The Description of Supply Openings in Numerical Models for Room Air Distribution

Nielsen, Peter V.

Publication date:
1991

Document Version
Accepted author manuscript, peer reviewed version

Citation for published version (APA):
Nielsen, P. V. (1991). *The Description of Supply Openings in Numerical Models for Room Air Distribution*. Dept. of Building Technology and Structural Engineering, Aalborg University. Gul Serie No. 9

The Description of Supply Openings in Numerical Models for Room Air Distribution

Peter V. Nielsen

Abstract

The velocity level in a room with jet ventilation is strongly influenced by the supply conditions. It is the momentum flow in the supply jets which controls the air movement in the room and it is therefore very important that the inlet conditions and the numerical method can produce a sufficient description of this momentum flow.

The paper will describe and discuss the following types of boundary conditions at the supply openings.

- Direct description where the boundary conditions are located at the surface of the air terminal device.
- Box method where the boundary conditions are located at the surface of a box in some distance from the air terminal device.
- Prescribed velocity method where some of the variables are described analytically inside a volume in front of the air terminal device and the rest of the variables are predicted by the numerical method.
- Computer generated supply conditions where the flow in the air terminal device is studied as a first step in the predictions of the flow in the whole room.

The paper will summarize experience including new experience from the IEA Annex 20 work.

Key words

Air distribution, air flow, modelling, space environment, ventilation.

The Description of Supply Openings in Numerical Models for Room Air Distribution

Peter V. Nielsen, Ph.D.*

Abstract

The velocity level in a room with jet ventilation is strongly influenced by the supply conditions. It is the momentum flow in the supply jets which controls the air movement in the room and it is therefore very important that the inlet conditions and the numerical method can produce a sufficient description of this momentum flow.

The paper will describe and discuss the following types of boundary conditions at the supply openings.

- Direct description where the boundary conditions are located at the surface of the air terminal device.
- Box method where the boundary conditions are located at the surface of a box in some distance from the air terminal device.
- Prescribed velocity method where some of the variables are described analytically inside a volume in front of the air terminal device and the rest of the variables are predicted by the numerical method.
- Computer generated supply conditions where the flow in the air terminal device is studied as a first step in the predictions of the flow in the whole room.

The paper will summarize experience including new experience from the IEA Annex 20 work.

INTRODUCTION

This paper discuss different possibilities for the description of supply openings in numerical models for room air distribution.

It is typical for all the diffusers that they supply a flow to the room which depends

* Peter V. Nielsen is professor at the University of Aalborg

on small details in the design. This means that a numerical prediction method should be able to handle both small details in dimensions of few millimetres as well as dimensions in metres. This wide range in geometry necessitates many grid points and therefore a large computer, and the paper discusses a number of methods which solve this problem in different ways.

The velocity in a room with jet ventilation is strongly influenced by the supply conditions. Figure 1 shows the decay of the maximum velocity in the flow which runs along the ceiling in a room with two-dimensional recirculating air movement. The velocity level obtained by two different inlet conditions, corresponding to two different supply openings, is retained in the flow along the ceiling and the difference in the velocity level will also be retained in the occupied zone. A sufficient description of the inlet conditions is therefore very important for the prediction of the flow in the whole room.

Figure 1 also shows that the velocity decay below the ceiling corresponds to the conditions in a wall jet, except close to the end wall opposite to the supply opening. This means that the air movement below the ceiling can be expressed by parabolic equations, although the flow as a whole is recirculating and therefore described by elliptic equations. This strong upstream influence in the first part of the flow is the background for different wall jet descriptions of boundary conditions for supply openings discussed in this paper.

It is the momentum flow in the wall jet below the ceiling which controls the air movement in the room. For example, the maximum velocity in the occupied zone is proportional to the inlet velocity times the square root of the supply area which expresses the square root of the supply momentum flow. Therefore it is very important that the inlet conditions and the numerical method can produce a sufficient description of this momentum flow.

Figure 2 shows examples of different air terminal devices. It is obvious that the velocity distribution - and the distribution of other variables as temperature, turbulent kinetic energy and dissipation of turbulent kinetic energy - can be very complicated. The air terminal device type A is an old design. It is mounted in the wall or in the side of the channel and the jet is directed parallel to the ceiling. The horizontal blades are typically adjusted for an upward directed flow and the vertical blades for a horizontal diffusion of the flow. The results are a slightly upward directed jet with two velocity maximums which merge into a three-dimensional wall jet with a moderate initial diffusion.

The air terminal device type B in figure 2 is a new design. It is wall mounted and jets from the openings are directed upward with an angle of 45°. The jets impinge on the ceiling and the resulting flow along the ceiling can be described as a three-dimensional wall jet containing some radial flows. Measurements on devices A and B are given by Nielsen and Möller (1985).

Air terminal device type C is a ceiling mounted diffuser giving a radial wall jet below the ceiling. This wall jet can have different peak velocities in different directions depending on the design of the diffuser. Generally there is a large induction of air inside the diffuser and this air flows through openings in the horizontal surface.

Device D is a ceiling mounted linear slot diffuser which uses the Coanda effect to deflect the jet along the ceiling. The jet can also be adjusted to the opposite direction or vertical downwards. A high entrainment of room air takes place in the deflected flow in the openings and the flow along the ceiling can be described as a two-dimensional wall jet in some distance from the opening as shown by Nielsen and Möller (1988).

The Annex 20 work within the International Engergy Agency (IEA) has studied the use of Computational Fluid Dynamics (CFD) in room air distribution as one of the main purposes. The description of an air terminal device is therefore also in focus and this paper will make references to some of this work.

