#6320

RADIAL SPREAD OF SUPPLY AIR AND HORIZONTAL DISPLACEMENT VENTILATION

Sture Holmberg and Yan-Qiu Tang
National Institute of Occupational Health
Ventilation Division
S-17184 Solna, Sweden

SUMMARY

In order to achieve a horizontally displaced air flow pattern in an isothermally ventilated laboratory room, the incoming air is radially spread over the inlet wall. This is done to avoid troublesome air jets in the inlet region and to create a steady unidirectional flow through the room. Measured velocities recorded around the cylindrical spreader of the inlet air are reported.

Altered inlet conditions will influence the velocity profile from the air spreader as well as the flow pattern in the room. This will lead to varying degrees of displaced flow in the room. Calculated velocity fields and measured air change efficiencies are shown in order to give a better understanding of the horizontal flow patterns investigated.

#6320

RADIAL SPREAD OF SUPPLY AIR AND HORIZONTAL DISPLACEMENT VENTILATION

Sture Holmberg and Yan-Qiu Tang
National Institute of Occupational Health
Ventilation Division
S-17184 Solna, Sweden

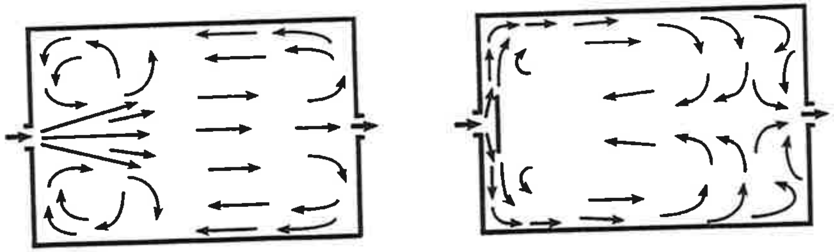
INTRODUCTION

There is today an ever increasing use of mechanical ventilation systems in our work places and dwellings. These systems are designed to supply the room with enough air to keep contaminant concentrations and temperature levels under control. Research in the field of heating and ventilation during the last decade has shown an increasing interest in studying the air flow patterns in rooms. It has been realized that it is not only the quantity of air that influences the conditions in the room, but also the distribution of the air in the room.

There has been a growing demand for a good air exchange in the whole room, with an optimized volume of the fresh air supplied. At the same time the flow velocities in the occupied zone must not be too high, since this would mean a lower degree of comfort and is not acceptable. A background to this article can be found in [1,2].

REFERENCE FLOW PATTERNS

In order to have an idea of what different inlet configurations will mean for the airflow in the room it is of interest to have a look at two reference examples, where the main behaviour of the flow can be considered to be more or less known. In the two examples given here high velocity jets are used to distribute the incoming air. In the first example, Figure 1a, a free jet flows straight into the room.



a) b)

Fig. 1. The average flow behavior of a free jet (a) and a wall jet (b) are fairly well understood [3]. Schematic illustration.

Such a jet passes rapidly through the room, creating vortices and mixing downward flows. Recirculation from the opposite wall occurs along the room surfaces and within a fairly short period of time a mixing ventilation flow dominates.

In the second example, Figure 1b, a circular plate has been placed perpendicular to the symmetry line of the incoming jet at a certain distance from the wall, blocking the incoming flow. This will lead to the creation of wall jets [3] which spread continuously along the boundaries of the room. When the air flow reaches the opposite wall a symmetrical recirculation will occur in the direction of the supply inlet.

The diffusor (jet killer) used in the present work has been designed with the help of experiences from the two reference examples studied. The radial spread is greater than in the case with the free jet. This is why the jet has to be prevented from going straight into the room. However, in order to avoid over-strong wall jets the radial spread must not be as strong as in Figure 1b. Instead there should be a fairly slow-moving flow towards the surrounding corners where a secondary flow into the room is created. A major difficulty is to avoid vortices and to find an efficient way of dissipating any vortices that have been created.

JET DISSIPATION IN BALANCED ROOM FLOWS

Our laboratory measurements were carried out in a full-scale test chamber, Figure 2.

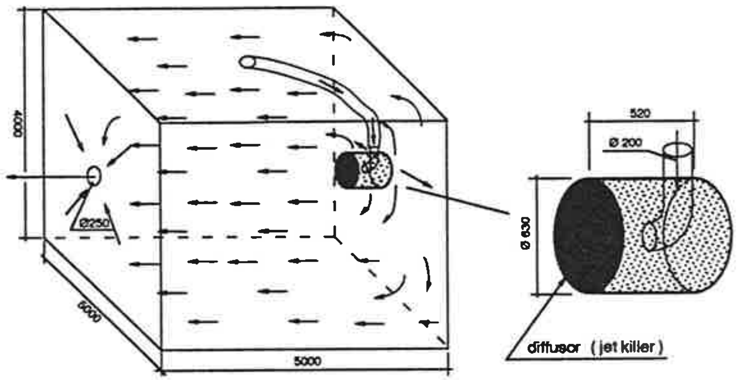


Fig. 2. Test chamber and diffusor (jet killer) for radial spread of supply air.

