#3057

H3057

10788.

NUMERICAL PREDICTION OF AIR DISTRIBUTION IN ROOMS - STATUS AND POTENTIALS

Peter V. Nielsen, University of Aalborg

1. INTRODUCTION

The purpose of an air distribution system in a ventilated room is to supply fresh air, remove heat load or supply heating, and create a pleasant and uniform climate in the occupied zone. A pleasant climate is in this context defined as a fairly low air velocity, small velocity and temperature gradients through the room and a low concentration of polluting substances, if any.

The air distribution system often functions as a jet ventilation where the supply jets form a plane or three-dimensional recirculating flow in the room. The air distribution system is designed so that there is a good entrainment in the room and a limited air velocity in the occupied zone. It has been shown that a numerical prediction of the recirculating flow in a room is very suitable to form the basis of the design. This method takes account of many different aspects such as supply jet geometry, geometry of the room, size and the distribution of heat sources and sources of emission, if any.

2. NUMERICAL METHOD

The numerical method is based on computer-solved flow equations, see e.g. [1], [2], [3], [4], [5] and [6]. Three momentum equations (Navier- Stokes equations) and the equation of continuity describe the flow. The turbulence is described by a transport equation for turbulent kinetic energy and an equation for dissipation of turbulent kinetic energy. Launder et al. [7] has developed this two-equation turbulence model to its present form which is used here. Generally, the model is frequently used for calculating different forms of recirculating flow, see e.g. Pope & Whitelaw [8].

For an ordinary case of thermal flow the equation system is expanded by the energy equation, and in cases where the distribution of polluting gases or particles is to be calculated the equation system is expanded by a transport equation for these values.

The area is divided into a number of grid points, and finite-difference equations combining the values in the individual points are established. In areas where the gradients of the different variables are large the points are to be closely spaced while there may be more space between the points in areas with smaller gradients. The finite-difference equations can be solved by line-by-line iteration as described in references [5] and [9].

In order to save points the flow close to wall surfaces is described by general boundary layer profiles, see e.g. [5].

2.1 Simulation of Supply Opening

For the numerical calculation of air flow in a room it is necessary to use special boundary values around a supply opening if the method is to have any practical value. If you consider a section in a typical modern slot diffuser as shown in figure 1 it is seen that it contains many complicated details.

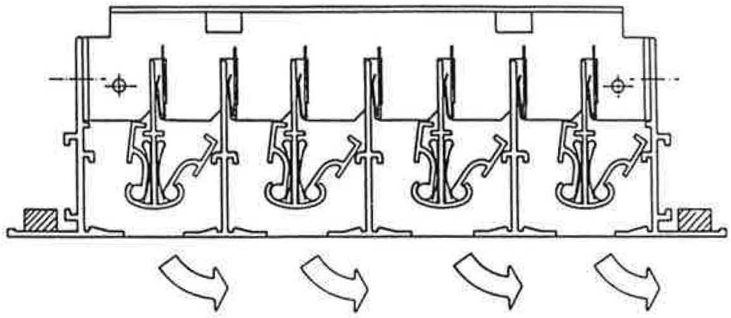

Figure 1. Section through a ceiling mounted slot diffuser.

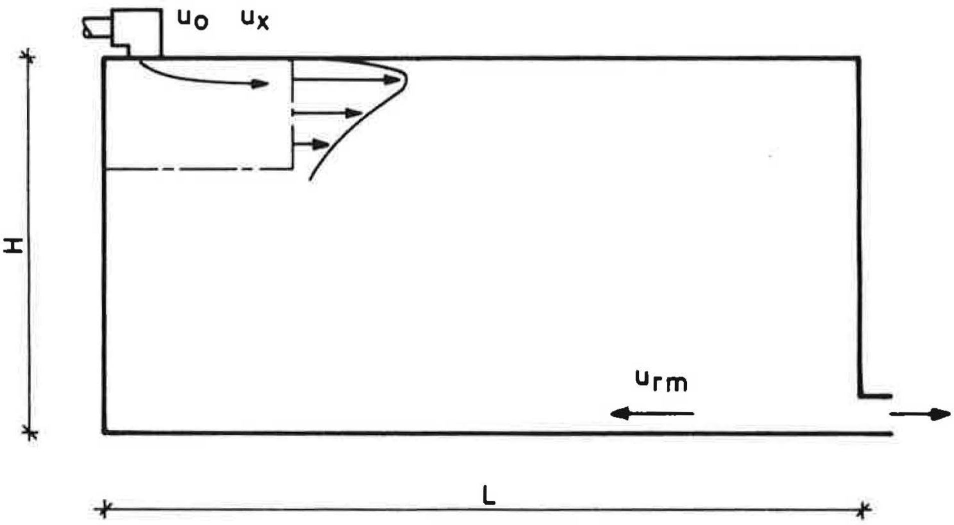

Figure 2. Section through a typical room ventilated by a slot diffuser. H is the height of the room and L is its length .