It was decided to use a diffuser in the IEA experiments and predictions which have elements of the complicated flow described in connection with the diffusers A to D in figure 2. The diffuser consists of 84 nozzles mounted in a rectangular area of the size 0.17 m × 0.71 m, see figure 2E. It is mounted in the wall 0.2 m below the ceiling and all the jets are directed upwards in an angle of 40°. A semi-radial wall jet flows along the ceiling which is typical for the flow from air terminal devices of modern design. The IEA diffuser is described in details by Nielsen (1988).

Direct description of the boundary conditions

This chapter describes different methods which can reduce the need for a high number of grid points at the air terminal device.

The most obvious method is to replace the actual diffuser with a less complicated geometry which supplies the same momentum flow to the room. Computer simulations in the IEA work are made by such simplifications. The 84 nozzles are replaced by a rectangular opening with the same supply area, aspect ratio and velocity direction as shown by Skovgaard and Nielsen (1991). The turbulent kinetic energy k_o at the supply opening is given for a turbulence intensity of 10% and the dissipation ϵ_o in the supply opening is calculated from k_o and from a length scale which is a fraction of the slot height.

Figure 3 shows the prediction in this case as well as the measurements made by the actual diffuser. u_x is the symmetry plane maximum velocity in the wall jet at the distance x from the supply opening and u_o is the supply velocity. x_o is the virtual origin for the jet and a_o is the supply area. It is very characteristic that the symmetry plane velocity level in the wall jet below the ceiling is decreasing too fast compared to the measurements but the predictions show a higher velocity level outside the centre plane. The thickness of the wall jet flow in the centre plane has a smaller growth rate than the measured value. Both the simplified and the real diffuser are supplying the same momentum flow but details in the first part of the flow and the deflection at the ceiling are different because the profiles in the two cases are different. This has also been verified by Heikkinen (1991) who made experiments both with the IEA air terminal device and with a diffuser with the same geometry as the simplified supply opening.

Figure 4 shows the predicted velocity distribution in case of isothermal flow in the IEA work. The velocity level is high compared to the measurements, see Skovgaard et al. (1990). The predicted location of the maximum velocity in the occupied zone is also obtained outside the area at the right end wall where the maximum velocity is measured.

Chen et al. (1991) has worked with four different approaches in the description of the supply opening. He uses the previously mentioned single opening with the same aspect ratio as the IEA diffuser and he uses 12 slots and also 84 openings (one grid point in each opening) as shown in figure 5. Finally he uses a single opening covering the same area as the IEA diffuser with an evenly distributed momentum flow which fulfills the conditions for the actual diffuser combined with the actual level of supply flow.

Figure 6 shows the predicted velocity profiles at $x = 3$ m. The approaches with 1 slot and 12 slots are underestimating the symmetry plane velocity level in the wall jet while 84 openings and the momentum method seem to predict the velocity level in a more sufficient way. It is diffucult to select the best method because the measurements in different test rooms also show some scattering.

Heikkinen (1991) has shown that different CFD-codes influence the results and also that the flow is sligthly grid dependent even at a relative high number of grid points.

It is always appropriate to describe the boundary conditions directly at the opening in special situations where the supply area a_o is large compared to other dimensions in the room. This is ,e.g., the case in clean rooms (laminar flow rooms) and in rooms with wall mounted low induction supply openings and displacement flow.

The box method

The box method has been used with success in the numerical prediction of room air movement as first shown by Nielsen (1976), and Nielsen et al. (1978). This chapter will describe the method in case of two-dimensional flow.

Figure 7 shows the location of the boundary condition. The boundary condition for the supply opening is given at an internal surface a at the distance x_a from the diffuser. The profiles for all the variables ϕ are the universal or the self-preserving profiles for the actual diffuser at the distance x_a. ϕ corresponds to velocity u, temperature T, concentration c, turbulent kinetic energy k and turbulent dissipation ϵ, respectively.

The surface b in figure 7 shows the other boundary in the box method. A parallel flow is assumed ($\partial\phi/\partial y = 0$).

The length of x_a should be sufficient to locate the surface a in an area with a fully developed wall jet. The selection of a large x_a reduces the gradients of the ϕ-values at the surface a and it reduces the solution domain which means a reduction in grid points and a reduction in computation time. The length x_a should on the other hand only be a small fraction of the room length L because the flow in the outer part of the wall jet is strongly influenced by the recirculating flow and it has to be predicted by the elliptic equations.

The height y_b of the surface a should have a certain size compared to the thickness δ of the wall jet to be able to describe the momentum flow and the wall jet. (The thickness δ of the wall jet is defined as the distance to the velocity $u_x/2$). Figure 8 shows that $y_b/\delta = 0.75$ and $y_b/\delta = 1.0$ are sufficient while $y_b/\delta = 0.5$ is too small in the given situation. It is necessary to check the velocity decay in the predictions as well as the continuity in all profiles at the point (x_a, y_b). It is not possible to use a large value of y_b/δ because the real profiles in the room are different from the universal wall jet profiles for $y_b/\delta > 1$ - 1.5.

The velocity profile at the surface a is given as an universal profile u/u_x, see e.g. Rajaratman (1976) and Verhoff (1963). The maximum velocity in the profile u_x at the distance x_a as well as δ is obtained from measurements on the actual diffuser used in the predictions.

The temperature level and the concentration level at surface a are influenced by the values at the surface b due to entrainment. Therefore it is necessary to calculate an energy balance and a mass fraction balance for the volume $x_a \cdot y_b$ in front of the diffuser in each iteration. The profiles are similar to the velocity profile except close to the wall where the values are constant corresponding to the minimum or the maximum value in the profile. The distribution of turbulent kinetic energy k

at the surface a is given from measurements of $\overline{u'^2}/u_x^2$, $\overline{v'^2}/u_x^2$ and $\overline{w'^2}/u_x^2$ where $\overline{u'^2}$, $\overline{v'^2}$ and $\overline{w'^2}$ are the turbulent normal stresses, see for example Nelson (1969).