The supply air was taken from a big hall surrounding the test chamber. The air passed through small circular holes in the cylindrical air diffusor. The holes were equally distributed over the diffusor envelope surface having a hole area shear of 0.33 of the total area. After passing the room the evacuated air was exhausted to the atmosphere.

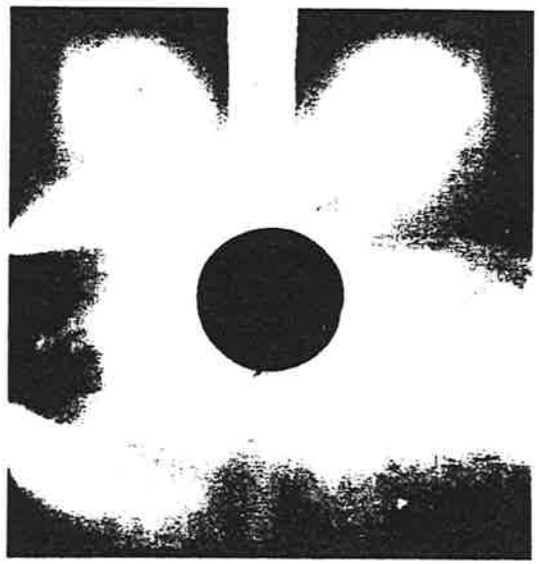


Fig. 3. Radial spread of incoming air, smoke visualization.

Air flow rates used in the room were balanced with fans for supply and exhaust. The diffusor was expected to minimize vortex creation and recirculation. A radial, wall parallel primary flow was obtained in the near zone. In the next step an air curtain was created on and out from the supply wall.

MEASUREMENTS

The general flow field was established by smoke experiments. Figure 3 showed that this was the case for the radial spread at the supply inlet as well. Measured flow velocities close to the diffusor are given in Figure 4.

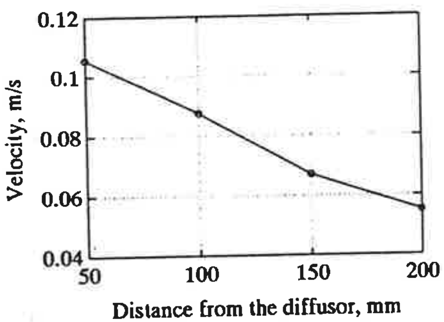


Fig. 4. Measured radial air supply velocities in the diffusor near zone.

From this curve it can be deduced that critical velocities from a thermal comfort point of view are never exceeded. In the example shown a flow rate of 400 m^3/h corresponding to four air changes per hour was used. At radial distances > 0.2 m from the diffusor the air velocity was too low to be measured with our hot wire anemometry technique. The flow rates in the occupied zone of the test chamber were also too low to be measured.

Air Change Efficiency

The air change efficiency is a measure of the degree to which the air is renewed per room-volume of supplied air. In an efficient system the greater part of the room volume is renewed, while in an inefficient system, where so-called short-circuit flow occurs, less than half of the room volume is renewed. In a completely mixed room the air change efficiency will amount to 0.5. For displaced flows values over 0.5 are expected. Measured air change efficiencies from the present study are given in Table 1.

Table 1. Measured mean age of air and air change efficiency for different air change rates in the test chamber.

n air changes per hour	τ_n hours	Mean age of air in the room $<\bar{\tau}>$ minutes	Air change efficiency ε_a, %
1	1	43.5±4.7	69±7
2	0.5	24.0±0.4	62±2
4	0.25	12.3±0.7	61±3

A concentration decay tracer gas method has been adopted to estimate the mean age of air $<\bar{\tau}>$ in the test chamber. The expression used for the mean age of air is given below [4]

$$<\bar{\tau}> = \frac{\int_0^\infty t c_e(t)\,dt}{\int_0^\infty c_e(t)\,dt} \tag{1}$$

$c_e(t)$ is the exhaust tracer gas concentration.
From the values obtained for the mean age of the air, the air change efficiencies are calculated by

$$\varepsilon_a = \frac{\tau_n}{2<\bar{\tau}>} \tag{2}$$

τ_n is the room volume divided by the volumetric flow rate used.
Table 1 shows that higher flow rates give lower ε_a values. One reason for this can be an expected increase in turbulence and mixing with increased Reynolds number. On the other hand the highest ε_a values measured at the lowest flow rates show the highest fluctuations. The flows with very low Reynolds number, at one air change per hour, can be assumed to be extremely sensitive to temperature fluctuations and other disturbances in the room [5]. Another reason for the differences in the ε_a values obtained at different Reynolds numbers can be the construction and installation parameters of the diffusor and their dependence on the Reynolds number.

