



AALBORG UNIVERSITY
DENMARK

Aalborg Universitet

Flow Pattern in Ventilated Rooms with Large Depth and Width

Yue, Zou; Nielsen, Peter V.

Publication date:
2000

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

[Link to publication from Aalborg University](#)

Citation for published version (APA):

Yue, Z., & Nielsen, P. V. (2000). *Flow Pattern in Ventilated Rooms with Large Depth and Width*. Dept. of Building Technology and Structural Engineering, Aalborg University. Indoor Environmental Engineering Vol. R0042 No. 117

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal -

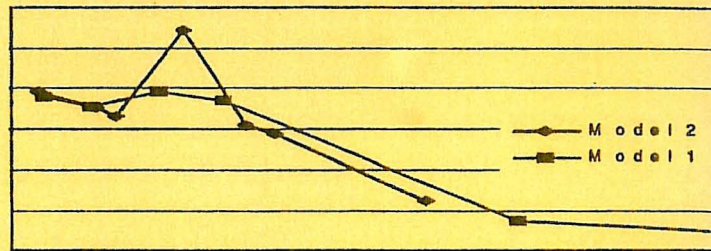
Take down policy

If you believe that this document breaches copyright please contact us at vbn@aub.aau.dk providing details, and we will remove access to the work immediately and investigate your claim.

Flow Pattern

Flow Pattern in
Ventilated Rooms with
Large Depth and Width

Z. Yue, P.V. Nielsen



Paper No 117

Indoor Environmental Engineering

In: Proceedings of ROOMVENT 2000, 7th International
Conference on Air Distribution in Rooms, Reading,
UK, July 9-12, 2000, Vol. 1, pp. 607-612

The *Indoor Environmental Engineering* papers are issued for early dissemination of research results from the Indoor Environmental Engineering Group at the Department of Building Technology and Structural Engineering, Aalborg University. These papers are generally submitted to scientific meetings, conferences or journals and should therefore not be widely distributed. Whenever possible, reference should be given to the final publications (proceedings, journals, etc.) and not to the Indoor Environmental Engineering papers.

Flow Pattern in
Ventilated Rooms with
Large Depth and Width

Z. Yue, P.V. Nielsen



FLOW PATTERN IN VENTILATED ROOMS WITH LARGE DEPTH AND WIDTH

Zou Yue¹, Peter V. Nielsen²

¹Department of Building Science Engineering, Royal Institute of Technology (KTH), Sweden

²Department of Building Technology and Structural Engineering, Aalborg University, Denmark

ABSTRACT

In many buildings, for instance tunnels, the underground, parking areas and industrial halls, the L/H is so large that the flow pattern induced by a two dimensional supply air jet along the ceiling can be completely different from the one in buildings of normal sizes. Earlier model experiments indicate that, in this case, the supply jet will have a limited penetration length (l_{re}) and the entrainment generates a backward flow in the lower part of a ventilated area and at a given distance this air will disperse or deflect the jet from the inlet openings.

In this paper both model experiments and Computational Fluid Dynamics (CFD) are employed to study the isothermal flow pattern in the ventilated room with different L/H and inlet velocities. The maximum size of the model is $1.4 * 0.72 * 0.0714m$ and the measurement is made by a Laser Doppler anemometer. The CFD simulation is carried out by Flovent code with a $k-\epsilon$ model. Although some discrepancies occur it is clear that the simulation correctly represents the general features of the measurements. Both measurement and CFD simulation show the tendency that l_{re}/H may be independent of W/H when $W/H > 5$. The velocity decay of the wall jet and the maximum velocity in occupied area are also studied in this paper.

KEYWORD

Deep ventilated room, penetration length, CFD, model, wall jet, maximum velocity

INTRODUCTION

In mixing ventilation, wall jets are extensively used for supply of ventilation or conditioned air to rooms and spaces. Before the air enters the occupied area of the room velocities and temperature differences must be decreased to an acceptable level. Considering that the test of supply devices is always done in rooms different from the room where the supply device finally is installed it is of practical interest to study the characterization of flow in rooms of different sizes.

