Investigation of Inlet Boundary Conditions for Numerical Prediction of Air Flow in Livestock Buildings

Svidt, Kjeld

Publication date:
1994

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

INSTITUTTET FOR BYGNINGSTEKNIK
DEPT. OF BUILDING TECHNOLOGY AND STRUCTURAL ENGINEERING
AALBORG UNIVERSITET • AUC • AALBORG • DANMARK

INDOOR ENVIRONMENTAL TECHNOLOGY
PAPER NO. 38

Presented at ROOMVENT '94, Fourth International Conference on Air Distribution in Rooms, June 15-17, 1994, Cracow, Poland

K. SVIDT

INVESTIGATION OF INLET BOUNDARY CONDITIONS FOR NUMERICAL PREDICTION OF AIR FLOW IN LIVESTOCK BUILDINGS

MARCH 1994 ISSN 0902-7513 R9407

INSTITUTTET FOR BYGNINGSTEKNIK

DEPT. OF BUILDING TECHNOLOGY AND STRUCTURAL ENGINEERING
AALBORG UNIVERSITET • AUC • AALBORG • DANMARK

INDOOR ENVIRONMENTAL TECHNOLOGY
PAPER NO. 38

Presented at ROOMVENT '94, Fourth International Conference on Air Distribution in Rooms, June 15-17, 1994, Cracow, Poland

K. SVIDT
INVESTIGATION OF INLET BOUNDARY CONDITIONS FOR NUMERICAL PREDICTION OF AIR FLOW IN LIVESTOCK BUILDINGS

MARCH 1994 ISSN 0902-7513 R9407

INVESTIGATION OF INLET BOUNDARY CONDITIONS FOR NUMERICAL PREDICTION OF AIR FLOW IN LIVESTOCK BUILDINGS

K. Svidt
Aalborg University, Aalborg
Royal Veterinary and Agricultural University, Copenhagen
Denmark

SUMMARY

In modern livestock buildings the design of ventilation systems is important in order to obtain good air distribution. The use of Computational Fluid Dynamics for prediction of the air distribution makes it possible to include the effect of room geometry and heat sources in the design of ventilation systems. This paper presents numerical prediction of air flow compared with laboratory measurements. A commercial air inlet device for livestock buildings was used in the experimental set-up. In the numerical predictions special attention was paid to the specification of boundary conditions for this type of inlet.

INVESTIGATION OF INLET BOUNDARY CONDITIONS FOR NUMERICAL PREDICTION OF AIR FLOW IN LIVESTOCK BUILDINGS

K. Svidt*
Aalborg University, Aalborg
Royal Veterinary and Agricultural University, Copenhagen
Denmark

INTRODUCTION

In modern livestock buildings the design of ventilation systems is important in order to obtain a good air distribution. The use of Computational Fluid Dynamics (CFD) for prediction of the air distribution makes it possible to include the effect of room geometry and heat sources on the design of buildings and ventilation systems.

The application of CFD to air flow in livestock buildings has been examined by a few authors. Choi et al. [1,2,3] used a two-dimensional model to investigate the distribution of velocity and contaminants as well as the effects of buoyancy and obstacles. Krause and Janssen [4] used a two-dimensional model to study how different air flow patterns could be used as a tool to minimize ammonia emission from livestock buildings. Hoff et al. [5] investigated three-dimensional effects of buoyant flow in a slot-ventilated enclosure. The modelling of dust and particle transport was described by Maghirang et al. [6]. Harral [7] studied air velocities and turbulence intensity, and Svidt [8] studied the effect of obstacles and buoyant flow.

This paper presents numerical prediction of air flow compared with full-scale laboratory measurements. A commercial air inlet device for livestock buildings was used in the experimental setup. In the numerical predictions special attention was paid to the specification of boundary conditions for this type of inlet.

---

*The author works at Aalborg University, Department of Building Technology and Structural Engineering. The research project is managed by the Royal Veterinary and Agricultural University in Copenhagen. The project is sponsored by the Danish Agricultural and Veterinary Research Council.

EXPERIMENTAL FACILITIES

Experiments were carried out in a full-scale test room which was 8 m long, 6 m wide and 3 m high (figure 1). A commercial inlet device for livestock buildings, DSI-WV1500, was mounted in the centre of the end wall 1.03 m below the ceiling. The inlet consists of a rectangular hole in the wall (55 x 27 cm) with a bottom-mounted, adjustable flap to control the opening area and the angle of the incoming air.

