INSTITUTTET FOR BYGNINGSTEKNIK

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

INDOOR ENVIRONMENTAL TECHNOLOGY
PAPER NO. 37

Presented at the Fourth International Livestock Environment Symposium, Coventry, England, July 6-9, 1993

K. SVIDT

NUMERICAL PREDICTION OF BUOYANT AIR FLOW IN LIVESTOCK BUILDINGS

DECEMBER 1993

ISSN 0902-7513 R9351

NUMERICAL PREDICTION OF BUOYANT AIR FLOW IN LIVESTOCK BUILDINGS

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

ABSTRACT

In modern livestock buildings air distribution and air quality are important parameters to animal welfare and to the health of full-time employees in animal production. Traditional methods for calculating air distribution in farm buildings are mainly based on formulas for air jets which do not include the effect of room geometry, obstacles or heat sources. This paper describes the use of Computational Fluid Dynamics to predict air flow patterns and temperature distribution in a ventilated space. Good agreement is found when results of numerical predictions are compared with experimental data.
KEYWORDS: Simulation, air flow, obstacles, buoyancy.

INTRODUCTION

During the last decades livestock production has developed rapidly towards larger and more intensive production systems. In modern production systems air quality is important to animal welfare and important to the health of people who are full-time employed in animal production. Under these conditions design of proper ventilation systems is getting increasingly important.

Traditional methods for calculating air distribution in farm buildings are mainly based on formulas for free jets or wall jets. Using these methods it is not possible to include the effect of room geometry or the effect of obstacles on air flow patterns. Furthermore, traditional methods provide no possibilities to calculate important parameters such as contaminant concentration and ventilation efficiency for the ventilation system.

Computational Fluid Dynamics (CFD) is a technique for calculating air velocities and temperature distribution in a ventilated space. In principle, the ventilated space is divided into a number of small control volumes. In the centre of each control volume air velocities, pressure and temperature are calculated. The solution is determined by the boundary conditions which include room geometry, position of air inlet and outlet, inlet air velocity and air temperature and position of heat sources and obstacles.

Once the air flow field is described in this way, it is possible to calculate contaminant distribution in the room provided boundary conditions for the contaminant sources are known.

---

*In previous publications the author has used the name *K.S. Christensen*. The author works at Aalborg University on a research project governed by the Royal Veterinary and Agricultural University in Copenhagen. The project is sponsored by the Danish Agricultural and Veterinary Research Council.

The principles of CFD have been used for about two decades (Launder and Spalding, 1974), (Nielsen, 1974). Today the required computer power is available to numerous researchers and many people are investigating the use of CFD for ventilation purposes.

The application of CFD to air flow in livestock buildings has been examined by a few authors. Choi et al. (1987, 1988, 1990, 1992) 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 (1990) 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. (1992) investigated three-dimensional effects of buoyant flow in a slot-ventilated enclosure.

MATHEMATICAL MODEL

A complete description of non-isothermal turbulent air flow in a room requires a number of partial differential equations. The governing differential equations can all be written in the following general form:

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

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 (1979).

The turbulent viscosity describes the effects of turbulence on the air flow. The local value of the turbulent viscosity should be determined in each point of the solution domain by a turbulence model. Rodi (1984) describes the principles of different turbulence models. In this project the standard k,ϵ turbulence model was used, in which the local value of turbulent viscosity is defined as:

$$\mu_t = \rho\, c_\mu \frac{k^2}{\epsilon} \qquad (2)$$

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

Like air velocities and air temperature the additional variables k and ϵ should be calculated in each point of the solution domain, which is done by equations of same type as eq. (1) with different source terms. After introducing these variables the equation system to be solved consists of 7 partial differential equations with seven unknowns. The unknowns are the velocities u , v and w, air pressure p, temperature T, and the turbulent quantities k and ε. Techniques for numerical solution of the equations are described by Patankar 1980.