It is a condition for calculating the flow in the room that velocity and turbulence profiles are known at the supply opening. If the prediction of these profiles is made by the numerical method a situation may arise where a large amount of the grid points is used for calculating the complicated conditions in the diffuser. The flow area of the diffuser is 0.06 x 0.16 m which is very small in comparison with the rest of the room with a typical dimension of 2.50 x 5.00 m, see figure 2.
It has been practical to move the boundary values a short distance from the supply opening and to give them in the form of normalized wall jet profiles which are often created at a distance from a supply opening, see e.g. [1], [2] and [5]. In figure 2 the location of the new boundary values is shown as a dotted line. The velocity- and turbulence gradients are not so significant in the wall jet profiles as in the diffuser which results in saving of grid points, and the wall jet profiles are universal profiles of a given shape.
The necessary design parameters are the velocity decay of the wall jet, growth rate and temperature conditions. These parameters are determined by tests on the relevant air terminal devices mounted in a large test room. The flow of the wall jet is parabolic and therefore, it is rather independent of the dimensions of the room in the isothermal case if only the room has a reasonable size. Measurements on wall jet profiles from different air terminal devices are shown in references [10], [11] and [12]. As an example it can be mentioned that the velocity in a plane wall jet is calculated from the following formula

$$\frac{u_x}{u_0} = K_p\sqrt{\frac{h}{x + x_0}} \tag{1}$$

where u_x and u_0 are the maximum air velocity at the distance x and supply velocity, respectively. K_p is a constant, h is the effective slot height, and x_0 is the virtual origin of the jet. The last-mentioned three quantities characterize a given air terminal device.

3. ROOM WITH PLANE FLOW

In this section some results are given showing the practical application of the numerical method.

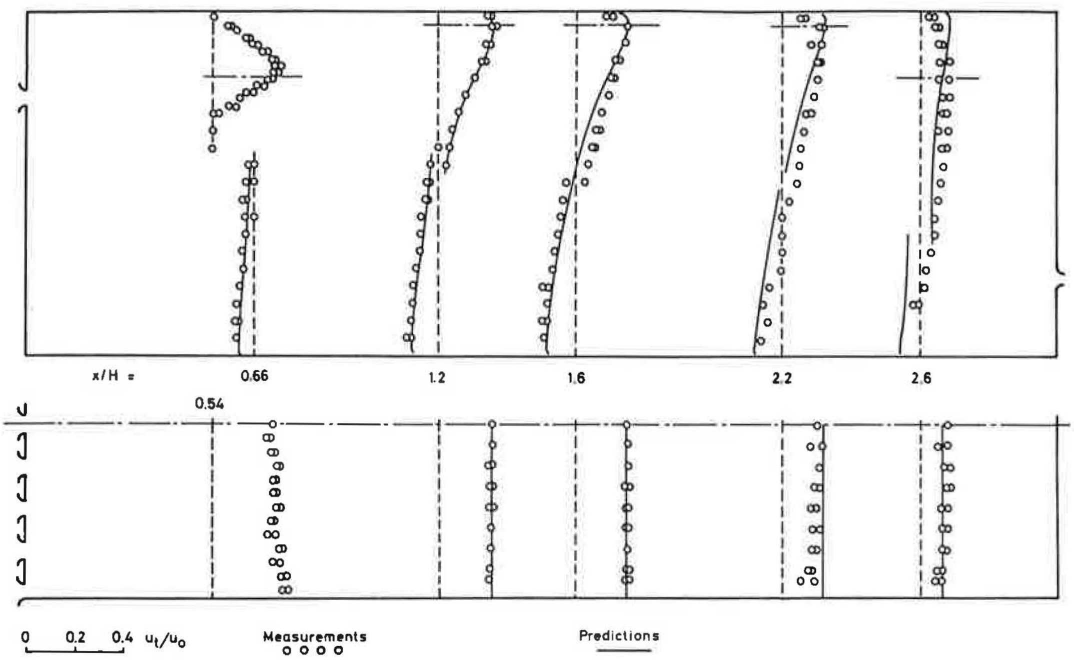

Figure 3. Measurements and prediction of velocity profiles in a room with nine supply nozzles placed at a distance from the ceiling. The upper figure shows a vertical section in the middle plane, and the lower figure shows a horizontal section at the heights listed in the upper figure. The calculated velocity u_t is the total velocity $(u_2+v_2)^{0.5}$. $L/H = 3.0$, $W/H = 1.0$, $h/H = 0.011$ and Reynolds' number $Re = 25000$. W is the width of the room.

Figure 3 shows an example of measured and predicted isothermal velocity profiles in a room where the supply is made by 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 measurings show that the recirculating flow formed in the greater part of the room is fairly plane. The measurings are made by Blum [13].

The calculated velocity profiles in figure 3 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 width barely reaches the measured value.

The use of a wall jet profile as boundary value in the calculations in figure 3 is a good example of the simplification that can be achieved. If the actual supply openings had been used the calculations should have been performed by an equation system for three-dimensional flow instead of the system for two-dimensional flow as used. However, this would result in a severe increase of the storage and computation time.