The turbulent dissipation ϵ and the distribution of the turbulent viscosity μ_t are found from the u, k and $\overline{u'v'}$ profiles as discussed in details by Nielsen (1976).

It is convenient to use the self-semilar profiles as boundary conditions but it is not necessary. Rheinländer (1981) uses a very short distance x_a equal to the length of the constant velocity core of a jet, see figure 9. This version of the box method is convenient when the constant velocity core is well defined and the development of the wall jet covers an essential part of the recirculating flow.

Measured values could also be used as profiles in the box method. This is shown by Knobloch and Mierzwinski (1988) for a two-dimensional jet with an upward trajectory.

A good prediction of the velocity decay u_x in the wall jet as well as a good prediction of the profile will together ensure the correct entrainment from the occupied zone. The velocity in the occupied zone is influenced by entrainment and it is also influenced by the deflected wall jet. It is therefore important that both velocity and profile - or momentum flow - are predicted to a sufficient accuracy.

Figure 10 shows an example of the use of the box method. The supply opening consists of 9 nozzles placed at the distance $H/4$ from the ceiling. The length of the room is three times its height and h/H is 0.011 where h is determined as the height in a slot giving the same supply area as the nozzles. The velocity profiles show that the supplied jets merge in a plane free jet which runs close to the ceiling in its further development and forms a wall jet. The flow around the supply opening is strongly three-dimensional, however, the measurements show that the recirculating flow formed in the greater part of the room is two-dimensional. The measurements were made by Blum (1956).

The calculated velocity profiles in figure 10 are determined as a numerical solution of the two-dimensional flow equations. In the predictions the supply opening is characterized by the plane wall jet profile which it forms at the distance x/H = 1.2. It is seen that the agreement between the measured and the calculated velocities is good. Thus, the deviation of the maximum velocity in the occupied zone is below 1% of the supply velocity. The agreement between the measured and the calculated velocity decay in the wall jet below the ceiling is also good. It is seen, however, that the calculated increase of the jet width barely reaches the measured value.

The use of a wall jet profile as the boundary values in the calculations in figure 10 is a good example of the simplification that can be achieved. If the actual supply openings had been used as boundary conditions the calculations should have been performed by an equation system for three-dimensional flow instead of the system for two-dimensional flow. However, this would result in a great increase in computer storage and computation time.

Prescribed velocity method

The prescribed velocity method has also been successfully used in the numerical prediction of room air movement, first by Gosman et al. (1980). Figure 11 shows the details of the method. The inlet profiles are given as boundary conditions at the diffuser although they are represented only by a few number of grid points. All the variables - except the velocities u and w - are predicted in the volume close to the diffuser as well as in the rest of the room. The velocities u and w are given in the volume in front of the diffuser as the analytical values in a three-dimensional wall jet from the diffuser or they are given as values measured in front of the diffuser.

The volume is surrounded by the surfaces a, b and c located at the distances x_a, y_b and z_c. The considerations on size and location of the surface as well as the number of grid points are the same as in the box method.

The prescribed velocity method is an easy method because it specifies the wall jet velocities in the volume $a \times b \times c$ and predicts the values of v, p, k and ϵ. It may also be necessary to prescribe the temperature distribution and contaminant distribution in the case of non-isothermal flow and flow with contaminant distribution.

The prescribed velocity method has been used in the IEA work by several of the participants. Figure 12 shows the results obtained by Skovgaard and Nielsen (1991). The figure shows the velocity decay in the wall jet below the ceiling and it is seen that the method is an improvement compared to the simulation of momentum flow at the supply opening (figure 3). The thickness of the wall jet flow is also close to the measured value. It should also be expected that good results will be obtained in this case when the velocities in the volume $a \times b \times c$ are known in details from measurements or analytical solutions.

The volume in connection with the predictions shown in figure 12 is located within the boundaries: 0.5 m $\leq$ x $\leq$ 1.5 m, y $\leq$ 0.1 m and -0.5 $\leq$ z $\leq$ 0.5 m. The wall jet description is made from measurements made by Skovgaard et al. (1990). Much effort has been put into the description of the wall jet in the prescribed volume. The flow has a small centre section with parallel flow due to the width of the reattachment line of the impinging jet on the ceiling. The flow outsite that area is semi-radial.

Figure 13 shows the velocity distribution in the IEA work when the prescribed velocity method has been used. The velocity level is decreased compared to the result shown in figure 4 and the prescribed maximum velocity in the occupied zone is 0.20 m/s which is comparable to the measured value of 0.15 m/s.

Heikkinen (1991) has also made a prediction of the flow in the IEA-case using the prescribed velocity method. He used a simplified version where the velocity profiles are only prescribed in a single plane like one of the cases shown by Gosman et al. (1980).

Computer generated inlet supply conditions

The role of computers in research and development work is changing. A research task will normally be an arrangement of experiments on equipment and a parallel theoretical analysis on a computer. The rapid development in the size of computers makes it possible to perform numerical experiments on the computer and to reduce the number of initial experiments on the equipment. This trend is seen in aerodynamics where large computers and wind tunnels are used in parallel in the development of an aircraft. The target is a computer system where Computer-Aided Engineering Design Systems (CAEDS) together with Computer-Aided Design (CAD) support and control the entire product development.

This development can also have an influence on the way air terminal devices are designed and it should be possible to make a computer-aided engineering design system for air distribution within a few years.

The aerodynamic part of a computer-aided engineering design system could work in the following way. The ϕ-profiles upstream in the supply duct are predicted from a short section where outlet conditions are transferred to the inlet in each iteration. The ϕ-profiles are then used as inlet conditions behind the diffuser. The flow profiles in the openings of the diffuser are predicted on the basis of outlet boundary conditions of the zero gradient type ($\partial\phi/\partial x = 0$). The flow profiles in the openings of the diffusers can now be used as inlet profiles for the prediction of the flow in the room and it is furthermore possible to predict values used for a "prescribed ϕ-method" in front of the diffuser. It is also possible to combine the prediction of the ϕ-profiles in the diffuser with the prediction of the initial wall jet inside the *abc* volume. This is especially necessary when the diffuser entrains room air in some areas.