Heat could be supplied through electric heating wires on the floor. In case of non-isothermal conditions, the temperature in the surrounding laboratory was kept close to the mean temperature in the test room to minimize heat loss through the walls.

Variable speed fans controlled the inlet and outlet air flow to maintain a slight over-pressure in the test room which minimized the effect of air leakage.

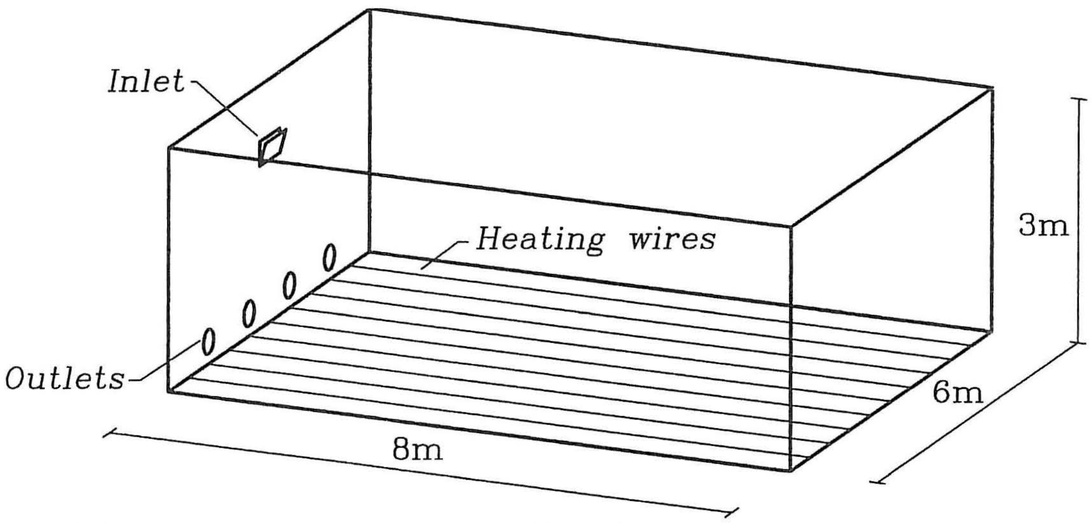

Fig. 1. Experiments were carried out in a full-scale test room. One inlet and four outlets were mounted in the end wall. To create non-isothermal conditions heat could be supplied through electric heating wires on the floor.

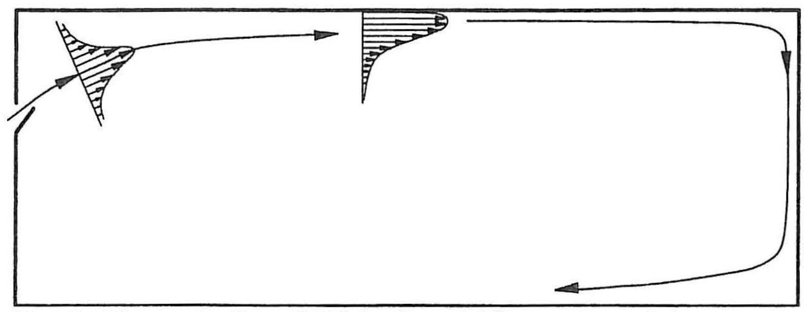

Fig. 2. The main flow field in the middle section of the room.

Figure 2 shows the main flow field in the middle section of the room. Air enters the room through the inlet in an angle that is determined by the position of the flap. The incomming air forms a free jet until it reaches the ceiling where it turns into a three-dimensional wall jet.

SIMULATION MODEL

The numerical predictions were performed by a CFD-code that solved the Navier-Stokes equations using the standard k-ϵ turbulence model. These equations and the turbulence model have been described by several authors, e.g. Launder and Spalding [9] and Rodi [10].

Equations are solved for air pressure, three velocity composants, air temperature and the turbulent quantities k and ϵ. The governing differential equations can all be written in the following general form:

$$\rho\left(u\frac{\partial\phi}{\partial x} + v\frac{\partial\phi}{\partial y} + w\frac{\partial\phi}{\partial z}\right) = \left(\mu_l + \mu_t\right)\left(\frac{\partial^2\phi}{\partial x^2} + \frac{\partial^2\phi}{\partial y^2} + \frac{\partial^2\phi}{\partial z^2}\right) + S_\phi \tag{1}$$

where
- ρ is the air density
- u is the velocity in the x - direction
- v is the velocity in the y - direction
- w is the velocity in the z - direction
- S_ϕ is a source term of the variable ϕ
- ϕ can be any of the variables to be solved
- μ_l is the constant laminar viscosity of the air
- μ_t is the turbulent viscosity of the air

The left-hand side of eq. (1) represents convection and the right-hand side represents diffusion and source terms. More details about the governing equations are described by Schlichting [11].