3.1 Maximum Air Velocity in the Occupied Zone

The maximum velocity in the return flow is called u_{rm}. It is located close to the floor at a distance of 2/3 L from the supply opening. In the normal cases where the occupied zone does not extend into the jet under the ceiling or to the end wall the velocity u_{rm} will also be the maximum velocity in the occupied zone.
There is an unambiguous relationship between the maximum air velocity in the occupied zone u_{rm} and the velocity distribution in the wall jet below the ceiling in a room with a plane isothermal flow. If the velocity in the wall jet is characterized by the value u_L, i.e. the calculated velocity in accordance with formula (1) at the distance $x = L$ from the supply opening, the following can be obtained

$$\frac{u_{rm}}{u_L} = K_{rm} \tag{2}$$

where K_{rm} is only a weak function of the geometry of the room and the geometry of the supply opening. For rough estimations $K_{rm} = 0.7$ can be assumed, see reference [4]. Insertion of (2) into (1) gives

$$u_{rm} = K_p\, K_{rm}\, u_0 \sqrt{\frac{h}{L + x_0}} \quad \text{(m/s)} \tag{3}$$

This equation is a simplified expression for the maximum air velocity in the occupied zone. It is seen that the velocity is proportional with u_0 and it is dependent on the constants K_p, h and x_0 of the supply jet. Further, it depends on the length of the room L while the influence of the complicated conditions of the recirculating flow can be expressed in the factor K_{rm}.

3.2 Distribution of Concentration in a Room with Emission Source

The purpose of the recirculating flow in a room with jet ventilation is to create a uniform climate in the entire occupied zone. It is evident that the pollution from a source in the occupied zone, e.g. welding smoke, by this ventilation will spread in the entire room. Therefore, the issue is to clarify the relations between the design of the air terminal device, the location of the source of emission and the concentrations of pollution that will arise in various parts of the room.