The flow in a room is influenced by a low-turbulent effect and some part of this effect may originate from the diffuser, see Skovgaard et al. (1990). It is only possible to include this effect in a computer generated inlet profile if special boundary conditions or turbulence models are used.

Conclusions

The boundary conditions at the supply opening have a strong influence on the flow in the room and they are therefore very important.

It is difficult to measure the profiles at the diffuser and to use them as boundary conditions. The flow in front of the diffuser will have a complicated structure but it will often develop into a free jet or into a wall jet. Therefore, it is appropriate to use the conditions in the jet as boundary values except in special situations when the supply opening is large compared to the dimensions in the room.

The box method locates the boundary values at some distance from the supply opening. This method saves computer storage and computation time. Self-preserving jet profiles and measurements from the actual diffuser may be used as boundary values.

The prescribed velocity method updates the velocities in a volume close to the supply opening and predicts the turbulent parameters. The method is easy to use and it is only necessary to measure the velocity parameters in the jet from the actual diffuser used in the predictions. It may also be necessary to update the temperature distribution and the contaminant distribution in case of non-isothermal flow and flow with tracer gas distribution, respectively. This method saves some computer storage and computation time.

Computer-aided engineering design systems will make it possible to generate the boundary conditions directly on the computer.

References

Blum, W. 1956. Diplomarbeit (M.Sc. thesis), T.H. Aachen.

Chen, Q, A. Moser and P. Suter. 1991. A Database for Assessing Indoor Airflow, Air Quality, and Draught Risk, Energy Systems Laboratory, Swiss Federal Institute of Technology.

Gosman, A.D., P.V. Nielsen, A. Restivo and J.H. Whitelaw. 1980. The Flow Properties of Rooms With Small Ventilation Openings, Journal of Fluids Engineering, Vol. 102, pp. 316 - 322.

Heikkinen, J. 1991. Modelling af a Supply Air Terminal for Room Air Flow Simulation, 12th AIVC Conference, Ottawa.

Knobloch, B. and S. Mierswinski. 1988. Experimental Choice of Boundary Conditions in Models of Ventilation Air Flow (in Polish), 3rd Seminar on "Application of Fluid Mechanics in Environmental Protection -88", Silesian Technical University, Gliwice, Poland.

Nelson, J.L. 1969. An Experimental Investigation of the Turbulent and Mean Flow Properties of a Plane Two-Dimensional Turbulent Wall Jet. Dissertation, University of Tennessee, Dep. of Chem. Eng.

Nielsen, P.V. 1976. Flow in Air Conditioned Rooms (English translation of Ph.D. thesis from the Technical University of Denmark, 1974), Danfoss A/S, Denmark.

Nielsen, P.V., A. Restivo and J.H. Whitelaw. 1978. The velocity Characteristics of Ventilated Rooms, Journal of Fluids Engineering, Vol. 100, pp. 291 - 298.

Nielsen, P.V. and Å.T.A. Möller, 1985. Measurement of the Three-Dimensional Wall Jet from Different Types of Air Diffusers, World congress on heating, ventilating and air-conditioning, Copenhagen.

Nielsen, P.V. 1988. Selection of Air Terminal Device, Internal Report for IEA Annex 20, University of Aalborg, ISSN 0902-7513 R8838.

Nielsen P.V. and Å.T.A. Möller. 1988. Measurements on Buoyant Jet Flows from a Ceiling-Mounted Slot Diffuser, III Seminar on Application of Fluid Mechanics in Environmental Protection, Silesian Technical University, Gliwice, Poland.

Rajaratnam, N. 1976. Turbulent Jets, Elsevier, Amsterdam.

Rheinländer, J. 1981. Numerische Berechnung von vorwiegend durch die Schwerkraft angetriebenen Raumströmungen, Fortschr. - Ber. VDI-Z, Reihe 7, Nr. 60, ISSN 0341 - 1753.

Skovgaard, M., C.E. Hyldgaard and P.V. Nielsen. 1990. 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.

Skovgaard, M. and P.V. Nielsen. 1991. Modelling Complex Inlet Geometries in CFD - Applied to Air Flow in Ventilated Rooms, 12th AIVC Conference, Ottawa.

Verhoff. 1963. The Two-Dimensional Turbulent Wall Jet with and Without an External Free Stream, Princeton university. Dep. Aeronautical Eng., Rep. No. 626.

Figure Captions

Figure 1. Velocity decay in the flow along the ceiling in a room. Predictions are shown for two different diffusers with the same slot height. $h/H = 0.0015$ and $L/H = 3$ where h, H and L are slot height, room height and room length, respectively.

Figure 2. Different air terminal devices used for room air diffusion. Devices A and B are wall mounted and they generate a three-dimensional wall jet below the ceiling. Devices C and D are ceiling mounted and they produce a radial and a plane wall jet, respectively. Device E is the diffuser selected in the IEA work.

Figure 3. Velocity decay in the three-dimensional wall jet below the ceiling. IEA Annex 20 case. Isothermal flow and 3 air change rates per hour. Skovgaard and Nielsen (1991).

Figure 4. Isothermal velocity distribution in the middle plane of the room with the IEA air terminal device. Velocity scalars are given in m/s and the air change rate is 3h^{-1}, see Skovgaard and Nielsen (1991).

Figure 5. Methods for the diffuser simulation. (a) one slot method; (b) 12 slots method; (c) 84 slots method; (d) momentum method.

Figure 6. Velocity distribution in the wall jet below the ceiling in the case of different boundary conditions for the IEA air terminal device. $x = 3.0$ m.

