Representation of Boundary Conditions at Supply Openings

Nielsen, Peter V.

Publication date:
1989

Document Version
Publisher's PDF, also known as Version of record

Citation for published version (APA):
Nielsen, P. V. (1989). *Representation of Boundary Conditions at Supply Openings*. Dept. of Building Technology and Structural Engineering, Aalborg University. Gul Serie Vol. R8902 No. 4

INSTITUTTET FOR BYGNINGSTEKNIK
INSTITUTE OF BUILDING TECHNOLOGY AND STRUCTURAL ENGINEERING
AALBORG UNIVERSITETSCENTER • AUC • AALBORG • DANMARK

International Energy Agency, Energy Conservation in Buildings and Community Systems, Annex 20: Air Flow Pattern Within Buildings

PETER V. NIELSEN

REPRESENTATION OF BOUNDARY CONDITIONS AT SUPPLY OPENINGS

FEBRUARY 1989 ISSN 0902-7513 R8902

RESEARCH ITEM NO. 1.11
"REPRESENTATION OF BOUNDARY CONDITIONS AT SUPPLY OPENINGS"

Peter V. Nielsen, University of Aalborg, Denmark

INTRODUCTION

The velocity level 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, see [1].

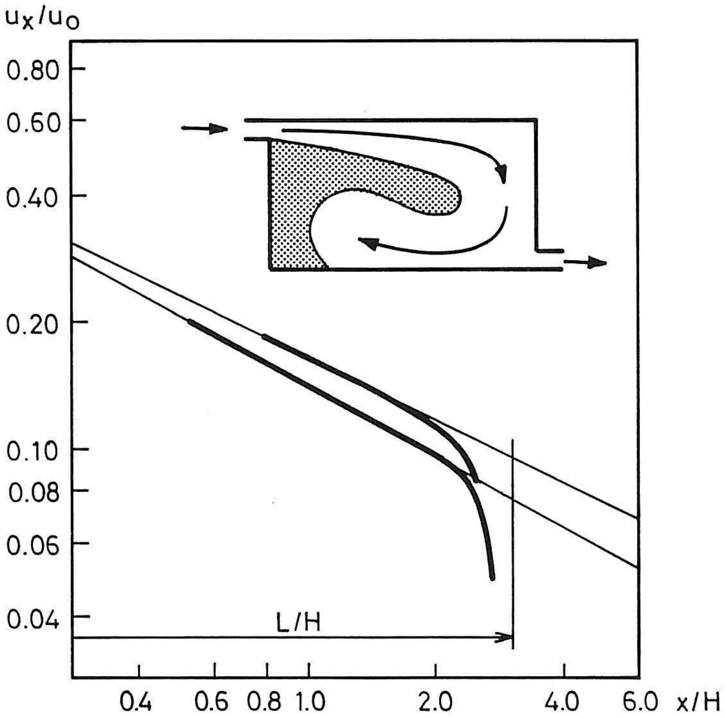

Figure 1. Velocity decay in the flow along the ceiling in a room. Predictions are shown for two different inlet conditions. h/H = 0.0015, L/H = 3.

The figure shows that the velocity level obtained by two different inlet conditions, corresponding to two different supply openings, is retained in the flow along the celing, 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. The semi-parabolic flow in the room is also the background for some of the simplified models, see [2].

It is the momentum flow in the wall jet below the ceiling which controls the air movement in the room. The maximum velocity in the occupied zone is for example proportional to the inlet velocity times square root of the supply area which expresses the square root of the supply momentum flow, see [2]. It is therefore very important that the inlet conditions and the numerical method can produce a sufficient description of this momentum flow.

The momentum flow in the supply opening is often higher than the momentum flow in the wall jet. This is for example shown for a slot located at a distance from the ceiling, see [3] and [4]. It is therefore important to test air terminal devices at a distance to the ceiling which corresponds to the situation in the rooms where they shall be used later, or to use measurements in the boundary conditions which are obtained for a location of the diffuser which corresponds to the situation in the predictions.