Flow visualizations and results from isothermal tracer gas experiments in Table 1 show the presence of a horizontally displaced flow in the room. Air change efficiencies close to 0.7 constitute perhaps the best evidence for this. Disturbances in the form of mixing and recirculation are certainly also present but in general the flow looks quite unidirectional, with a small exception for the zone in front of the diffusor, Figure 5.

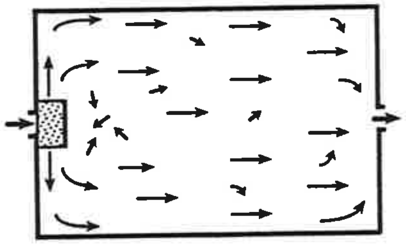


Fig. 5. Schematic illustration of the observed flow pattern, when a radial spread of the supply air was used.

NUMERICAL SIMULATIONS

As was mentioned above, the air velocities in the test room were too low to be successfully measured. Consequently numerical simulations were done for comparable flow situations. The simulations were expected to give more information about the flow behaviour. In the simulations the influence of the exhaust inlet was not taken into consideration. The outlet wall was kept fully open and the outgoing velocity profile was studied with this configuration.

We started with two-dimensional simulations. The program used is called CALC-BFC and comes from Chalmers University of Technology, Gothenburg. It is a finite volume computer code which uses a collocated variable arrangement.

A small-scale (1:5) model simulation of the air flow in the test chamber, Figure 2, was done. The Reynolds number similarities mean higher air supply velocities in the small-scale model. Supply air velocities of 0.5 m/s were used. We started with a free jet simulation. In the next step a simple line blockage was used to stop the incoming flow jet. The stop line and the supply inlet were the same length.

Results from the two-dimensional, laminar calculations are shown in Figures 6 to 8. In Figure 6 the free jet simulation is shown. The jet generated vortices down streams. An uneven velocity profile is observed for the outgoing flow. When the 'diffusor' was used, Figure 7, the free jet dominance was decreased. The incoming airflow was better distributed and we had a fairly stable, horizontally displaced flow through the room. Pressure differences in the room as calculated from the simulation in Figure 7 are shown in Figure 8.

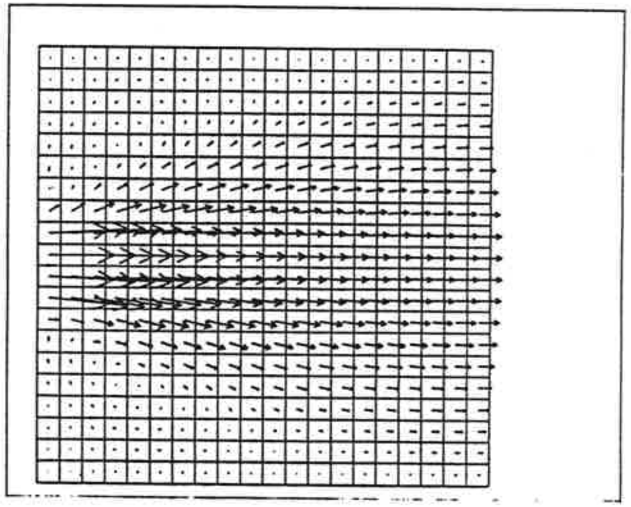


Fig. 6. Laminar two-dimensional simulation of the flow pattern in a small-scale model. Vortex creation and an uneven outgoing (exhaust) velocity profile are observed. Free jet air supply.