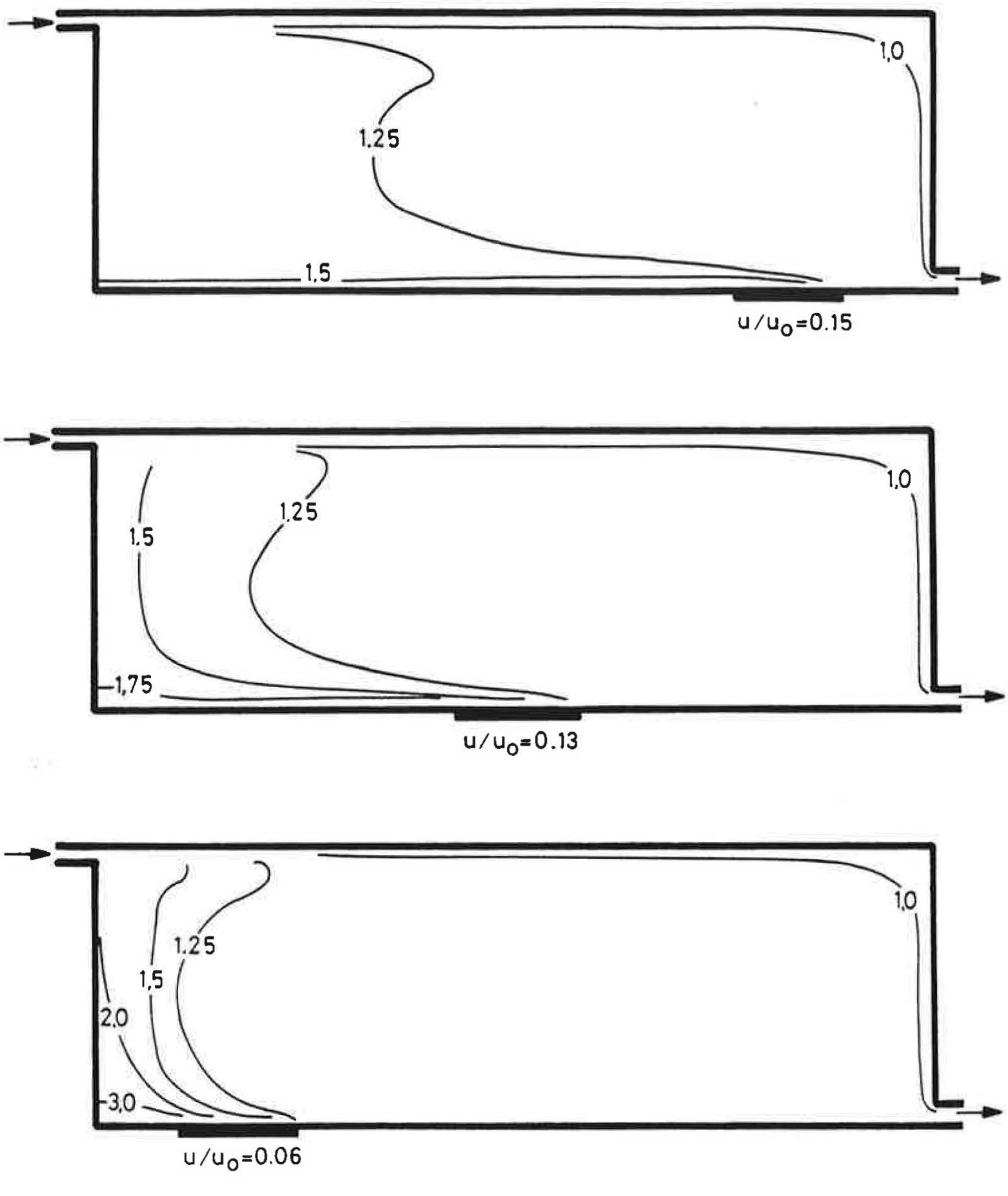

Figure 4. Distribution of concentration c/c_R in a room with plane isothermal flow using three different locations of a line source. $h/H = 0.01$ and $L/H = 3.0$. From reference [14].

The concentration distribution is determined by extending the flow equations by a differential equation for mass transport. In principle it is assumed that there is no differences in the density of the polluting gas and the air in the room, and often in practice any differences of density are not very significant (high turbulence level, low concentrations). The predicted concentration distribution can be interpreted

as a mass fraction of a gas as well as a particle or droplet density. The droplet and particle size, however, should not be of such magnitude that the settling speed becomes significant in relation to the air velocities in the ventilated room.
The location of the emission source in a ventilated room is a decisive parameter for the concentration distribution that is created in the occupied zone. The calculations in figure 4 explain these conditions using three different locations of the emission source, see [14]. In the upper drawing in figure 4 the emission source is located close to the area where the velocity is largest in the occupied zone (15% of the supply velocity), and the maximum size of the concentration c/c_R is app. 1.5 in the area below the supply opening. c is the local concentration, and c_R is the concentration in the return opening. If the source is placed below the supply where the local air velocity is 6% of the supply velocity concentrations up to $c/c_R = 3.0$ will take place, i.e. concentrations three times the concentration in the exhaust.

3.3 Thermal flow in a Room

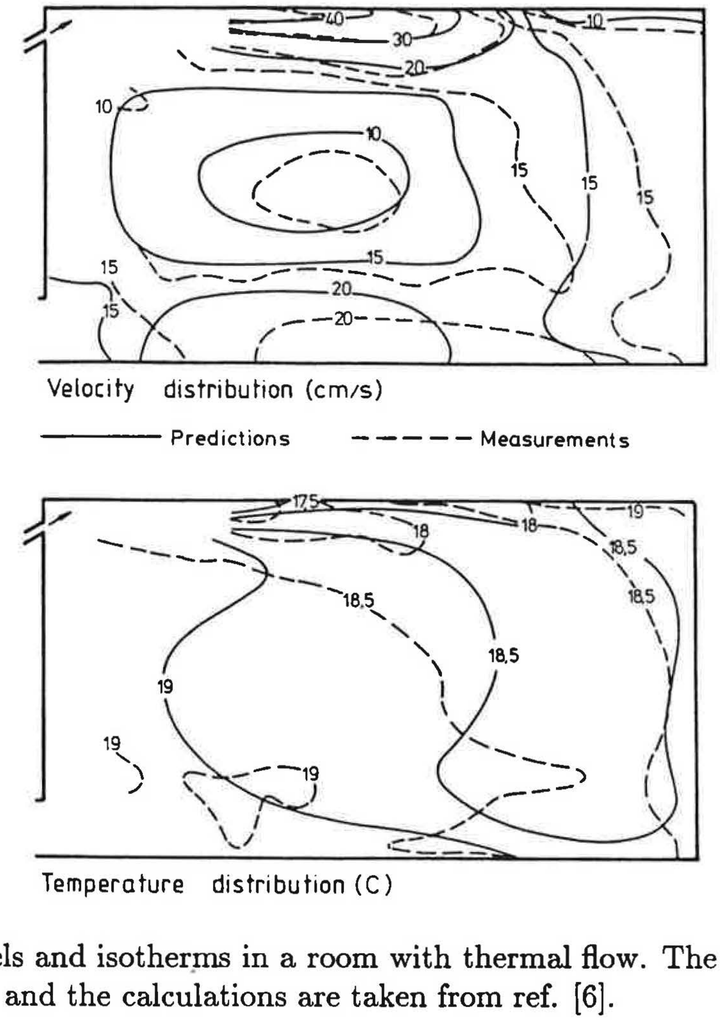

Figure 5. Isovels and isotherms in a room with thermal flow. The tests are made by Hestad [16] and the calculations are taken from ref. [6].

Often there will be temperature gradients in a room due to thermal loss through windows or walls or due to heat sources such as solar gain, persons or machines. To calculate the velocity and temperature distribution in a room in this case the equation system must be extended with an energy equation describing the energy transport (heat) in the flow. Local temperature gradients form a buoyancy which affect the vertical momentum equation and the equation for turbulent kinetic energy and the dissipation of turbulent kinetic energy with an extra source term, see ref. [6]. In the present calculations it is furthermore assumed that the turbulent Prandtl number is dependent upon the buoyancy in accordance with a suggestion proposed by Gibson and Launder [15] for horizontal flow.
Figure 5 shows the predicted and the measured isovels and isotherms in cases of plane thermal flow in a room. It is seen that there is fairly good agreement for the general flow and also for details such as maximum air velocity in the occupied zone and penetration depth of the wall jet.