BOUNDARY CONDITIONS AT THE AIR TERMINAL DEVICE

Figure 2 shows an example of a velocity profile at the opening of an air terminal device. It is obvious that the velocity distribution can be very complicated with variation both in level and direction and therefore it will be difficult to measure in detail. Although it is not the case for the HESCO diffuser, some ceiling mounted diffusers have areas with return flow for induction inside the diffuser which is a further complication.

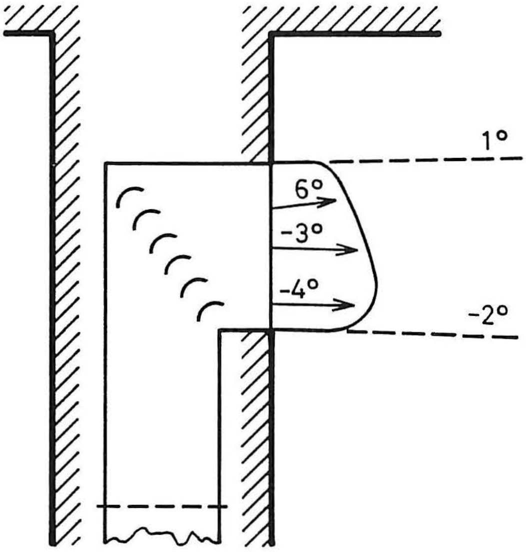

Figure 2. Typical wall mounted grille.

It is difficult to measure the distribution of turbulent kinetic energy k and it is especially difficult to obtain a distribution of turbulent dissipation ε in front of the air terminal device as the one shown in figure 2.

The conditions at the supply opening have strong influence on the flow in the room and there are problems involved in measuring directly at the opening. This combination makes it interesting to give the boundary conditions as wall jet or free jet profiles at a distance from the opening.

The procedure will also save grid points and computation time as described in the following chapters.

It is 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 for example 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, see [1] and [4]. This chapter will describe the method in case of two-dimensional flow.

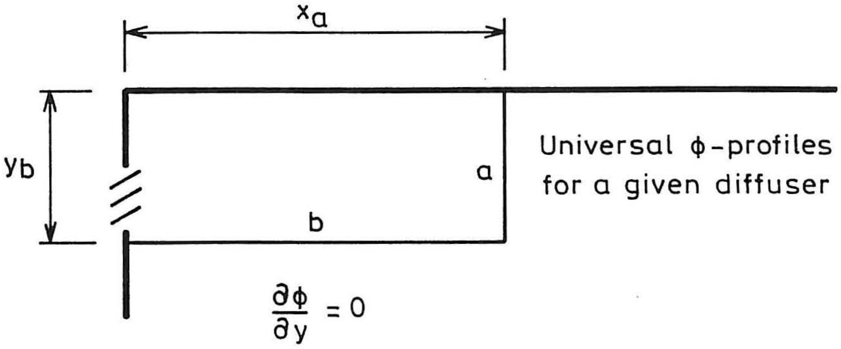

Figure 3. Location of boundary conditions in the box method.

Figure 3 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 3 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 all in all 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 equation.

The height y_b of the surface a should be adequate for the momentum flow to be established in the wall jet. Figure 4 shows that $y_b/\delta = 0.75$ and $y_b/\delta = 1.0$ show good results 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 $y_b/\delta > 1 - 1.5$.