For rooms with small characteristic dimensions ($W/H < 2$, $L/H < 3$), where W , L and H are width, length and height of the ventilation section, respectively, the wall jet undergoes a number of deflections at the corners it meets during its course from the supply to the floor. When the jet is approaching an opposing side (room corner), an adverse pressure gradient is built up and the jet "restarts" again, see Sandberg (1998). When the ratio L/H is larger than a certain value, entrainment implies that the air must be led back along the floor and this will disperse the jet. The distance from the wall with the inlet to the

stagnation point where the flow diverges is called penetration length l_{re} , see figure 1. The penetration length is a significant parameter for proper room air distribution design. At a distance from the supply opening which is larger than l_{re} the velocity is very low since the supply air is distributed over the whole cross area, while the velocities are very high at distances smaller than l_{re} because large volume of air is set into motion by the entrainment below the wall jet. For a normal room the ventilated section should always be smaller than the calculated penetration length so a rotary airflow pattern can be established. However, some applications, such as mining tunnels, require a penetration length as long as possible. Therefore, there is a need for a procedure which allows the penetration length and the velocity distribution to be determined as a function of the room geometry and the inlet condition. The results presented here demonstrate that this can be achieved by the CFD simulation.

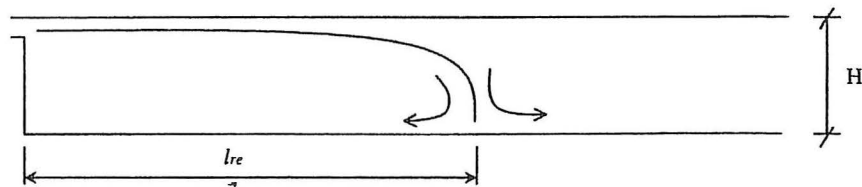


Figure 1: Penetration length of long room.

Experimental investigations of the penetration length can be found in e. g. Urbach (1971), Katz (1974); Förthmann (1934), Nielsen (1976) and Nielsen et al. (1987). With the aid of smoke visualization Urbach (1971) makes tests in a model with $W/H=1$ and found l_{re}/H is about 3 for values of h/H between 0.02 and 0.1 and Reynolds number between 3500 and 12000. Katz's (1974) experiments were also carried out with small W/H but in an open water channel. His tests showed that the penetration length is somewhat dependent on the location of the end wall and l_{re}/H was found between 3 and 4.5. In Förthmann (1934) the velocity profiles in a deep model with $h/H = 0.17$ and $W/H = 3.6$ were measured. Based on these data l_{re}/H was calculated as 5.3. Nielsen et al. (1987) investigated air distribution in rooms with ceiling-mounted obstacles and found that penetration length can be strongly influenced in some cases. Detailed tests of penetration length with different model geometric parameter can be found in Nielsen (1976). Some of them will later be discussed.

EXPERIMENT SETUP AND SIMULATION

The measurements were performed in a model as shown in figure 2 which also indicates the coordinate system used. This model was made from perspex and a full-width slot was placed just below the ceiling. Supply air was led into the model by a ventilator located downstream of the exhaust air terminal device to make sure the tests were held under isothermal conditions. The supply outlet was also preceded by a smooth curved area contraction to produce a uniform velocity profile and reduce the turbulence level. To determine the influence of width and length on the velocity characteristics two different model sizes were applied in the tests. Geometric parameters of these two models are summarized in table 1. For all tests the supply air velocity is 15.1 m/s , corresponding to a Reynolds number about 4000. The measurements of mean velocities and the corresponding turbulence intensities were carried out by a Laser Doppler anemometer. The minimum measurement time for each point is 30 seconds.