In the standard k-ϵ turbulence model the local value of the turbulent viscosity is defined as:

$$\mu_t = \rho\, c_\mu \frac{k^2}{\epsilon} \tag{2}$$

where
- c_μ is an empirical constant
- k is the turbulent kinetic energy
- ϵ is the dissipation of turbulent kinetic energy

The equations were solved by iteration using the SIMPLE algorithm as described by Patankar [12].

INLET BOUNDARY CONDITIONS

A direct description of the inlet boundary conditions is an easy-to-use method because the inlet is described in a way that is close to the actual geometry of the inlet, but because the inlet is small compared with the total solution domain which is the entire room, it can be difficult to obtain a satisfactory grid resolution near the inlet. These problems can be reduced by using the prescribed velocity method where the air velocity is prescribed in a larger volume in front of the inlet. It is described below how these methods are implemented in the present case.

Direct description of the inlet boundary conditions

In this case the inlet is represented by an opening that is very close to the actual geometry of the inlet.

The actual inlet air velocity was measured in 20 positions across the inlet area and the average inlet velocity was specified at the inlet boundary. The effective inlet area was determined from the average inlet velocity and the measured air flow rate. The effective inlet area was specified as a rectangular opening as close as possible to the actual opening which was not exactly rectangular. By specifying the measured velocity and the effective inlet area it was ensured that the correct momentum was supplied to the room.

Prescribed velocity method

With the prescribed velocity method the air velocity is prescribed in a volume in front of the inlet. The method was first described by Gosman et al. [13] and it has later been used by several authors as described by Nielsen [14].

In the present case it was found that the flow along the ceiling could be described as a three-dimensional wall jet characterised by a certain velocity decay and different spreading rates in the horizontal and the vertical directions.

The maximum velocity u_x at the distance x from the inlet wall is given by:

$$\frac{u_x}{u_0} = K_a \frac{\sqrt{a_0}}{x + x_0} \qquad (3)$$

where
- u_0 is the inlet air velocity
- K_a is a constant that depends on the inlet geometry
- a_0 is the effective inlet area
- x_0 is the distance to the virtual origin of the jet

Inverting equation (3) gives:

$$\frac{u_0}{u_x} = \frac{1}{K_a \sqrt{a_0}} \left(x + x_0\right) \tag{4}$$

Equation (4) shows that a plot of u_0/u_x versus x as shown in figure 3 can be used to determine x_0 and K_a from the intersection with the x-axis and the slope of the regression line. Furthermore, the plot shows that the experimental data actually fit into this model. In this case it is found that $x_0 = -1.7\ m$ and $K_a = 3.1$.

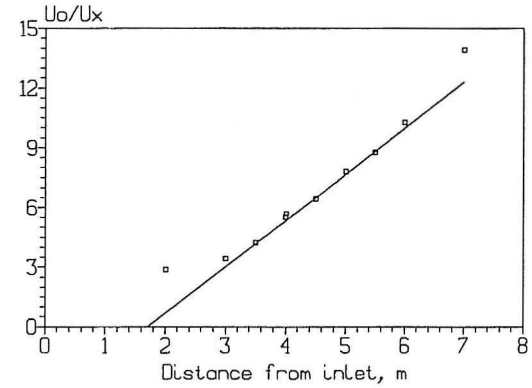

Fig. 3. The jet constants, K_a and x_0 are determined graphically by the intersection and the slope of the regression line.

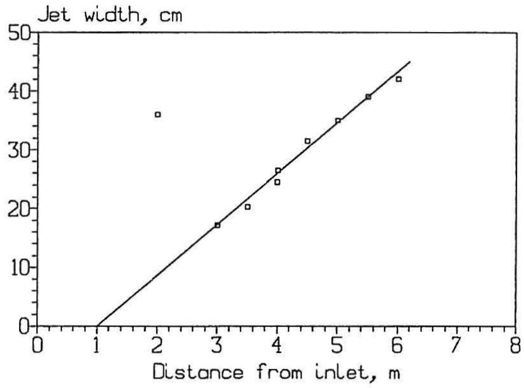

Fig. 4. Graphical description of the vertical spreading rate.