Figure 7. Location of boundary conditions in the box method.

Figure 8. Velocity decay in a predicted wall jet for different values of y_b/δ. The velocity decay for $y_b/\delta = 1.0$ corresponds to measurements in the given situation. $h/H = 0.003$ and $L/H = 1.9$.

Figure 9. Boundary conditions located close to the constant velocity core of a jet.

Figure 10. Measurements and prediction of velocity profiles in a room with nine supply nozzles placed at a distance from the ceiling. The upper figure shows a vertical section in the middle plane and the lower figure shows a horizontal section at the heights listed in the upper figure. The calculated velocity u_t is the total velocity $(u^2 + v^2)^{0.5}$. $L/H = 3.0$, $W/H = 1.0$ and $h/H = 0.011$. L, H and W are the length, height and width of the room, respectively.

Figure 11. Prescribed velocity field close to the supply opening.

Figure 12. Velocity decay in the three-dimensional wall jet. The prescribed velo-

city method is used on the IEA air terminal device. The flow is isothermal and the air change rate is $3h^{-1}$, see Skovgaard and Nielsen (1991).

Figure 13. Isothermal velocity distribution in the middle plane of the room with the IEA air terminal device. Velocity scalars are given in m/s and the air change rate is $3h^{-1}$, see Skovgaard and Nielsen (1991).

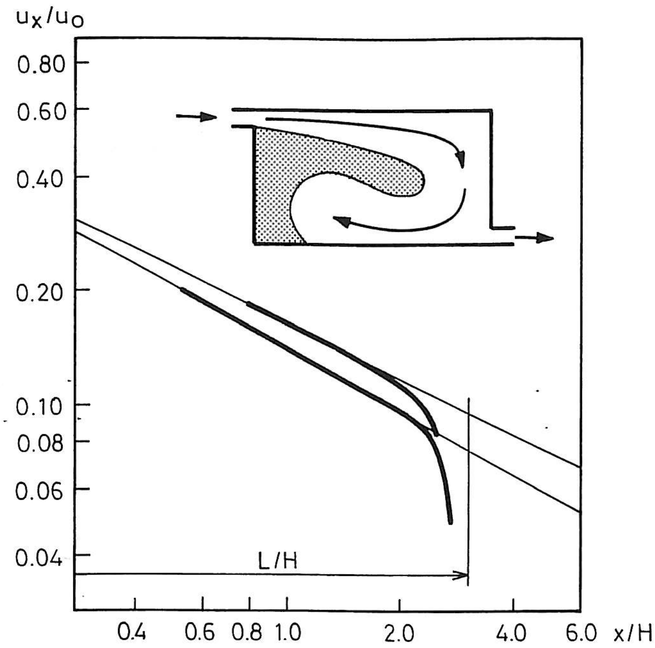

Figure 1. Velocity decay in the flow along the ceiling in a room. Predictions are shown for two different diffusers with the same slot height. $h/H = 0.0015$ and $L/H = 3$ where h, H and L are slot height, room height and room length, respectively.

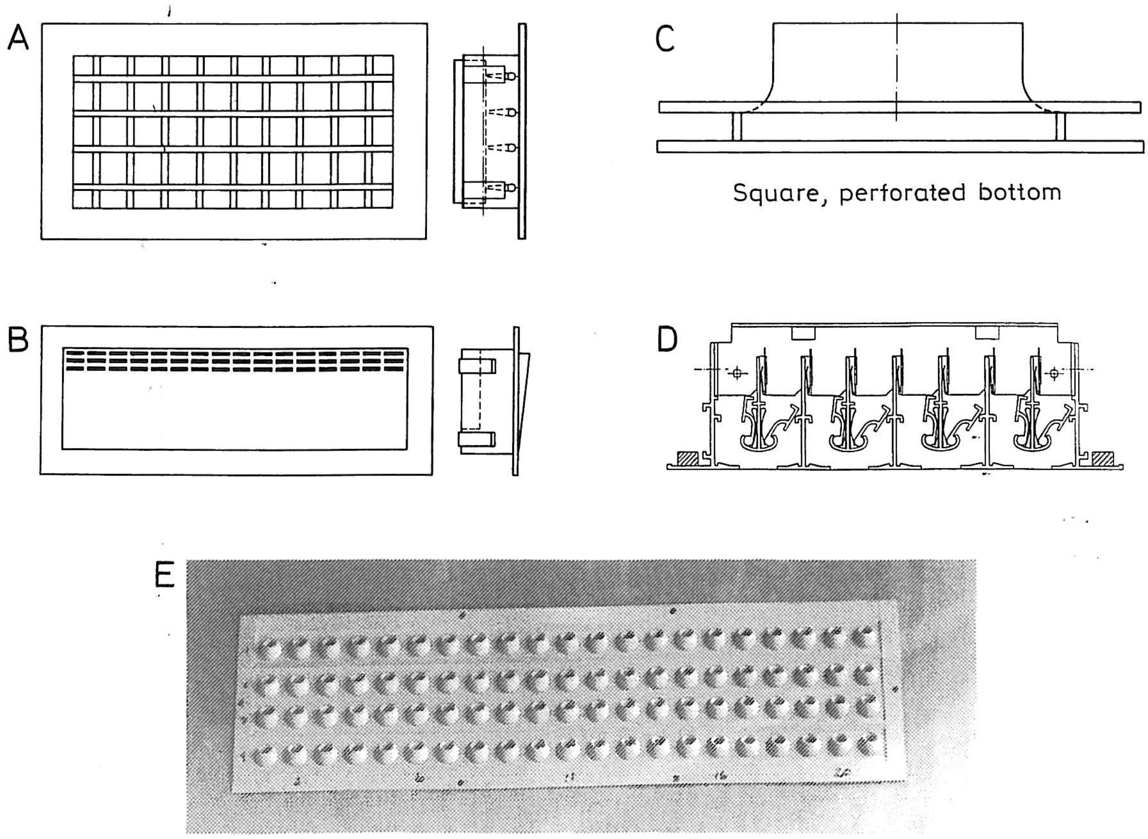

Figure 2. Different air terminal devices used for room air diffusion. Devices A and B are wall mounted and they generate a three-dimensional wall jet below the ceiling. Devices C and D are ceiling mounted and they produce a radial and a plane wall jet, respectively. Device E is the diffuser selected in the IEA work.