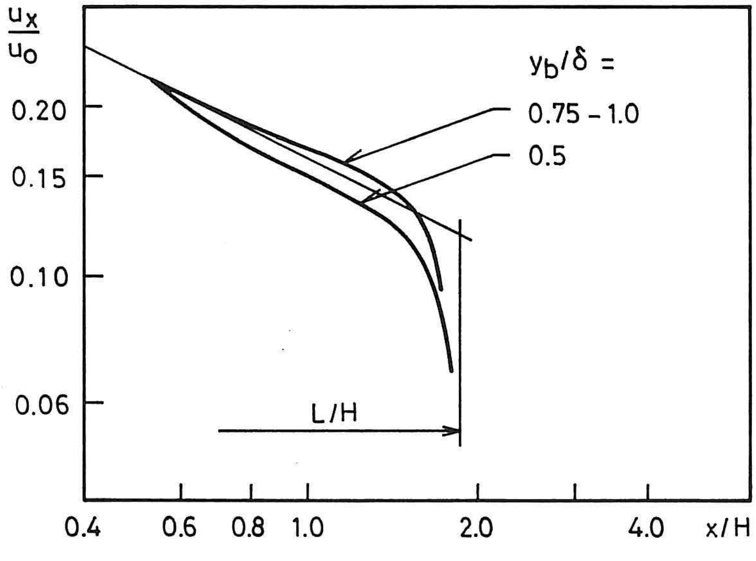

Figure 4. Velocity decay in a predicted wall jet for different values of y_b/δ. The velocity decay for y_b/δ = 1.0 corresponds to measurements in the given situation. $h/H = 0.003$, $L/H = 1.9$ and $Re = 1400$.

The velocity profile at the surface a is given as an universal profile u/u_x, see for example Rajaratnam [5] and Verhoff [6]. The maximum velocity in the profile u_x at the distance x_a is obtained from measurements on the actual diffuser used in the predictions or from the K-value of the diffuser. δ and x_o are given from measurements.

The temperature level and the concentration level at surface a are influenced by the values at the surface b due to entrainment. It is therefore 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. K_T is given from measurements. 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 e.g. Nelson [7].
The turbulent dissipation ε and the distribution of the turbulent viscosity μ_t are found from the u, k and $\overline{u'v'}$ profiles, see [6] and [7]. The turbulent viscosity may be determined from the Boussinesq hypothesis

$$-\rho\ \overline{u'v'} = \mu_t \frac{\partial u}{\partial y} \tag{1}$$

where ρ is the density and $\overline{u'v'}$ the turbulent shear stress. The equation assumes that there is a vanishing shear stress at the velocity maximum. This is not the case in asymmetrical jets such as wall jets and μ_t will follow the dotted line in figure 5 when it is calculated from equation (1).

The turbulence length scale ℓ is determined from the μ_t distribution and the k distribution according to the following equation

$$\ell = \mu_t / c_\mu k^{0.5} \rho \tag{2}$$

where c_μ is a constant or a variable in case of low turbulent flow. The dotted curve in figure 6 shows this distribution of the length scale ℓ.

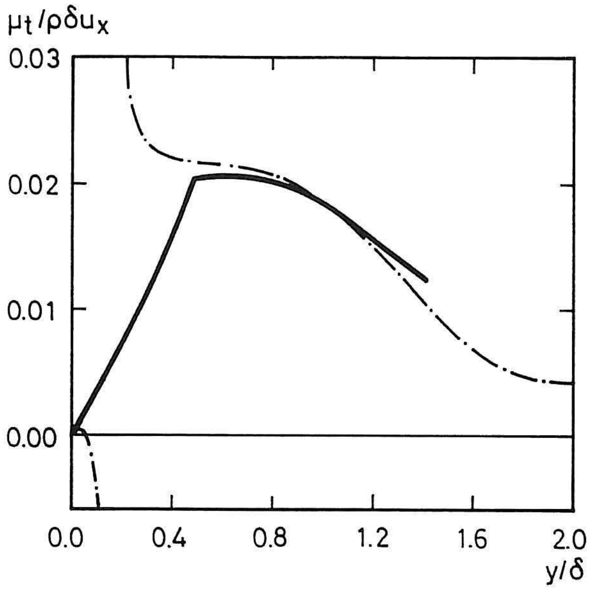

Figure 5. Distribution of turbulent viscosity in a two-dimensional wall jet.