The vertical jet width δ_y is defined as the distance from the ceiling to the position where the air velocity is half of the maximum velocity u_x. The jet width at the distance x from the inlet wall is given by:

$$\delta_y = D_{a,y} \left(x + x_{0,y}\right) \tag{5}$$

In figure 4 it is found by linear regression that $x_{0,y} = -1.0$ m and $D_{a,y} = 0.087$. In the same way it was found that the horizontal spreading of the jet was characterised by $x_{0,z} = -1.0$ m and $D_{a,z} = 0.610$.

At the distance x from the inlet wall the velocity u_y at the distance y from the ceiling was prescribed by the profile given by Verhoff [15]:

$$\frac{u_y}{u_x} = 1.48 \left(\frac{y}{\delta_y}\right)^{1/7} \left(1 - erf\left(0.678\ \frac{y}{\delta_y}\right)\right) \tag{6}$$

At each distance y the velocity u_z at the horizontal distance z from the centre line was given by a Gaussian profile:

$$\frac{u_z}{u_y} = \exp\left(-\frac{1}{2}\left(\frac{z}{\sigma}\right)^2\right) \qquad (7)$$

where
$$\sigma = \frac{\delta_z}{\sqrt{2\ \ln(2)}}$$

The prescribed velocity method was implemented in the CFD-code in such a way that the velocity would only be prescribed in a volume inside the width of the jet as defined above. In the x-direction the velocity was prescribed in a range where the figures 3 and 4 showed that the model would match the experimental data.

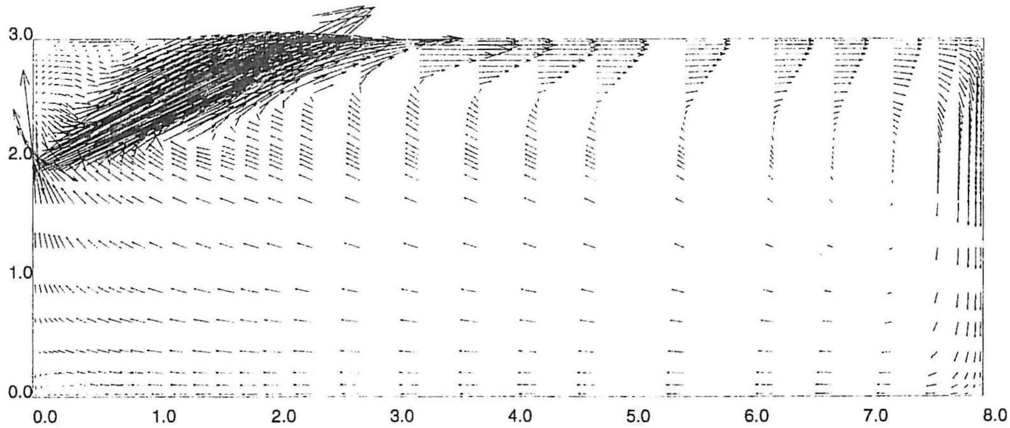

Fig. 5. The calculated flow field in the middle section of the room. The inlet is located to the left about 2 m above the floor.

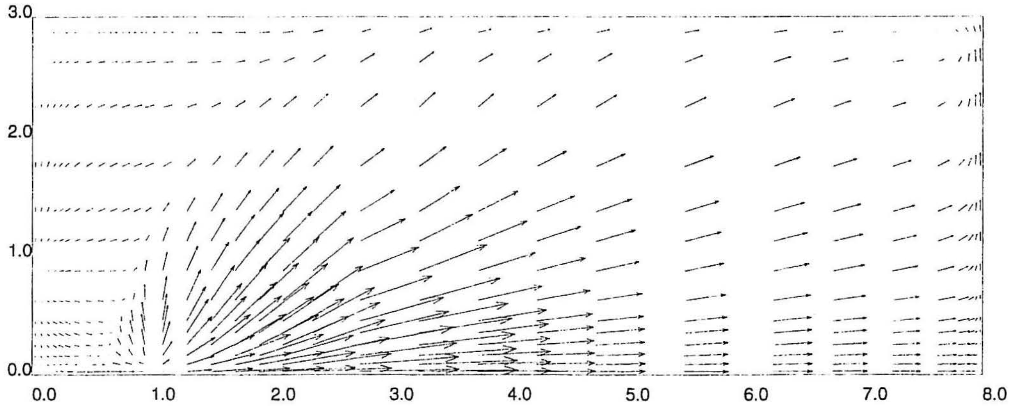

Fig. 6. The calculated flow field in a horizontal plane near the ceiling shows a semi-radial flow field with its origin near the point where the incoming jet hits the ceiling. (Only one half of the room is shown).