RESULTS

The first test of the model was made on an isothermal two-dimensional model with a ceiling-mounted obstacle. The two-dimensional test case is shown in figure 1. The room has a length L of three times the height H. The inlet is in the upper left corner of the room and the outlet is in the lower left corner. Inlet height is $h = 0.02\ H$. The inlet jet will form a horizontal two-dimensional wall jet. A ceiling-mounted obstacle of height f is placed at the distance x_f from the inlet. Values of $x_f/h = 10, 30$ and 60 are used in this paper.

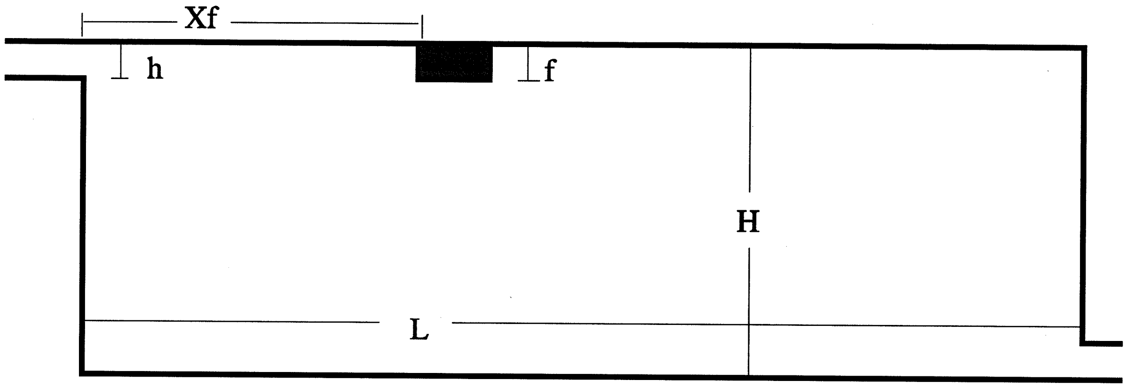

Figure 1. Geometry of the test case.

Nielsen (1983) presented experimental results of this test case. Velocity profiles were measured in the wall jet downstream of the obstacle in the distance $x = 1.7\ H$ from the inlet and also in the occupied zone at the distance $x = 2.1\ H$ from the inlet. Figure 2 shows calculated velocity vectors of a simulation where it can be seen how the inlet wall jet reattaches to the ceiling downstream of the obstacle.

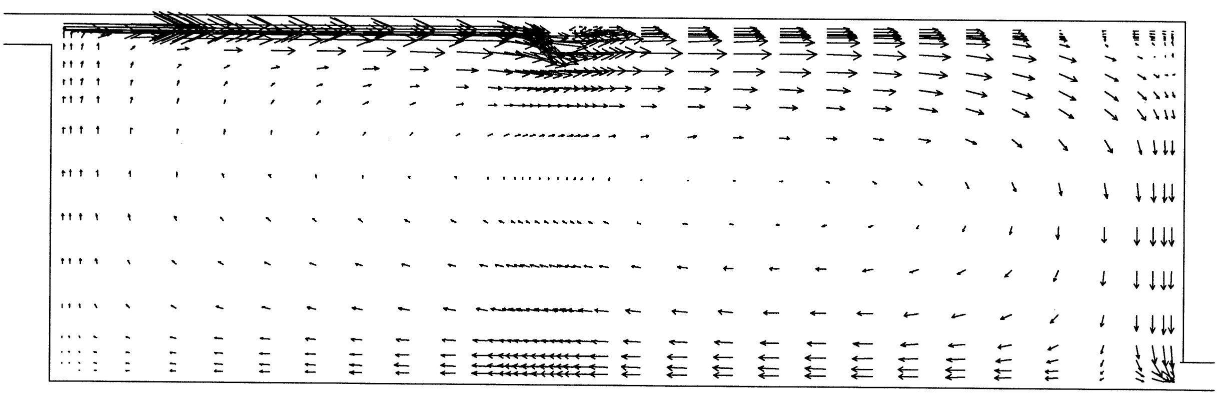

Figure 2. Calculated isothermal flow field with a ceiling-mounted obstacle.