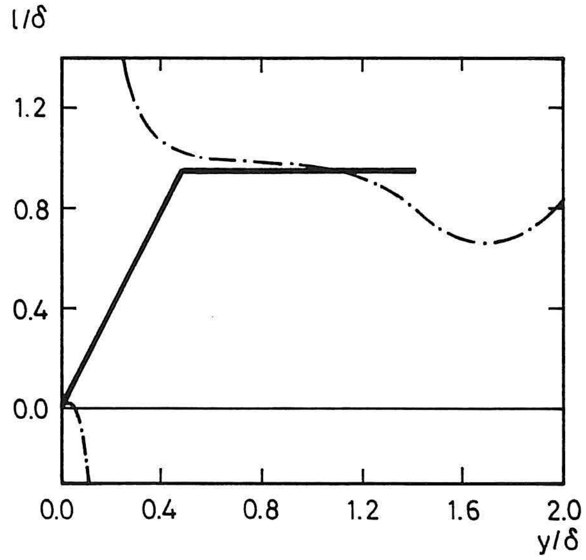

Figure 6. Distribution of turbulent length scale in a two-dimensional wall jet.

The obtained values for μ_t and ℓ cannot be used as boundary values since they assume conditions which are disregarded in the turbulent model. New values are based on the length scale shown as an uninterrupted line in figure 6. This length scale is proportional to the distance from the wall up to $y/\delta = 0.5$ and it has a value close to the level found according to equation (1) for $y/\delta > 0.5$. If this length scale is used in equation (2) it is possible to obtain the μ_t distribution shown as an uninterrupted line in figure 5.

The new length scale ℓ is used to determine the distribution of dissipation along the surface a according to the equation

$$\varepsilon = k^{3/2}/\ell \tag{3}$$

Other suggestions for length scale distribution are shown in [8] and [9].

It is convenient to use the self-similar profiles as boundary conditions but it is not necessary. Rheinländer, [10], uses a very short distance x_a equal to the length of the constant velocity core of a jet, see figure 7. 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.

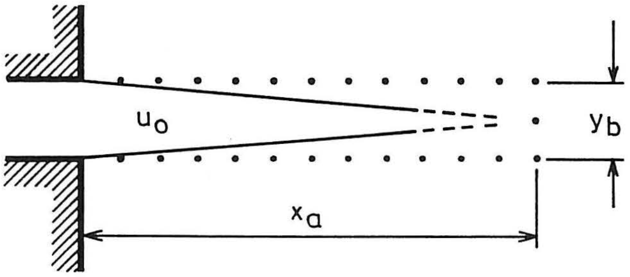

Figure 7. Boundary conditions located close to the constant velocity core of a jet.

Measured values could also be used as profiles in the box method. This is shown in reference [11] for a two-dimensional jet with an upward trajectory.

PRESCRIBED VELOCITY METHOD

The prescribed velocity method has also been successfully used in the numerical prediction of room air movement, see [8]. Figure 8 shows the details of the method. The inlet profiles are given as boundary conditions at the supply opening although they are represented only by a low number of grid points. It is important that the volume flow, the energy supply and the contaminant supply are correctly given at the boundary.

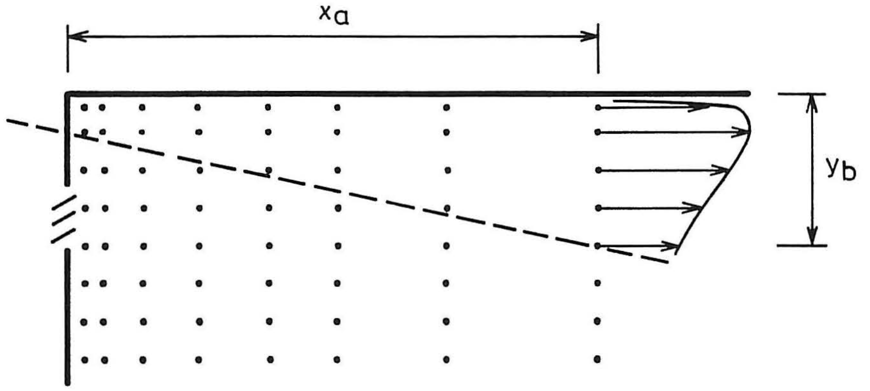

Figure 8. Prescribed velocity field close to the supply opening.

All the variables ϕ are predicted in the volume close to the diffuser as well as in the rest of the room. The velocities u and w are updated in the small volume in front of the diffuser after each iteration. They are analytical values for a three-dimensional wall jet from the given diffuser, or they are given the values measured in front of the diffuser.