RESULTS

Isothermal conditions

Figure 5 shows the calculated flow field in the middle section of the room. It shows how the free jet of incoming air attaches to the ceiling and continues as a wall jet. Figure 6 shows a top view of the situation where it appears that the jet results in a semi-radial flow after having reached the ceiling. However, it can still be regarded as a three-dimensional wall jet with a large spreading rate as shown in the previous section.

In a case with direct description of the inlet boundary conditions, the calculated velocity decay in the jet and in the occupied zone is compared with experimental results in figure 7. It appears that the air velocity in general is underestimated and that the shape of the velocity decay in the occupied zone does not match the experimental results very well. The maximum velocity in the occupied zone was underestimated by 27 %.

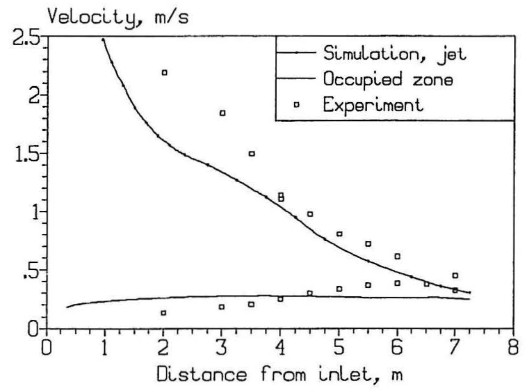

Fig. 7. Measured and simulated air velocities with direct description of the inlet boundary conditions.

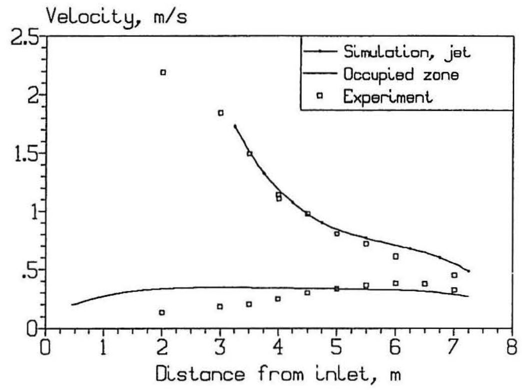

Fig. 8. Measured and simulated air velocities with the prescribed velocity method.

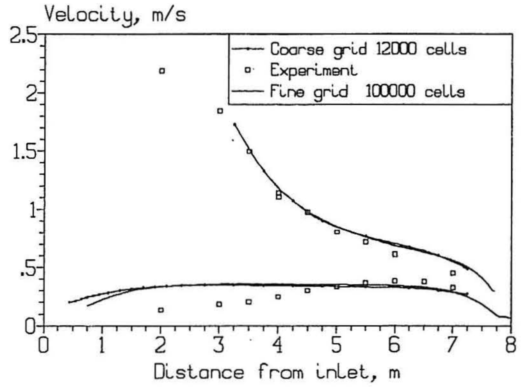

Fig. 9. Fine grid and coarse grid simulations compared with experimental results.

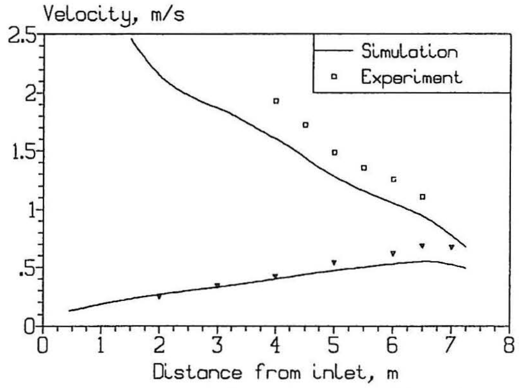

Fig. 10. Measured and simulated air velocities under non-isothermal conditions in the jet and the occupied zone.

Figure 8 shows the result after introducing the prescribed velocity method. Now the max-imum velocity in the occupied zone is underestimated by 9 % which clearly is better than in the first case, but still the shape of the velocity decay in the occupied zone is far from the experimental results.