In figure 3 calculated velocity profiles for different positions of the obstacle are compared with the experimental data of Nielsen (1983). It appears that good agreement is obtained, except when the obstacle is placed very close to the inlet. More details on this case are described by Christensen (1992).

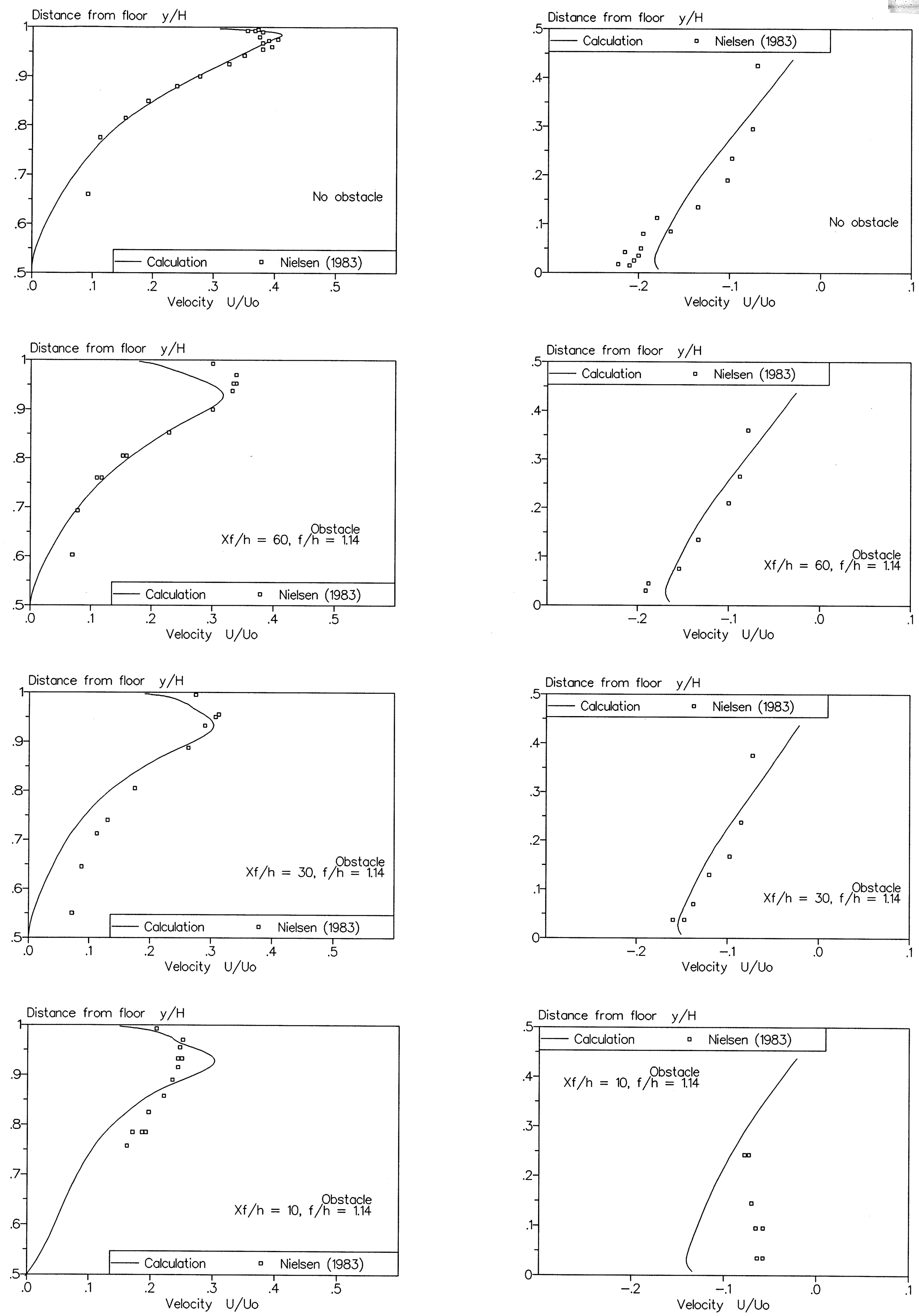

Figure. 3. Calculated and measured velocity profiles in the jet (left) and in the occupied zone (right).