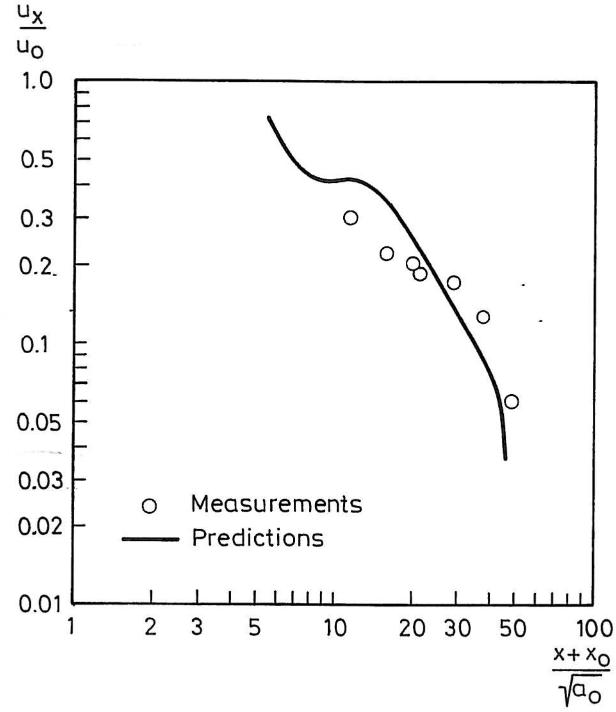

Figure 3. Velocity decay in the three-dimensional wall jet below the ceiling. IEA Annex 20 case. Isothermal flow and 3 air change rates per hour. Skovgaard and Nielsen (1991).

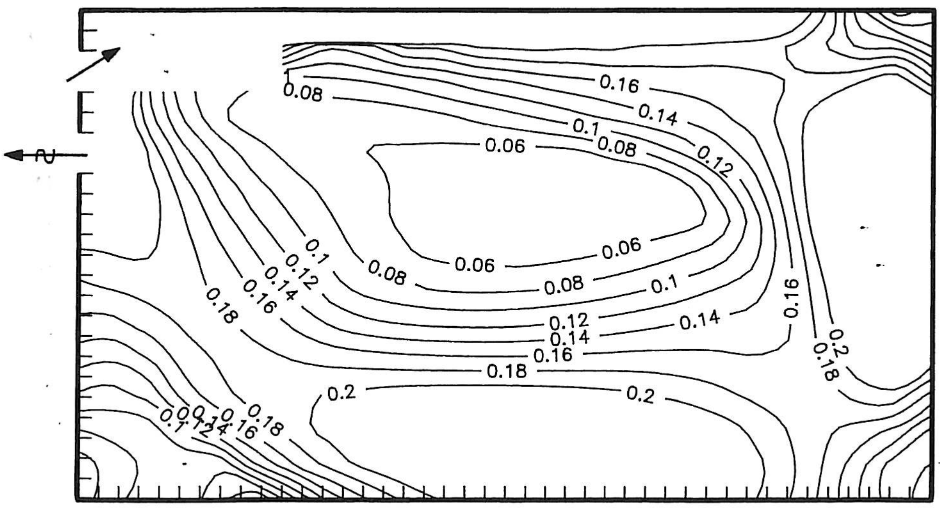

Figure 4. Isothermal velocity distribution in the middle plane of the room with the IEA air terminal device. Velocity scalars are given in m/s and the air change rate is $3h^{-1}$, see Skovgaard and Nielsen (1991).

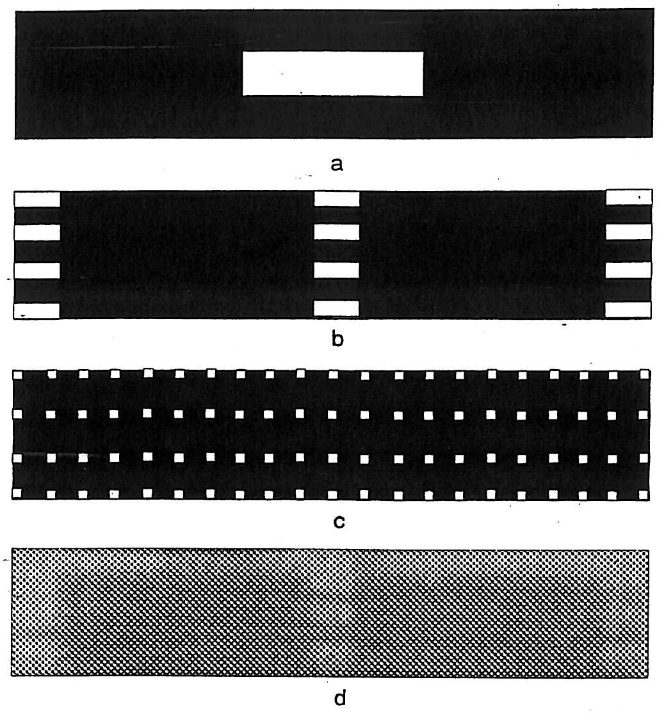

Figure 5. Methods for the diffuser simulation. (a) one slot method; (b) 12 slots method; (c) 84 slots method; (d) momentum method.