The simulations in figs. 7 and 8 were performed with a 34·31·14 grid which gives approximately 15000 control volumes. To check if the solution was grid-independent a 58·53·33 grid with approximately 100000 control volumes was tested on the prescribed velocity case. The extra 85000 control volumes were mainly supplied in the lower part of the occupied zone and near the end wall where the wall jet is deflected down into the occupied zone. In figure 9 the results of the fine grid and the coarse grid are compared. The figure shows that the fine grid changes the result slightly in the right direction (now the maximum velocity in the occupied zone is underestimated by 6.5 %) but the changes are small considering the large increase in the number of grid points.

Non-isothermal conditions

A non-isothermal case was studied with direct description of the inlet boundary conditions. Heat was supplied through the heating wires (figure 1) to obtain a temperature difference of 10 °C between the inlet air and the outlet air. The Archimedes number was still so low that the air jet would remain attached to the ceiling in the full length of the room as shown in figure 2.

The experiments and the simulations showed that the three-dimensional wall jet at the ceiling was strongly affected by the heat source. The top view in figure 11 shows that the jet has a smaller spreading rate than the isothermal jet in figure 6. Measured and calculated air velocities are compared in figure 10. Again it can be seen that the velocities are generally underestimated. The maximum velocity in the occupied zone is underestimated by 20 % but in this case the shape of the curve in the occupied zone gets much closer to the experimental results than it did in the isothermal cases.

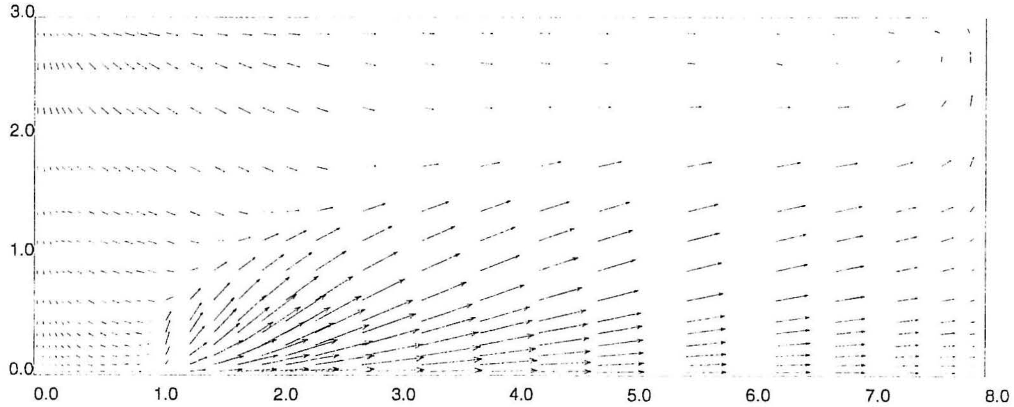

Fig. 11. The calculated flow field in a horizontal plane near the ceiling under non-isothermal conditions shows that the air flow pattern is different from the isothermal case shown in fig. 6.

CONCLUSIONS

Numerical prediction of the air flow using a CFD-code with the standard k-ϵ model was compared with experimental results from a full scale laboratory. The predicted air flow patterns were in good agreement with smoke experiments in the laboratory, but there were some problems in predicting the velocity distribution in the occupied zone of the room. Different inlet boundary conditions and grid refinements were tested.

Under isothermal conditions it was found that the prescribed velocity method would predict the maximum velocity in the occupied zone significantly better than a method with direct description of the inlet boundary conditions. The increase of the number of grid points from 15000 to 100000 would again increase the accuracy a little but the changes were small considering the large increase in the number of grid points.

Even though the prediction of the maximum velocity in the occupied zone was improved, the velocity distribution in the occupied zone was still not very well predicted in the isothermal case.

In a case with non-isothermal flow the predicted velocity distribution in the occupied zone was better than in the isothermal case, but still the velocities were underestimated.

REFERENCES

[1] Choi, H.L., L.D. Albright, M.B. Timmons and Z. Warhaft, "**Air Velocity and Contaminant Distribution in a Slot-Ventilated Enclosure**". ASAE Paper 87-4036.

[2] Choi, H.L., L.D. Albright, M.B. Timmons, "**An Application of the K-Epsilon Turbulence Model to Predict how a Rectangular Obstacle a Slot-Ventilated Enclosure Affects Air Flow**". Transactions of the ASAE, Vol. 31 (6), November-December 1988.

