Horizontal Nonisothermal 3-D Jet in Room Based on k-ε Model."
To appear in ASHRAE Transactions, Vol. 97, part 1, 1991.

15. GRIMTLIN, M.
"Zuluftvertailung in Räumen."
Luft- und Kältetechnik 1970, Nr.5, pp. 247 - 257.

16. EWERT, M. and ZELLER, M.
"Turbulence Parameters at Supply Opening (measurements)"
IEA Annex 20, Internal Report, Rheinisch-Westfälischen Technischen Hochshule, Aachen, 1991.