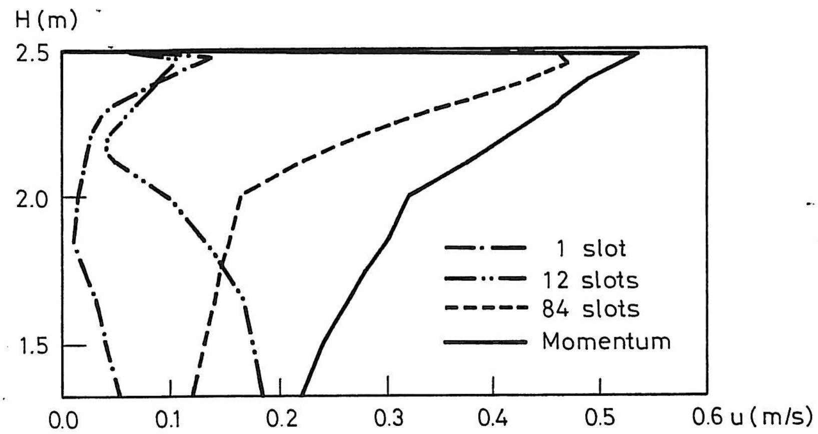

Figure 6. Velocity distribution in the wall jet below the ceiling in the case of different boundary conditions for the IEA air terminal device. $x = 3.0$ m.

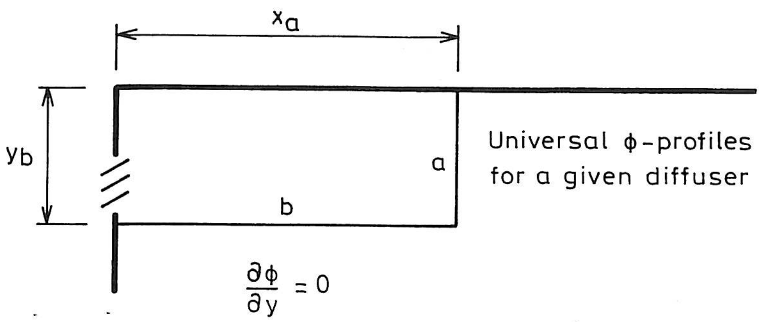

Figure 7. Location of boundary conditions in the box method.

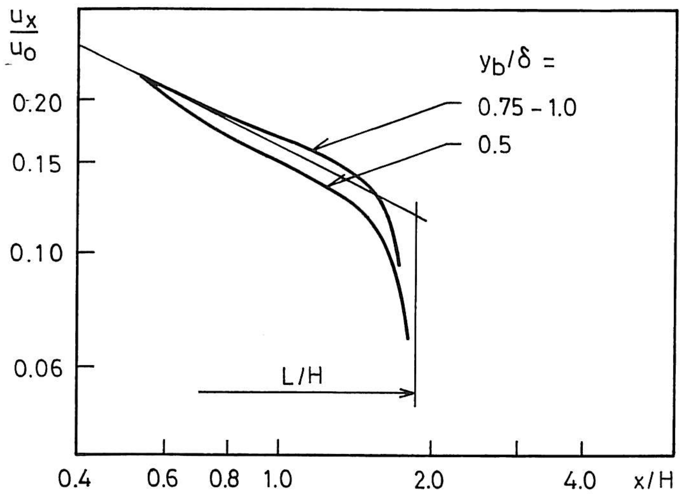

Figure 8. Velocity decay in a predicted wall jet for different values of y_b/δ. The velocity decay for $y_b/\delta = 1.0$ corresponds to measurements in the given situation. $h/H = 0.003$ and $L/H = 1.9$.

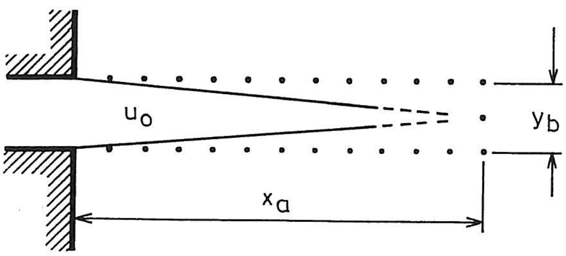

Figure 9. Boundary conditions located close to the constant velocity core of a jet.