4. ROOM WITH THREE-DIMENSIONAL FLOW

If a single air terminal device with a limited width in relation to the width of the room is used or if there is a long distance between several openings a three-dimensional flow arises. In many practical cases of three-dimensional flow the supply opening has a width-height ratio of approximately 1, and a fully developed three-dimensional wall jet is created in the room.

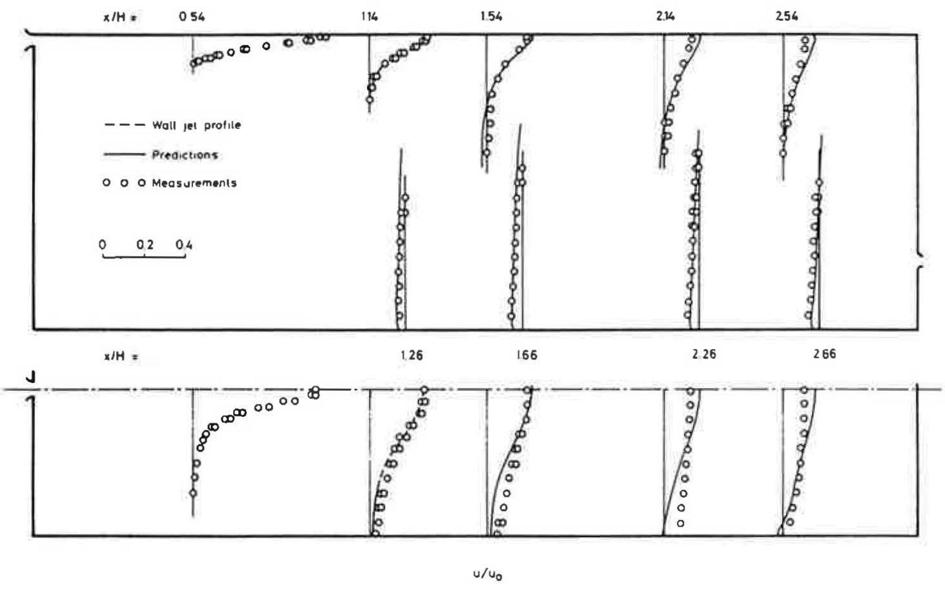

Figure 6. Measurements and predictions of velocity profiles in a room with circular nozzle placed in the middle plane of the room close to the ceiling. The upper figure shows a vertical section, and the lower figure shows a horizontal section through the symmetry line of the nozzle. $L/H = 3.0$, $W/H = 1.0$, $a/A = 0.00126$ and $Re = 93000$.

Figure 6 shows an example of measured and predicted velocity profiles in a room where the supply consists of a single nozzle placedin the end wall close to the ceiling. The length of the room L is three times its height, and the relative supply area a/A is equal to 0.00126, where $A = H \cdot W$ is the area of the end wall. The upper drawing shows a vertical section through the middle of the room, and the lower drawing shows a horizontal section through the centre line of the supply opening.
The measurings in figure 6 show that the flow under the ceiling forms a typical three-dimensional wall jet with the strong horizontal growth rate. It is also seen that a recirculating motion with return flow is formed in the lower part of the room. The measurings are performed by Blum [13]. The calculated velocity profiles in figure 6 are determined as a numerical solution of the three-dimensional flow equations, see ref. [5]. In the calculation the supply opening is characterized by the three-dimensional wall jet which is formed up to the distance $x/H = 1.14$. The calculations show good agreement with the measurings. Thus, the maximum deviation of the velocity in the occupied zone is below 1% of the supply velocity. However, it is seen that the calculated horizontal growth rate of the jet does not reach the measured value.

4.1 Calculation of Thermal Comfort

The calculations are often performed for the purpose of giving a velocity distribution and a temperature distribution in the room or just to determine the maximum velocity in the occupied zone. It would be appropriate to extend the method so that a direct calculation of the thermal comfort in the room will be possible.
Fanger [17] has developed a thermal comfort index which is used for calculating the distribution of the number of dissatisfied persons (PPD) in the room shown in figure 6. Fanger defines a dissatisfied person as a person giving the vote "cool, cold" or "warm, hot" in an ASHRAE 7-point scale of thermal comfort. He has shown that even in a perfect thermal environment it is not possible to achieve PPD-values lower than 5% for similarly dressed persons with the same activity. This is due to the deviation in the individual sense of thermal comfort of a group of persons.

Figure 7 shows the distribution of the number of dissatisfied persons through the occupied zone of the room given in figure 6, see ref. [18]. Summer conditions are assumed, i.e. light clothes (0.5 clo) and low activity (58 W/m_2). Further, it is assumed that the flow is isothermal corresponding to a low thermal load of the room.