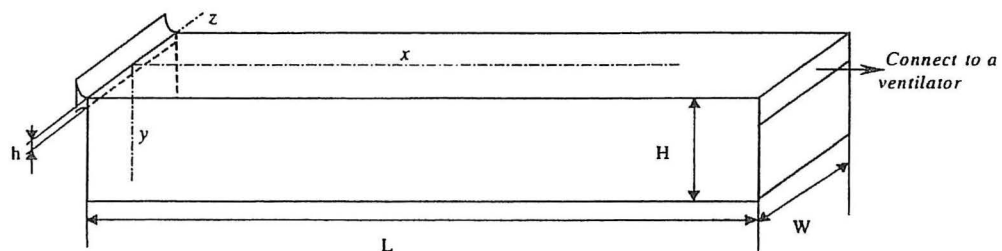


Figure 2: Layout of model

TABLE 1
PARAMETER OF TESTS

	H	h	h/H	L/H	W/H
Model 1	7.14 cm	4mm	0.056	20	10
Model 2	7.14cm	4mm	0.056	15	5

The CFD simulations were carried out under steady state condition. The flow field was divided into a collection of small rectangular cells and the flow at each cell is determined by numerically solving the governing conservation equations of fluid dynamics. The grid density was 80*80*50 grids along the length (x), width (y), height (z), respectively. The grids have smaller spacing at locations where more flow detail are needed (the supply and the jet flow area). Turbulence is modelled by standard $k-\epsilon$ model. For all cases, in the absence of turbulence data, the turbulence intensity at inlet is assumed to be 4% and the turbulence kinetic energy k and its dissipation rate ϵ at the inlet are determined by

$$k = \frac{3}{2} (IU_o)^2 \quad (1)$$

$$\epsilon = \frac{0.09k^{3/2}}{h} \quad (2)$$

where

I = Relative turbulence intensity (-)

U_o = Outlet velocity evaluated from the estimated mass flow (m/s)

RESULTS AND DISCUSSION

Flow pattern and comparison

Figure 3 presents profiles of the longitudinal velocity component at three x and z positions for model 2. As it can be seen, the flow pattern is two-dimensional at $x/H=2.8$. However, the flow is not always symmetric around the mid plane within measurement precision at other positions. This fact indicates that care must be given when making tests which only represent a part of room.

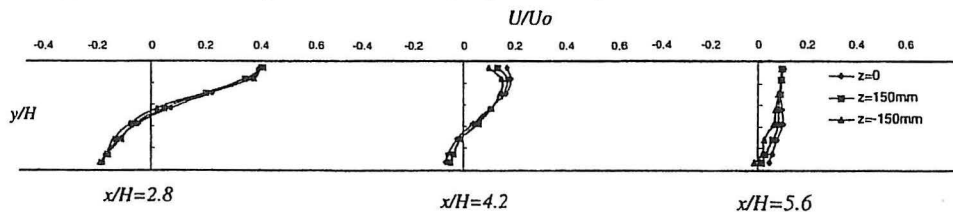


Figure 3: Measured mean velocity profile at different x and z positions

In a model with similar characteristic size ($W/H=4.7$) to model two, Nielsen (1976) observed that an instantaneous flow occurred in the z direction for flow between $x/H = 4.2$ and 5. In our test, such evidence is not found. However, the RMS (Root Mean Square) value of longitudinal velocity component at $y=4\text{mm}$ of model 2 had peak value in this area, see figure 4.

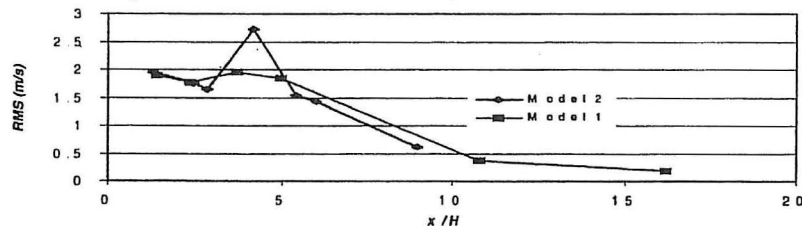


Figure 4: RMS value distribution along the centreline at 4mm from the top for model 1 and 2

Three-dimensional CFD simulations show that the flow patterns in both models are close to be two-dimensional. No evidence of non-symmetry around the mid plane can be found. In figure 5 comparisons of both velocity profile and velocity distribution at ceiling level and floor level along the centre line in the model 1 are presented. Although some discrepancies occur it is clear that the simulation correctly represents the general features of the measurements.