Next step in the model validation was to include the temperature equation and buoyancy forces to predict non-isothermal air flow in a two-dimensional test case with a heat source evenly distributed along the floor and the same dimensions as in the previous case. In this case there were no obstacles. According to the test case specifications (Nielsen 1990) simulations should be carried out with increased Archimedes number until a reduced penetration depth of the inlet air jet is obtained.

Simulations carried out with a two-dimensional model were not able to predict a reduced penetration depth. Below a certain Archimedes number the inlet air jet would always penetrate to the end wall of the room (figure 4) and above this Archimedes number the inlet air would fall down immediately after entering the room (figure 5).

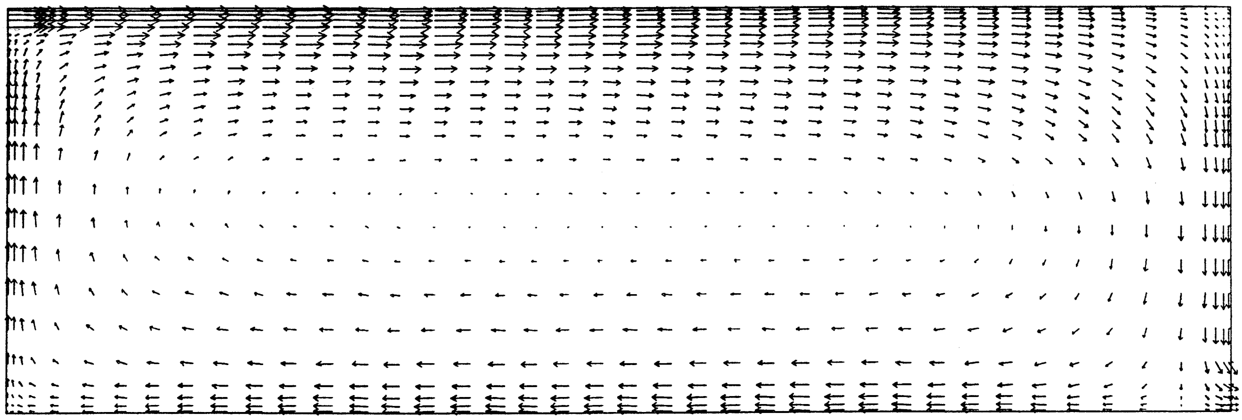

Figure 4. Below a certain Archimedes number the inlet air jet penetrates to the end wall.