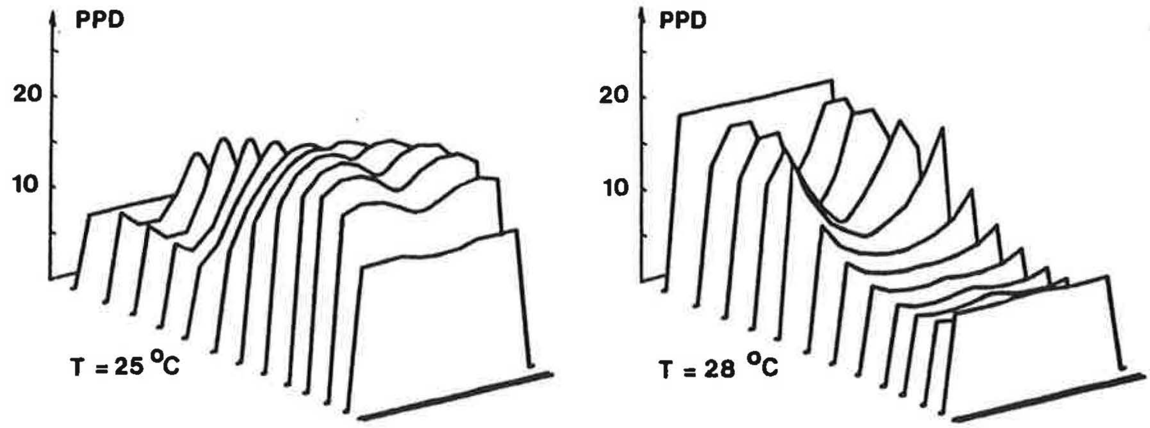

Figure 7. Distribution of predicted percentage of dissatisfied persons (PPD) in the room in figure 6. The supply velocity u_0 is 5 m/s, see ref. [18].

The curves on the left-hand side in figure 7 show the PPD-distribution for a supply velocity of 5 m/s and an air temperature of 25°C. The PPD-distribution is rather identical with the velocity distribution in the occupied zone because the temperature is relatively low. The maximum value is 23% and the minimum value is 7% corresponding to a local velocity less than 0.1 m/s. The mean value of the room is PPD_{avg} = 15%, and this quantity describes the thermal comfort of the entire room in the given situation. The curves on the right-hand side in figure 7 show the distribution of PPD at an air temperature of 28°C. The low PPD-value is obtained in areas of high velocity because cooling by forced convection counteracts the high temperature level. The minimum, maximum and mean values of PPD are 8%, 19% and 12%, respectively. Even though the mean value differs only little from the mean value previously found it is seen that different areas of the room have a high level of PPD in the two cases. An air temperature of 26.5°C is optimal because it gives the lowest mean value PPD_{avg} = 6% (also called LPPD) and this value is close to the lowest possible value of 5%.
It is to be noted that the PPD and the LPPD-values will be increased due to the effect of the velocity and temperature gradients dependent on the height of a person and due to the effect of the turbulence level, see [19].

5. DEVELOPMENTS OF THE PREDICTION METHOD

This section will deal with some of the development activities that are necessary in order that the numerical calculation of air distribution in a room may be applicable in practice. An improved handling of boundary values such as air terminal devices and other special geometries, development of a low-turbulence model and development of software capable of calculating directly the thermal comfort will be dealt with.

5.1 Air Terminal Device and Low-Turbulence Flow

Turbulence models used for the present calculations assume a fully developed turbulence and, consequently, a similar flow that is independent of the Reynolds number. (A slight influence from the wall functions is disregarded). This indicates among other things that an air velocity at a given point is proportional with the supply velocity or that the maximum velocity in the occupied zone in the isothermal case is proportional with the volume flow to the room or proportional with the air exchange rate n of the room

$$u_{rm} = const \cdot n \tag{4}$$

Figure 8 shows the maximum air velocity in the occupied zone as a function of the air exchange rate for 5 different air terminal devices, see [20]. The tests were performed in a full-scale room with the dimensions 2.4 x 3.6 x 5.4 m. The different air terminal devices are of the following types.

A. Nozzle

B. Grille

C. Grille with blades adjusted for high diffusion

D. Wall mounted diffuser

E. Ceiling mounted diffuser

It is seen from figure 8 that u_{rm} is proportional with the air exchange rate n at high air flow rates while there are deviations at low air flow rates. Figure 8 also shows that the low air flow rates have practical relevance ($u_{rm} \sim 0.1 \sim 0.15$ m/s). There may be some uncertainty involved in measuring the low velocities, but it is evident that there must be some low turbulence effect in this situation, which is especially noticable for air terminal devices D and E.