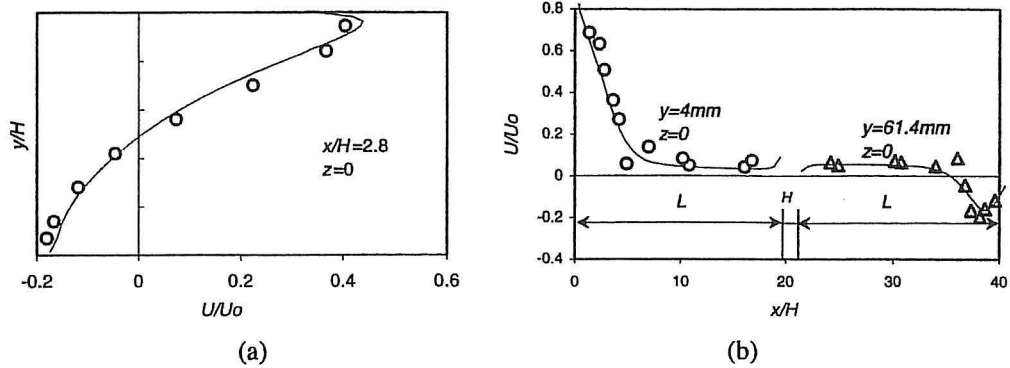


Figure 5: Comparisons in velocity profile (a) and velocity distribution (b) along the centreline in the model 1

Penetration length

Dimensional analysis indicates that the flow in a ventilated room can be described by the geometrical parameters, the Reynolds number Re and the boundary conditions, see Nielsen (1976). Therefore the penetration length can be expressed as:

$$\frac{l_{re}}{H} = f\left(\frac{W}{H}, \frac{L}{H}, \frac{h}{H}, Re\right) \quad (3)$$

where three variables on the right hand side represent the geometry normalized by the height of the room. The Reynolds number is given by

$$Re = \frac{U_o h}{\nu} \quad (4)$$

where ν is the kinematic viscosity

For a fully developed turbulent flow the normalized velocity distribution in ventilated rooms is independent of the Reynolds number, see Nielsen (1976) and Sandberg et al (1998). Figure 6 shows that the normalized return flow in the model expressed by the U/U_o in model 1 at $y = 6.64\text{cm}$ and $x = 20\text{cm}$ is almost independent of the Reynolds number for $Re \geq 4000$.

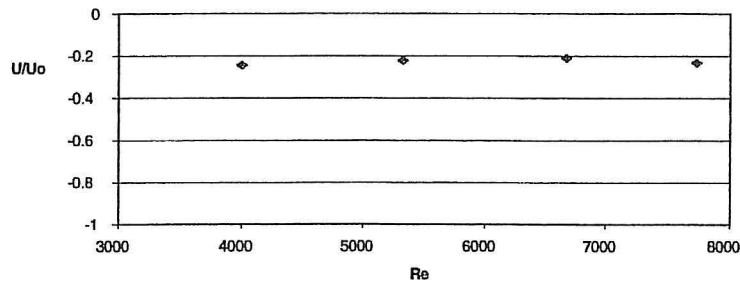


Figure 6: Normalized velocities in model 1 at 5mm from bottom and 20cm from supply outlet for different outlet Reynolds numbers

According to measurement data at 0.5cm from bottom the penetration lengths in both model 1 and model 2 are estimated to be between $4.5 H$ and $5 H$ while CFD calculation gives $5.9 H$ and $5.6 H$ for model 1 and 2, respectively. In figure 7 available penetration length data is presented for different W/H and a best-fitted line for penetration length for all measurements with $h/H = 0.056$ is also given. This line shows a tendency that l_{re}/H may be independent of W/H when $W/H > 5$.