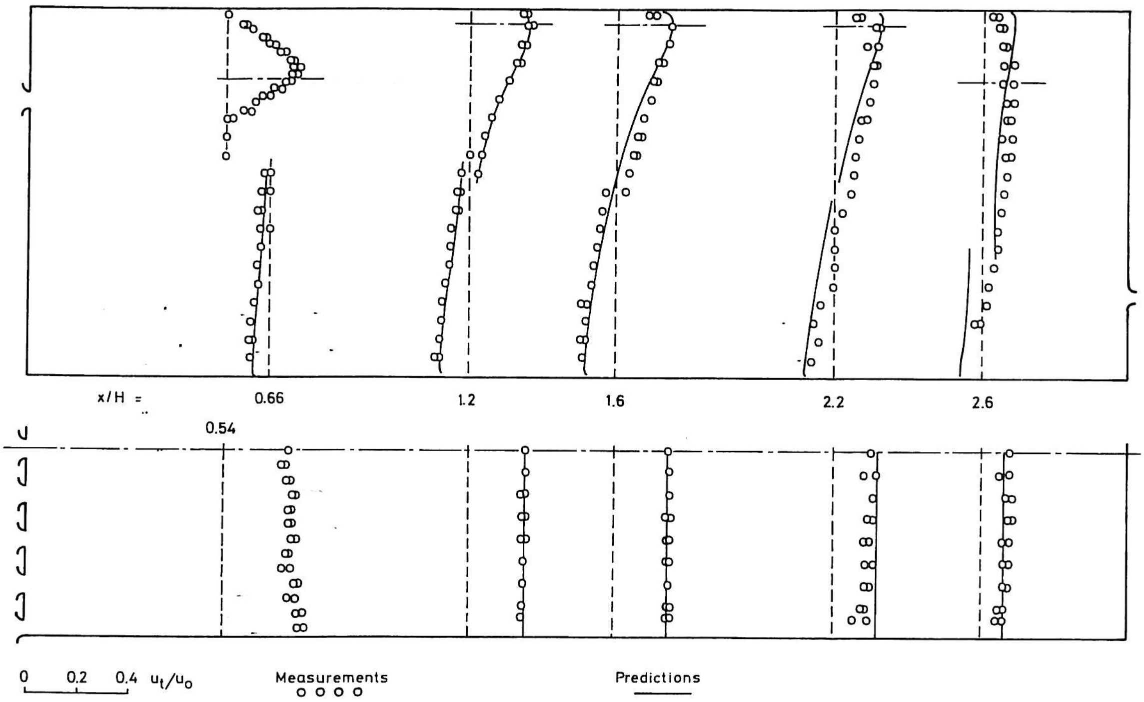

Figure 10. Measurements and prediction of velocity profiles in a room with nine supply nozzles placed at a distance from the ceiling. The upper figure shows a vertical section in the middle plane and the lower figure shows a horizontal section at the heights listed in the upper figure. The calculated velocity u_t is the total velocity $(u_t^2 + v^2)^{0.5}$. $L/H = 3.0$, $W/H = 1.0$ and $h/H = 0.011$. L, H and W are the length, height and width of the room, respectively.

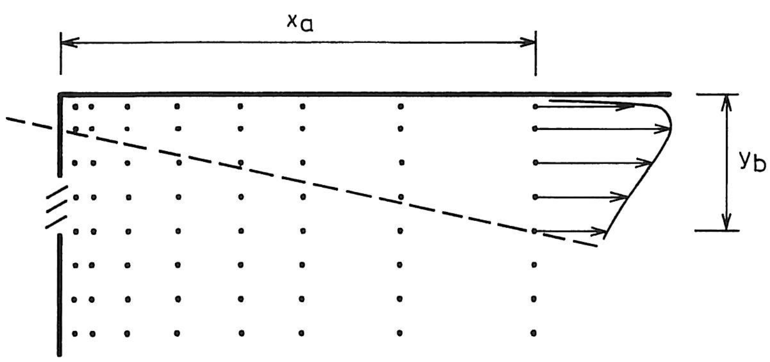

Figure 11. Prescribed velocity field close to the supply opening.

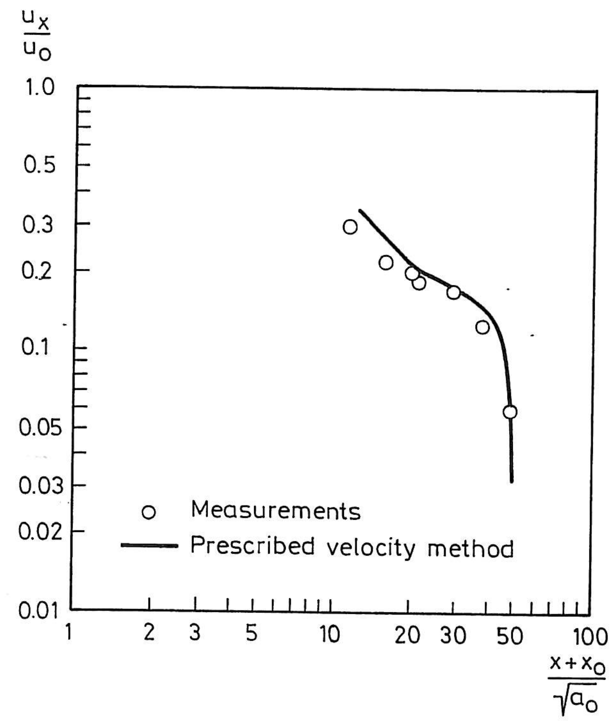

Figure 12. Velocity decay in the three-dimensional wall jet. The prescribed velocity method is used on the IEA air terminal device. The flow is isothermal and the air change rate is $3h^{-1}$, see Skovgaard and Nielsen (1991).

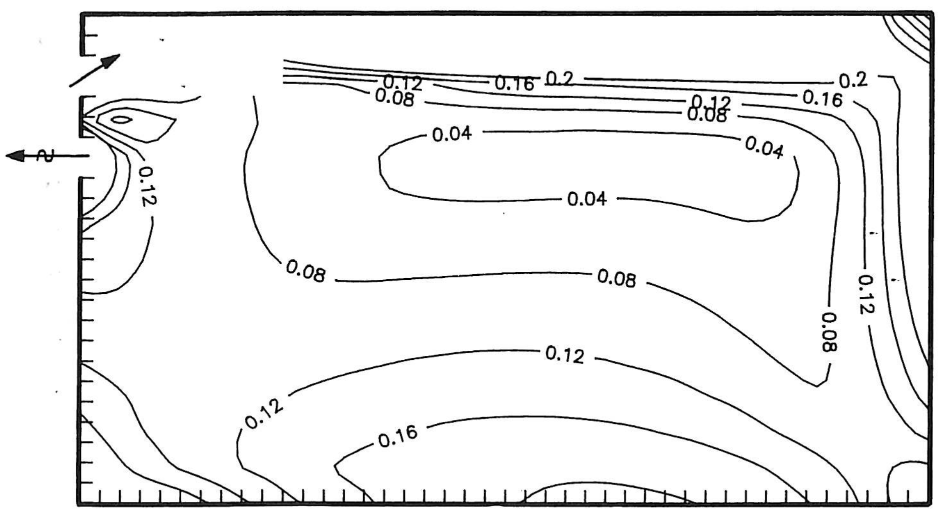

Figure 13. Isothermal velocity distribution in the middle plane of the room with the IEA air terminal device. Velocity scalars are given in m/s and the air change rate is $3h^{-1}$, see Skovgaard and Nielsen (1991).