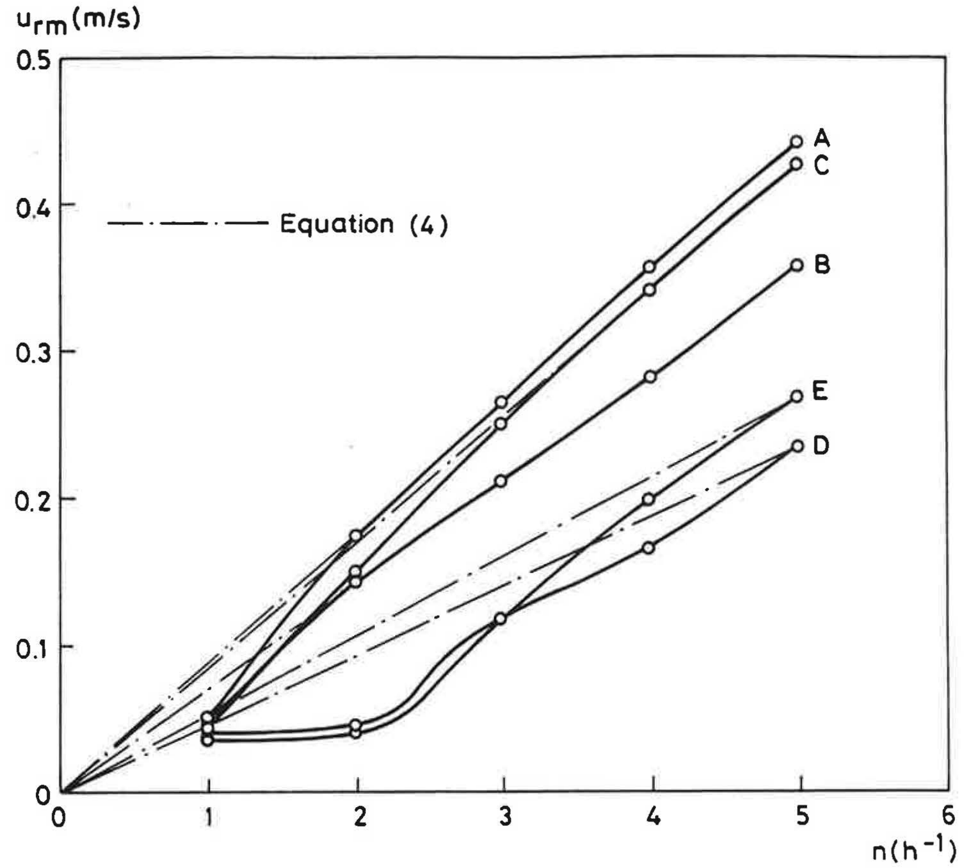

Figure 8. Maximum velocity in the occupied zone as a function of the air exchange rate. The tests were performed for five different air terminal devices (A to D).

If formula (3) for the maximum air velocity in a room with plane flow is considered it is seen that three areas can be isolated in which there may be low turbulence flow, i.e. flow as a function of Reynolds' number. In the air terminal device h, x_0 and K_p may be functions of Reynolds' number which is often the case with the supply velocities used in practice, see [10] and [12]. The low air flow rates below the ceiling may be uncharacteristic of a wall jet, and this signifies low turbulence effect. Finally, formula (3) shows that low turbulence effects in the recirculating flow can be expressed by a value of K_{rm} which varies with Reynolds' number.
It can be concluded that some of the low-turbulence effect can be included in the boundary values of an air terminal device if they have been measured in advance. However, there is still a need for a turbulence model capable of treating flow conditions that do not have a fully developed turbulence level.

5.2 Other Possibilities of Development

There are other situations in a ventilated room where internal boundary values can give rise to saving of grid points. Figure 9 gives a description of the thermal flow over a heat source and a cold downward flow, which is given as internal boundary values for the numerical method.

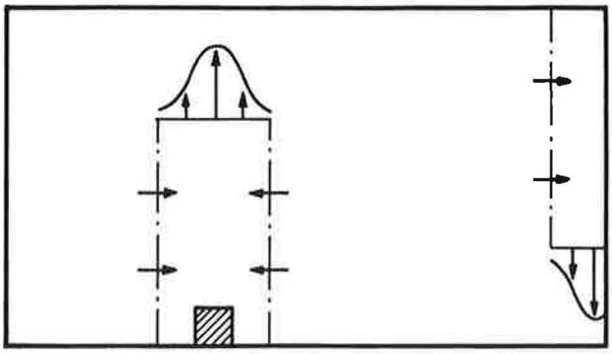

Figure 9. Thermal flow over a heat source and a cold downward flow given as boundary values for the numerical method.

In the long run efforts should of course be made to find a design method capable of making a direct calculation of the conditions in an air terminal device and other special areas. Such efforts may result in methods first solving the flow conditions of the various areas and later connecting them in the final iteration.
It is necessary to develop methods to deal with special room geometries such as ceiling obstacles and rooms that are not cubical. Awbi and Setrak [21] have worked with the flow around obstacles, and especially in the German Democratic Republic some research have been made on special room geometries such as buildings with pitched roofs, concert halls and auditoria with many supply and return openings, etc., see [22].
It will be possible to calculate new parameters in the air movement such as thermal comfort, see [18]. It will also be possible to calculate special quantities such as the age distribution of the air at a point. This is a quantity used for calculating the ventilation efficiency and the air exchange efficiency, see Davidson and Olsson [23].
The radiation exchange between the individual surfaces can have significance for the energy transport in the room. It will be possible to extend the numerical method to include routines performing such a calculation of the radiation parallel with the convective calculation. The method is e.g. used in combustion calculation.
In 1987 ASHRAE has implemented research on the numerical calculation of air movement in rooms, and in 1988 the IEA-countries will begin a three year programme within the same field. Undoubtedly the method will be subject to a rapid development in the years to come. Perhaps this will result in a software package which will be accessible and applicable in practical design in the future.