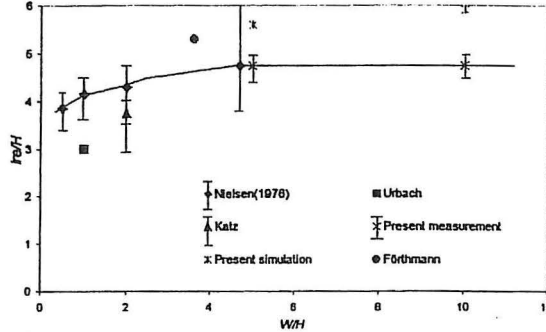


Figure 7: Penetration length versus W/H

Velocity Distribution

In figure 8 the normalized maximum measurement velocities U_m / U_o of model 2 are plotted log-log as a function of the normalized distance from the inlet x/h . The velocity decay has a slope of $-0,5$ in the first part. Obviously this part belongs to the zone of fully established turbulent flow where the maximum velocities can be expressed as:

$$\frac{U_m}{U_o} = K \sqrt{\frac{h}{x + x_o}} \quad (5)$$

where

K = Velocity decay coefficient for wall jet

x_o = Distance to virtual origin

In this test, K and x_o are found to be 3 and 25mm which is in agreement with other experiments, see Malmstrom (1974).

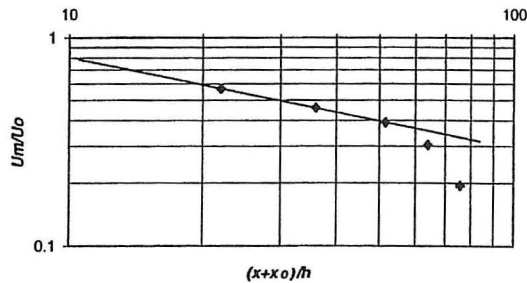


Figure 8: Maximum velocity decay for wall jet

The maximum velocity in the occupied zone U_m is another important parameter in the design of an air distribution system. In a “shorter” room with mixing ventilation this maximum velocity is located close to the floor at $2/3L$ from the inlet, see Zou (1999). Table 2 shows that data of U_m and the locations of

U_{rm} from the inlet x_{rm} are related to the penetration length. It is interesting to note that the measured data of x_{rm}/l_{re} are also close to 2/3.

TABLE 2
MAXIMUM VELOCITY IN OCCUPIED AREA

		U_{rm}/U_o	x_{rm}	l_{re}	x_{rm}/l_{re}
Model 1	Measurement	-0.2	2.8H	4.5H-5H	0.56-0.63
	simulation	-0.17	2.4H	5.9H	0.41
Model 2	Measurement	-0.19	2.8H	4.5H-5H	0.56-0.63
	simulation	-0.17	2.4H	5.6H	0.43

Conclusion

In this paper both model experiments and Computational Fluid Dynamics (CFD) are carried out to study the isothermal flow pattern in the ventilated room with different L/H and inlet velocities. Although some discrepancies occur it is clear that the simulation correctly represents the general features of the measurements. Both measurement and CFD simulation show a tendency that l_{re}/H may be independent of W/H when $W/H > 5$. The maximum velocity decay of the wall jet and the maximum velocity in the occupied area are also studied in this paper.

Acknowledgement

The measurement part of this work is made in cooperation with one student group in Aalborg University, Denmark. The first author would also like to thank Aalborg University that sponsored this project.

REFERENCES

- Förthmann E. (1934). Über turbulente Strahlausbreitung. Ing. Archiv 5.
- Katz P. (1974). Wurfweite, Eindringtiefe und Lauflänge von Zuluftstrahlen im klimatisierten Raum. *HLH* No. 3.
- Malmström T.-G. (1974). Om Funktionerna hos Tilluftsgaller. *TM 49*, Department for Heating and Ventilation, KTH, Stockholm, Sweden.
- Nielsen P.V. (1976). *Flow in Air Conditioned Rooms*, PhD thesis, DTH, Copenhagen
- Nielsen P.V., Evensen L., Grabau P. and Thulesen-Dahl J. H. (1987). Air Distribution in Rooms with Ceiling-Mounted Obstacles and Three-Dimensional Isothermal Flow. *ROOVENT 87*, Stockholm
- Nielsen P.V. (1995). *Lecture Note on Mixing Ventilation*. Dept. of building technology and structural engineering, Aalborg University, Denmark
- Sandberg M. (1998). Some Confinement Effects of Jets in Ventilated Rooms. *ASHRAE Transaction*, SF-98-28-2, pp 1748-1754
- Sforza P.M. (1977). Three-Dimensional Free Jet and Wall Jet: Application to Heating and Ventilation. *ICHMT Congress*, "Heat transfer in buildings", Dubrovnik
- Urbach, D. (1971). *Modelluntersuchungen zur Strahl Lüftung*, Diss. RWTH Aachen
- Zou, Y. (1999). A CFD Study for Airflow Distribution at Floor Level in a Slot-Outlet and Slot-Inlet Ventilation Room. *The 3rd International Symposium on HVAC*, Shenzhen, China