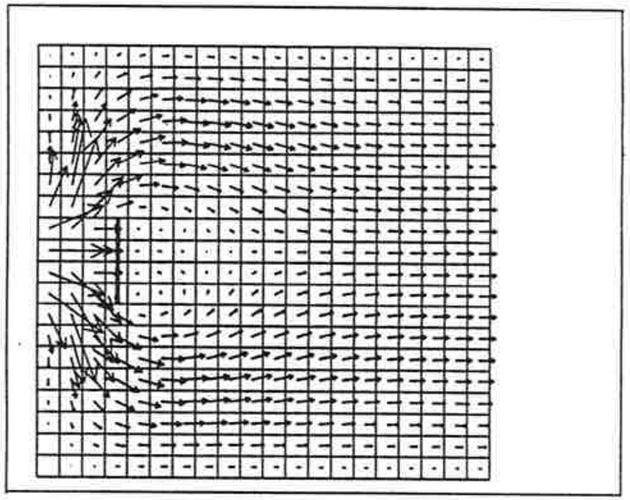


Fig. 7. The free jet dominance in Figure 6 can be decreased by using a 'diffusor' to spread the incoming air. This gives a balancing wall jet influence resulting in a better outgoing velocity profile and less vortex creation. With a finer grid and the plate closer to the wall a better result will most likely be reached.

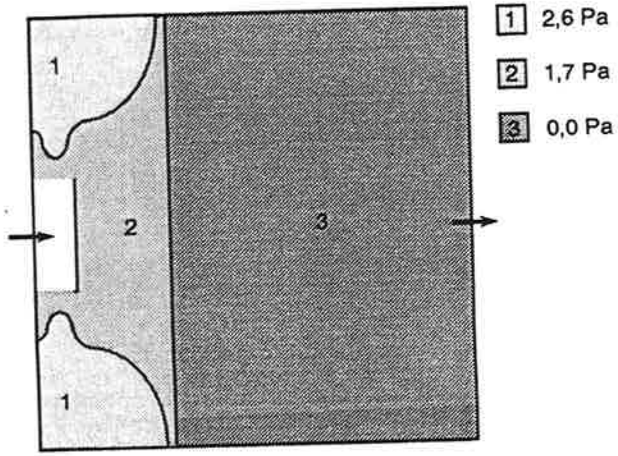


Fig. 8. An approximate pressure distribution for the flow field in Figure 7 is given above. Relative pressures for different flow zones in the small-scale model.

A computer code for three-dimensional ventilation simulations, VentAir2, was used for a full view of the flow in the test chamber. The VentAir2 code has been developed at the Royal Institute of Technology, Stockholm in cooperation with the National Institute for Occupational Health, Solna and The National Swedish Institute for Building Research, Gävle. This is a finite volume program with staggered grid arrangements i.e. velocity locations at the boundaries of the grid cells. Wall functions and a standard k-ε turbulence model have been employed. The program has also been equipped with a multi-grid solver to speed up the convergence rate and a heat radiation model for non-isothermal simulations.

To simplify the simulations the diffusor in Figure 2 was opened, i.e. the perforated surface envelope was taken away. This means that the incoming free jet was stopped and spread by the circular plate in front of the supply opening. The flow simulations were done for the test room shown in the figure with the supply inlet in the wall plane 0.52 m from the plate. Supply air velocities around 0.1 m/s, corresponding to 1.6 air changes per hour, and a Reynolds number comparable to the one in the two dimensional case were used.

Results from the three dimensional simulations are shown in Figures 9 to 11. In Figure 9 the radial spread along the inlet wall is illustrated. In Figure 10 a vertical mid-width cut of the flow pattern in the room is shown. The flow structures in the turbulent, three-dimensional simulations were more wall jet influenced and showed a stronger tendency to recirculation than the laminar two-dimensional ones. We were not able to stop the recirculation in the test chamber simulation, in other words a unidirectional horizontally displaced flow was not fully achieved.

One reason for this can be the diffusor, Figure 2, which in the simulations was only replaced by a plate. When the supply outlet was smaller than the stop plate the free jet was not as strong as in the two dimensional case and the wall jet structure took over. Another reason can be the difficulties in finding correct boundary conditions for the model.

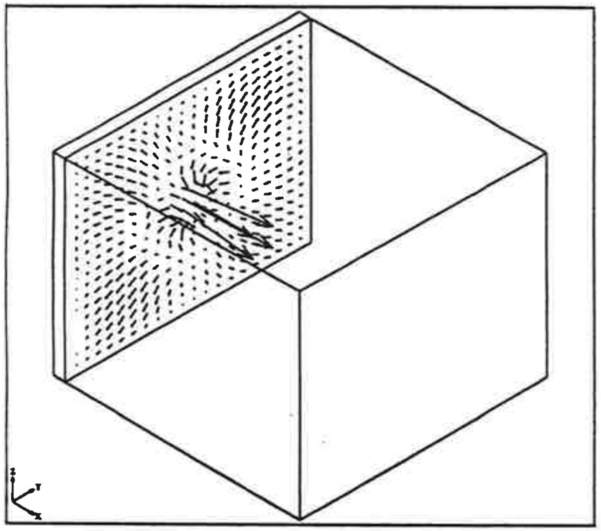


Fig. 9. Simulated radial spread along the inlet wall.