The volume is surrounded of 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 surfaces 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 at surfaces a and c combined with the predicted values of p, k and ε.

The flow from the diffuser used in the International Energy Agency Annex 20 work can be compared with an oblique impinging jet. Belatos, [12], shows equations for the u and w velocity in the radial jet below the ceiling which may be useful as a structure in the prescribed velocity method. It will be necessary to update the temperature distribution and the concentration distribution in the volume a **b** c if the flow is non-isothermal, or if it is a flow with concentration distribution. See also reference [13] for a distribution of the flow from the Annex 20 diffuser.

COMPUTER GENERATED INLET BOUNDARY 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 performe 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 distribution systems 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 systems could work in a way which is explained in connection with figure 2.

The ϕ-profiles 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 for the diffuser and they are located at the dotted line in figure 2. The flow profiles in 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 diffuser can now be used as inlet profiles for the prediction of the flow in the room, and it is further possible to use the "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 a b c 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 [13]. It is only possible to include the effect in a computer generated inlet profile if special boundary conditions or turbulence models are used.

It is not difficult to predict the details in the flow close to the diffuser, assuming that the supply profiles are known and a sufficient number of grid points is available. Figure 9 shows the velocity decay in a two-dimensional free jet from a slot. The prediction shows a correct representation of the constant velocity core and it also shows a velocity decay which corresponds to constant momentum flow in the free jet ($u_x/u_o \sim 1/\sqrt{x}$). u_o, u_x and h is supply velocity, maximum velocity at the distance x and slot height, respectively.

Figure 10 shows the velocity profile at different distances from the slot. It is seen that the profiles develop into a self-preserving profile at a distance of $x/h \sim 10$. The predictions in figures 9 and 10 are made in a grid of size 15 × 15.

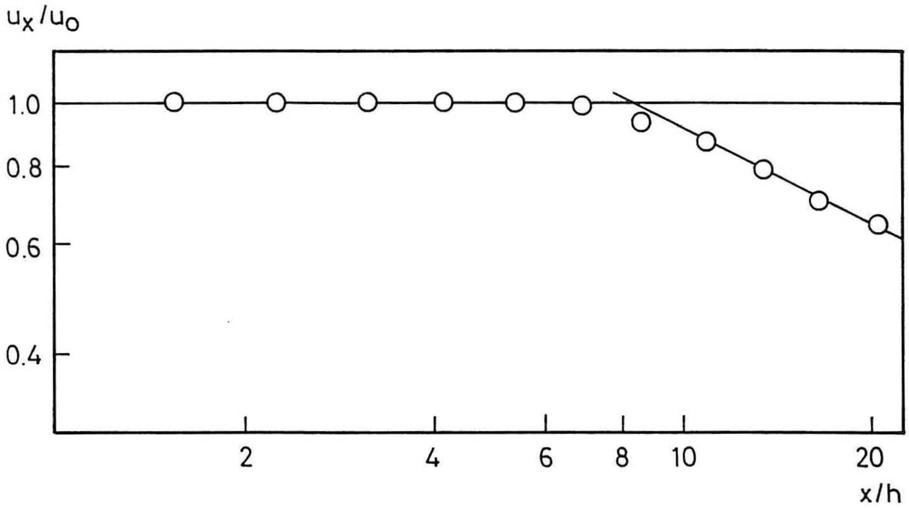

Figure 9. Velocity u_x/u_o versus distance x/h in a two-dimensional free jet. Re = 26000.