6. CONCLUSION

The numerical calculation of the air flow in a room has proved to give good results

for two-dimensional as well as for three-dimensional flow. It is possible to perform satisfactory calculations of the flow conditions in the case of thermal fields with buoyant effect, and it is also possible to calculate the concentration distribution in a room with an emission source.
Development work is necessary to give a practical description of air terminal devices. Development of a turbulence method to deal with low turbulence flow is also desirable. In the long run it can be expected that calculations will be performed from which the thermal comfort in a room can be taken stated directly.

7. REFERENCES

[1] Nielsen, P.V., Berechnung der Luftbewegung in einem zwangsbelüfteten Raum, Gesundheits-Ingenieur, Oktober, 1973.

[2] Nielsen, P.V., Prediction of air flow and comfort in air conditioned spaces, ASHRAE Transactions 1975, Vol. 81, Part II.

[3] Holmberg, R., M. Larsson and S.G. Sundkvist, Beräkning av hastighedsfördelningen i en ventilerad lokal, Svensk VVS, No. 10, 1975.

[4] Nielsen, P.V., A. Restivo and J.H. Whitelaw, The Velocity Characteristics of Ventilated Rooms, J. Fluids Eng., Vol. 100, 1978.

[5] Gosman, A.D., P.V. Nielsen, A. Restivo and J.H. Whitelaw, The Flow Proporties of Rooms With Small Ventilation Openings, J. Fluids Eng., Vol. 102, September 1980.

[6] Nielsen, P.V., A. Restivo and J.H. Whitelaw, Buoyancy-affected flows in ventilated rooms, Numerical Heat Transfer, Vol. 2, 1979.

[7] Launder, B.E., D.B. Spalding and J.H. Whitelaw, Turbulence models and their experimental verification, Imperial College, Heat Transfer Section, Reports HTS/73/16-30, 1973.

[8] Pope, S.B. and J.H. Whitelaw, The calculation of near-wake flows, J. Fluid Mech., Vol. 73, 1976.

[9] Gosman, A.D. and W.M. Pun, Lecture notes for the course entitled "Calculation of Recirculating Flows", Imperial College, Heat Transfer Section, Report HTS/74/2, 1974.

[10] Nielsen, P.V. and Å.T.A. Möller, Measurement of the three-dimensional wall jet from different types of air diffusers, World congress on heating, ventilating and air-conditioning, Copenhagen 1985.

[11] Nielsen, P.V. and Å.T.A. Möller, Measurements on buoyant wall jet flows in air-conditioned rooms, Room Vent 87, International conference on air distribution in ventilated spaces, Stockholm, 1979.

[12] Nielsen, P.V. and A.T.A. Möller, Measurements on buoyant jet flows from a ceiling-mounted slot diffuser, internal report, The University of Aalborg, 1988.

[13] Blum, W., Diplomarbeit, T.H. Aachen, 1956.

[14] Nielsen, P.V., Contaminant distribution in industrial areas with forced ventilation and two-dimensional flow, IIR Joint Meeting, University of Essen, 1981.

[15] Gibson, M.M. and B.E. Launder, Ground Effects on Pressure Fluctuations in the atmospheric Boundary Layer, J. Fluid Mech., Vol. 86, p. 491, 1978.

[16] Hestad, T., Private Communication, Farex Fabrikker A/S, Norge, 1974.

[17] Fanger, P.O., Thermal Comfort, McGraw-Hill Book Company, New York, 1973.

[18] Nielsen, P.V., The Distribution of Air Velocity in Large Rooms With Small Side-wall Mounted Supply Openings, 15th international congress of refrigeration, Venice, 1979.

[19] Hanzawa, H., A.K. Melikow and P.O. Fanger, Airflow characteristics in the occupied zone of ventilated spaces, ASHRAE transactions, Vol. 93, part 1, 1987.

[20] Heiselberg, P. and P.V. Nielsen, The contaminant distribution in a ventilated room with different air terminal devices, Room Vent 87, International conference on air distribution in ventilated spaces, Stockholm, 1987.

[21] Awbi, H.B. and A.A. Setrak, Air jet interference due to ceiling-mounted obstacles, Room Vent 87, International conference on air distribution in ventilated spaces, Stockholm, 1987.

[22] Ehle, A. and R. Scholz, Beispiele für die numerische Berechnung von zweidimensionalen Geschwindigkeits- und Temperaturfeldern in Raümen, Luft- und Kältetechnik, No. 4, 1984.

[23] Davidson, L. and E. Olsson, A numerical investigation of the local age and the local purging flow rate in two-dimensional ventilated rooms. Room Vent 87, International conference on air distribution in ventilated spaces, Stockholm, 1987.