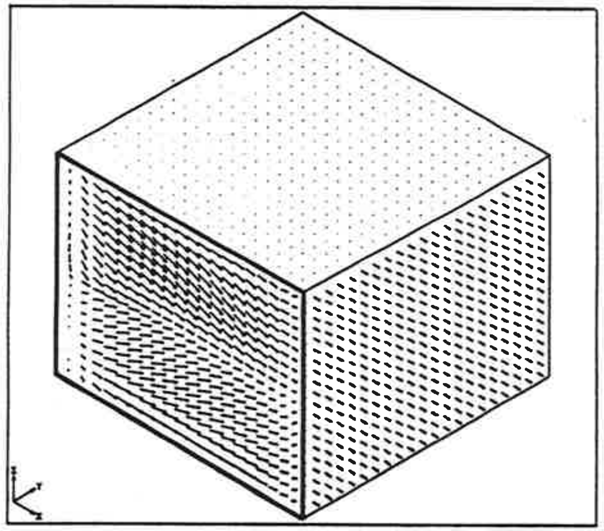


Fig. 10. Side view. The wall jet influence seems strong.

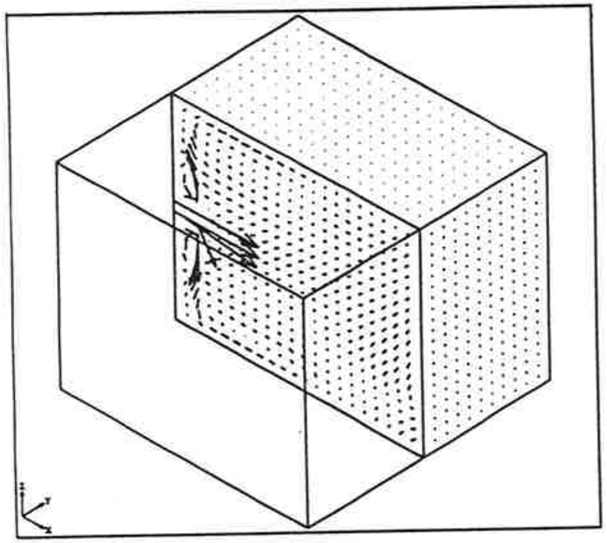


Fig. 11. Vertical mid-width cut of three-dimensional simulation. Recirculation occurs. This was expected after seeing the side view in Figure 10.

CONCLUSION

The main conclusion from this investigation is that it is possible to achieve air change efficiencies greater than 0.5 in an isothermally ventilated full-scale test chamber. This means that a displaced air flow pattern has been obtained without help from thermal forces. The radial spread of the incoming air has minimized the air mixing in the room and has made a horizontally displaced flow pattern possible. Neither the wall jet nor the free jet influence must be too strong.

It is, however, too early to draw any general conclusions from what has been reported. The investigation is limited and the test room has a special geometry. Also the fact that the supply outlet and the exhaust inlet are on opposite walls is an advantage which is not always present in real systems.

The choice of diffusor for the radial spread of incoming air influences the flow pattern in the room. The measured air change efficiencies vary with the total air flow through the room. Very little is known today about the relationships. Because of very low flow rates in the test chamber it has been difficult to establish the influence of changed inlet conditions on the flow patterns in the room. Better visualization methods or measurements in small scale-models with higher velocities are interesting future aspects.

ACKNOWLEDGEMENTS

The authors are grateful to Björn Nilsson who helped us with figures, to Yuguo Li for his help with numerical solutions and to Mrs. Ann Brown for checking the language. The Swedish Work Environment Fund has sponsored this work.

REFERENCES

[1] Holmberg, S. 'A method of creating plug flow in ventilated rooms', Roomvent'90 proceedings, Oslo, June 1990

[2] Tang, Y-Q, Holmberg, S. 'An Experimental Investigation of a Horizontal Displacement Ventilation System, Undersökningsrapport 1992:2, National Institute of Occupational Health, Solna, Sweden, 1992

[3] Waschke, G. 'Über die Lüftung mittels isothermer, turbulenter, radialer Deckenstrahlen, Klima + Kälte-Ingenieur 7-8/1976, pp.467-474.

[4] Sandberg, M., Skåret E. 'Air Change and Ventilation Efficiency - new aids for HVAC designers', The National Swedish Institute for Building Research, Gävle, 1989 (in Swedish)

[5] Skovgaard, M. et al, 'High and Low Reynolds Number Measurements in a Room with an Impinging Isothermal Jet, Roomvent'90 proceedings, Oslo, June 1990