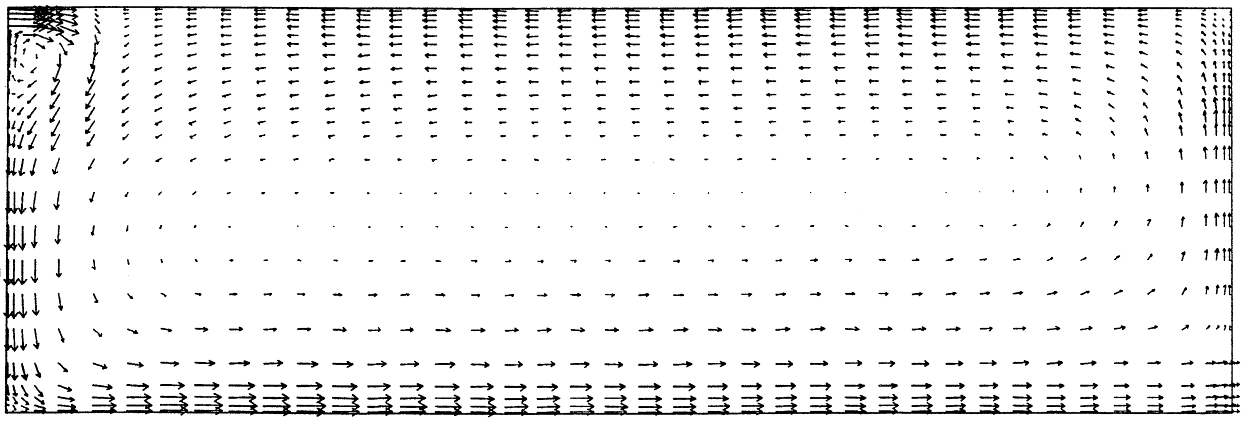

Figure 5. At higher Archimedes numbers the inlet air falls down directly from the inlet opening.

Now the three-dimensional simulation model was set up with the boundary conditions of the two-dimensional test case. It is shown in figure 6 that this model is able to predict a reduced penetration depth of the air jet. Furthermore the simulations show that there may be some instabilities and some three-dimensional effects in this case. Figure 7 shows an example of a calculated flow field in a horizontal plane at the same height as the inlet and figure 8 shows the corresponding flow field a few centimetres above the floor. Figures 6 - 8 show the three-dimensional nature of the flow. Three-dimensional effects in a similar test case with two-dimensional boundary conditions were also reported by Hoff (1992).

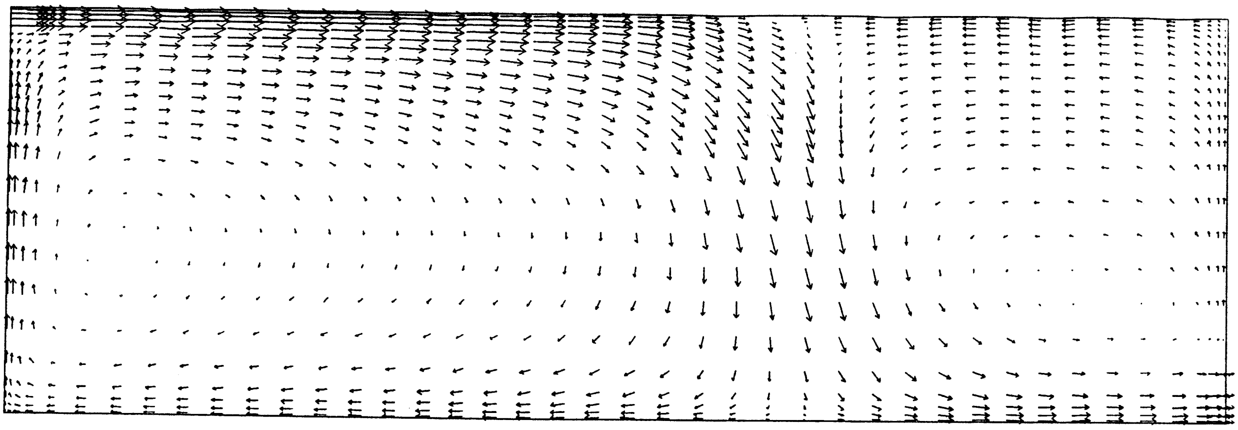

Figure 6. A reduced penetration depth was predicted by a three-dimensional simulation model.