RECENT PAPERS ON INDOOR ENVIRONMENTAL ENGINEERING

PAPER NO. 109: E. Bjørn: *Simulation of Human Respiration with Breathing Thermal Manikin*. ISSN 1395-7953 R9944.

PAPER NO. 110: P. Heiselberg: *Hybrid Ventilation and the Consequences on the Development of the Facade*. ISSN 1395-7953 R0033.

PAPER NO. 111: P. Heiselberg, M. Pedersen, T. Plath: *Local Exhaust Optimization and Worker Exposure*. ISSN 1395-7953 R0034.

PAPER NO. 112: K. Svidt, P. Heiselberg: *Push-Pull Ventilation in a Painting Shop for Large Steel Constructions*. ISSN 1395-7953 R0035.

PAPER NO. 113: P. Heiselberg: *Design Principles for Natural and Hybrid Ventilation*. ISSN 1395-7953 R0036

PAPER NO. 114: A. Andersen, M. Bjerre, Z.D. Chen, P. Heiselberg, Y. Li: *Experimental Study of Wind-Opposed Buoyancy-Driven Natural Ventilation*. ISSN 1395-7953 R0037

PAPER NO. 115: P. Heiselberg, K. Svidt, P.V. Nielsen: *Windows - Measurements of Air Flow Capacity*. ISSN 1395-7953 R0040

PAPER NO. 116: K. Svidt, P. Heiselberg, P.V. Nielsen: *Characterization of the Airflow from a Bottom Hung Window under Natural Ventilation*. ISSN 1395-7953 R0041

PAPER NO. 117: Z. Yue, P.V. Nielsen: *Flow Pattern in Ventilated Rooms with Large Depth and Width*. ISSN 1395-7953 R0042.

PAPER NO. 118: P.V. Nielsen, H. Dam, L.C. Sørensen, K. Svidt, P. Heiselberg: *Characteristics of Buoyant Flow from Open windows in Naturally Ventilated Rooms*. ISSN 1395-7953 R0043.

PAPER NO. 119: P.V. Nielsen, C. Filholm, C. Topp, L. Davidson: *Model Experiments with Low Reynolds Number Effects in a Ventilated Room*. ISSN 1395-7953 R0044.

PAPER NO. 120: P. Lengweiler, P.V. Nielsen, A. Moser, P. Heiselberg, H. Takai: *Experimental Method for Measuring Dust Load on Surfaces in Rooms*. ISSN 1395-7953 R0045.

PAPER NO. 121: L. Davidson, P.V. Nielsen, C. Topp: *Low-Reynolds Number Effects in Ventilated Rooms: A Numerical Study*. ISSN 1395-7953 R0046.

PAPER NO. 122: F. Haghghat, H. Brohus, C. Frier, P. Heiselberg: *Stochastic Prediction of Ventilation System Performance*. ISSN 1395-7953 R0047.

PAPER NO. 123: H. Brohus, F. Haghghat, C. Frier, P. Heiselberg: *Quantification of Uncertainty in Thermal Building Simulation. Part 1: Stochastic Building Model*. ISSN 1395-7953 R0048.

PAPER NO. 124: H. Brohus, F. Haghghat, C. Frier, P. Heiselberg: *Quantification of Uncertainty in Thermal Building Simulation. Part 2: Stochastic Loads*. ISSN 1395-7953 R0049.

Complete list of papers: <http://iee.civil.auc.dk/i6/publ/iee.html>

Flow Part

ISSN 1395-7953 R0042

Dept. of Building Technology and Structural Engineering
Aalborg University, December 2000

Sohngaardsholmsvej 57, DK-9000 Aalborg, Denmark

Phone: +45 9635 8080 Fax: +45 9814 8243

<http://iee.civil.auc.dk>