[3] Choi, H.L. and K. Hyeon-Tae, "**An Application of K-Epsilon Turbulence model to Predict How a Rectangular Obstacle with Heat Flux in a Slot-Ventilated Enclosure Affects Air Flow**". International Symposium on Room Air Convection and Ventilation Effectiveness, University of Tokyo, July 22 - 24, 1992.

[4] Krause, K.H. and J. Janssen, "**Measuring and Simulation of the Distribution of Ammonia in Animal Houses**". Proceedings, Roomvent'90, International Conference on Engineering Aero- and Thermodynamics of Ventilated Rooms, Oslo 1990.

[5] Hoff, S.J., K.A. Janni and L.D. Jacobson, "**Three-Dimensional Buoyant Turbulent Flows in a Scaled Model, Slot-Ventilated, Livestock Confinement Facility**". Transactions of the ASAE, Vol. 35 (2), March-April 1992.

[6] Maghirang, R.G and H.B. Manbeck, "**Modelling Particle Transport in Slot-Inlet Ventilated Airspaces**". Transactions of the ASAE, Vol. 36(5):1449-1459, September-October 1993.

[7] Harral, B.B and C.R. Boon, "**Modelling the Air Flow and Environment in a Pig House**". Proceedings of the Fourth International Livestock Environment Symposiom, Coventry, England 1993.

[8] Svidt, K. "**Numerical Prediction of Buoyant Air Flow in Livestock Buildings**". Paper no. 37. Dept. of Build. Tech. and Struct. Engineering, Aalborg University.

[9] Launder, B.E. and D.B. Spalding, "**The Numerical Computation of Turbulent Flows**". Computer Methods in Applied Mechanics and Engineering 3, pp 269-289, 1974.

[10] Rodi, W. "**Turbulence Models and their Application in Hydraulics - a State of the Art Review**". IAHR 1984.

[11] Schlichting, H. "**Boundary-Layer Theory**". McGraw-Hill 1979.

[12] Patankar, S.V. "**Numerical Heat Transfer and Fluid Flow**". Hemisphere Publishing Corporation 1980.

[13] 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 102: 316-322. 1980.

[14] Nielsen, P.V. "**Description of Supply Openings in Numerical Models for Room Air Distribution**". ASHRAE Transactions, Vol 98, Part 1, 1992.

[15] Verhoff, A. "**The Two-Dimensional Turbulent Wall Jet without an External Free Stream**". Report no. 626, Princeton University, 1963.

PAPERS ON INDOOR ENVIRONMENTAL TECHNOLOGY

PAPER NO. 8: Peter V. Nielsen, Lars Hoff, Lars Germann Pedersen: *Displacement Ventilation by Different Types of Diffusers.* ISSN 0902-7513 R8834.

PAPER NO. 9: Per Heiselberg, Peter V. Nielsen: *Flow Conditions in a Mechanically Ventilated Room with a Convective Heat Source.* ISSN 0902-7513 R8835.

PAPER NO. 10: Peter V. Nielsen: *Displacement Ventilation in a Room with Low-Level Diffusers.* ISSN 0902-7513 R8836.

PAPER NO. 11: Peter V. Nielsen: *Airflow Simulation Techniques - Progress and Trends.* ISSN 0902-7513 R8926.

PAPER NO. 12: M. Skovgaard, C. E. Hyldgaard & P. V. Nielsen: *High and Low Reynolds Number Measurements in a Room with an Impinging Isothermal Jet.* ISSN 0902-7513 R9003

PAPER NO. 13: M. Skovgaard, P. V. Nielsen: *Numerical Prediction of Air Distribution in Rooms with Ventilation of the Mixing Type using the Standard* K, ε-*Model.* ISSN 0902-7513 R9042.

PAPER NO. 14: P. Kofoed, P. V. Nielsen: *Thermal Plumes in Ventilated Rooms - Measurements in Stratified Surroundings and Analysis by Use of an Extrapolation Method.* ISSN 0902-7513 R9043.

PAPER NO. 15: P. Heiselberg, M. Sandberg: *Convection from a Slender Cylinder in a Ventilated Room.* ISSN 0902-7513 R9044.

PAPER NO. 16: C. E. Hyldgaard: *Water Evaporation in Swimming Baths.* ISSN 0902-7513 R9045.