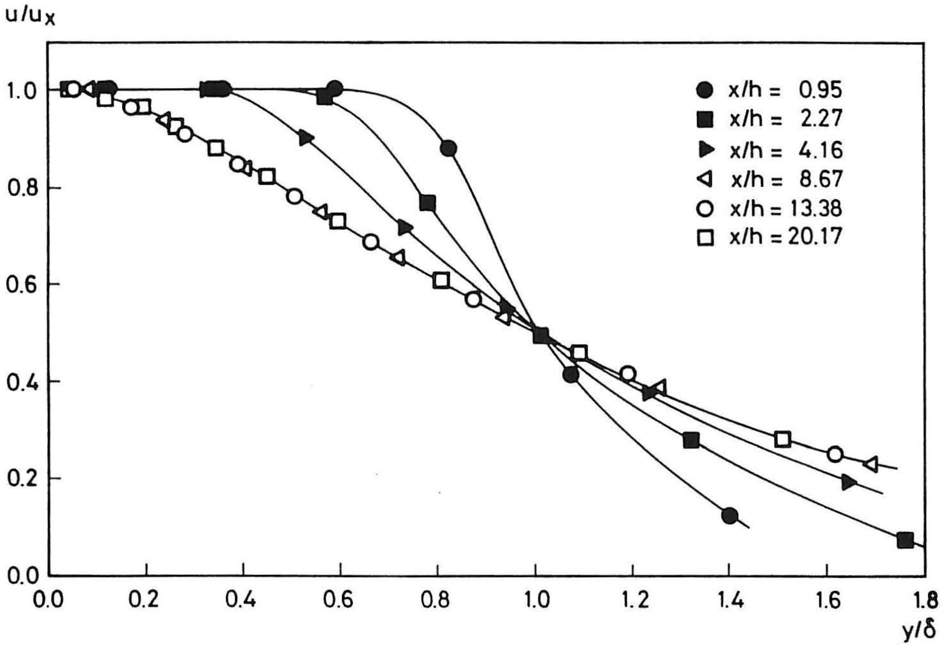

Figure 10. Velocity profiles in a two-dimensional free jet. Re = 26000.

CONCLUSIONS

The boundary conditions at the supply opening have 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 a wall jet. It is therefore 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.

A few years of development work will make it possible to generate the boundary conditions directly on the computer, but this method is beyond the scope of the IEA Annex 20 work.

REFERENCES

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

[2] Nielsen, P.V., Simplified Models for Room Air Distribution, Internal Report for IEA Annex 20, University of Aalborg, 1988, ISSN 0902-7513 R8831.

[3] McRee, D.I., and H.L. Moses, The Effect of Aspect Ratio and Offset on Nozzle Flow and Jet Reattachment, Advances in Fluidics, The 1967 Fluidics Symposium, ASME, 1967.

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

[5] Rajaratnam, N., Turbulent Jets, Elsevier, Amsterdam, 1976.

[6] Verhoff, The Two-Dimensional Turbulent Wall Jet with and without an External Free Stream, Princeton University. Dep. Aeronautical Eng., Rep. No. 626, May 1963.

[7] Nelson, J.L., An Experimental Investigation of the Turbulent and Mean Flow Proporties of a Plane Two-Dimensional Turbulent Wall Jet. Dissertation, University of Tennesee, Dep. of Chem. Eng., 1969.

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

[9] Scholz, R., and B. Hanel, Computergestützte Berechnung der Raumluftströmung, Reihe Luft- und Kältetechnik, VEB Verlag Technik, Berlin, 1988.

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

[11] Knobloch, B., and S. Mierzwinski, 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, 1988.

[12] Beltaos, S., Oblique Impingement of Circular Turbulent Jets, Journal of Hydraulic Research 14, No. 1, 1976.

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

Appendix A

IEA Annex 20 Air Flow Patterns within Buildings, subtask 1

Research Item Description No. 1.11

Title: Representation of boundary conditions at supply opening

Coordinator/PI: Peter V. Nielsen Country: DK

Institution University of Aalborg

Contributing countries:

Project phase: Main Planned effort: person-months

Start: Jan. 1989 End: Febr. 1989

Objectives: To get the necessary boundary conditions around the air terminal device for simulation exercises.

To get the necessary parameters for simplified models. (This is described in: Research item no. 1.2).

To have recommendations on incorporating the boundary conditions in the numerical code. Different suggestions are given.

Method:

Related Annex-20 Research Items:

Input from 1.2

Output from Participants of subtask 1.

Milestones with goals and products:	Date:
Report	Febr. 1989