PAPER NO. 17: H. Overby, M. Steen-Thøde: *Calculation of Vertical Temperature Gradients in Heated Rooms.* ISSN 0902-7513 R9046.

PAPER NO. 18: P. V. Nielsen, U. Madsen, D. Tveit: *Experiments on an Exhaust Hood for the Paint Industry.* ISSN 0902-7513 R9146.

PAPER NO. 19: L. Germann Pedersen, P. V. Nielsen: *Exhaust System Reinforced by Jet Flow.* ISSN 0902-7513 R9147.

PAPER NO. 20: P. V. Nielsen: *Models for the Prediction of Room Air Distribution.* ISSN 0902-7513 R9148.

PAPER NO. 21: M. Skovgaard, P. V. Nielsen: *Modelling Complex Inlet Geometries in CFD - Applied to Air Flow in Ventilated Rooms.* ISSN 0902-7513 R9149.

PAPER NO. 22: M. Skovgaard, P. V. Nielsen: *Numerical Investigation of Transitional Flow over a Backward Facing Step using a Low Reynolds Number* $k-\varepsilon$ *Model.* ISSN 0902-7513 R9150.

PAPER NO. 23: P. Kofoed, P. V. Nielsen: *Auftriebsströmungen verschiedener Wärmequellen - Einfluss der umgebenden Wände auf den geförderten Volumenstrom.* ISSN 0902-7513 R9151.

PAPERS ON INDOOR ENVIRONMENTAL TECHNOLOGY

PAPER NO. 24: P. Heiselberg: *Concentration Distribution in a Ventilated Room under Isothermal Conditions.* ISSN 0902-7513 R9152.

PAPER NO. 25: P. V. Nielsen: *Air Distribution Systems - Room Air Movement and Ventilation Effectiveness.* ISSN 0902-7513 R9250.

PAPER NO. 26: P. V. Nielsen: *Description of Supply Openings in Numerical Models for Room Air Distribution* ISSN 0902-7513 R9251.

PAPER NO. 27: P. V. Nielsen: *Velocity Distribution in the Flow from a Wall-mounted Diffuser in Rooms with Displacement Ventilation.* ISSN 0902-7513 R9252.

PAPER NO. 28: T. V. Jacobsen & P. V. Nielsen: *Velocity and Temperature Distribution in Flow from an Inlet Device in Rooms with Displacement Ventilation.* ISSN 0902-7513 R9253.

PAPER NO. 29: P. Heiselberg: *Dispersion of Contaminants in Indoor Climate.* ISSN 0902-7513 R9254.

PAPER NO. 30: P. Heiselberg & N. C. Bergsøe: *Measurements of Contaminant Dispersion in Ventilated Rooms by a Passive Tracer Gas Technique.* ISSN 0902-7513 R9255.

PAPER NO. 31: K. S. Christensen: *Numerical Prediction of Airflow in a Room with Ceiling-Mounted Obstacles.* ISSN 0902-7513 R9256.

PAPER NO. 32: S. G. Fox & P. V. Nielsen: *Model Experiments in 1990 and On-Site Validation in 1992 of the Air Movement in the Danish Pavilion in Seville.* ISSN 0902-7513 R9335.

PAPER NO. 33: U. Madsen, N. O. Breum & P. V. Nielsen: *Local Exhaust Ventilation - a Numerical and Experimental Study of Capture Efficiency.* ISSN 0902-7513 R9336.

PAPER NO. 34: T. V. Jacobsen, P. V. Nielsen: *Numerical Modelling of Thermal Environment in a Displacement-Ventilated Room.* ISSN 0902-7513 R9337.

PAPER NO. 35: P. Heiselberg: *Draught Risk from Cold Vertical Surfaces.* ISSN 0902-7513 R9338.

PAPER NO. 36: P. V. Nielsen: *Model Experiments for the Determination of Airflow in Large Spaces.* ISSN 0902-7513 R9339.

PAPER NO. 37: K. Svidt: *Numerical Prediction of Buoyant Air Flow in Livestock Buildings.* ISSN 0902-7513 R9351.

PAPER NO. 38: K. Svidt: *Investigation of Inlet Boundary Conditions Numerical prediction of Air Flow in Livestock Buildings.* ISSN 0902-7513 R9407.

Department of Building Technology and Structural Engineering
The University of Aalborg, Sohngaardsholmsvej 57. DK 9000 Aalborg
Telephone: 45 98 15 85 22 Telefax: 45 98 14 82 43