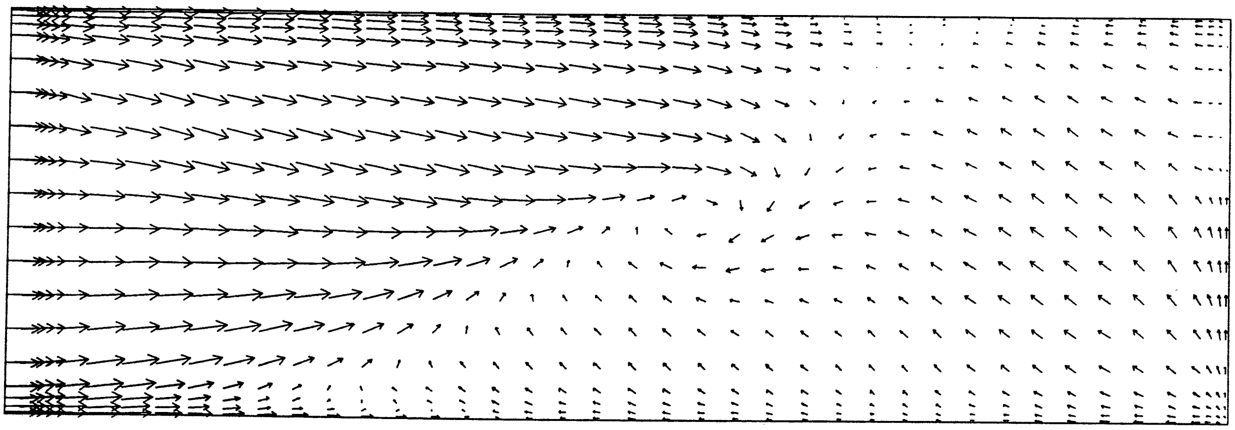

Figure 7. The simulated flow field in a horizontal plane near the ceiling.

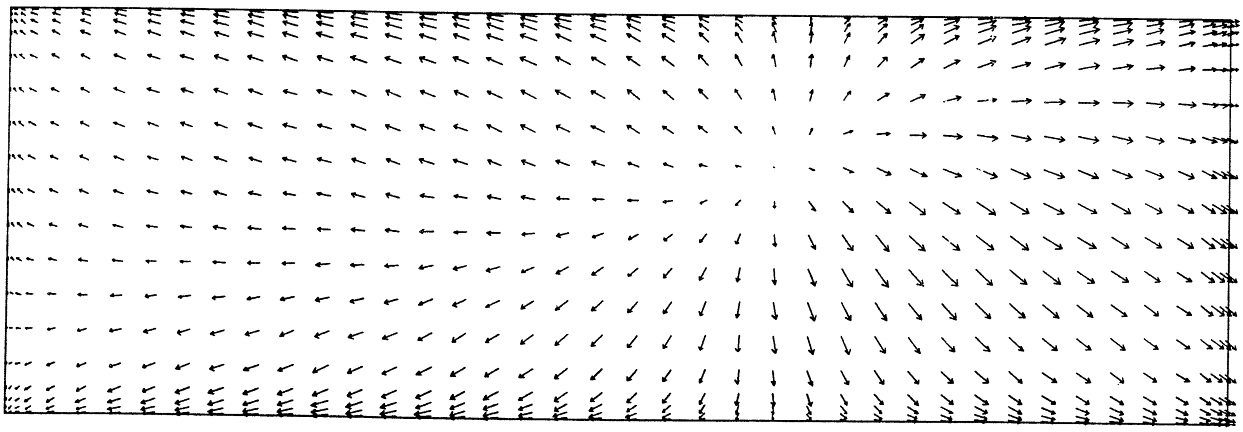

Figure 8. The simulated flow field in a horizontal plane near the floor.

CONCLUSIONS

The numerical prediction of air flow in livestock buildings has been investigated by using a test case with ceiling-mounted obstacles such as light fittings or ceiling beams which are often found in livestock buildings. Predicted velocity profiles were in good agreement with experimental results for this test case.

Using a three-dimensional simulation model including the temperature equation and buoyancy forces it was possible to predict a reduced penetration depth of a non-isothermal inlet air jet.

Based on the results obtained in this project and results presented in the literature it is concluded that numerical prediction of air flow will be a valuable tool for research and design of ventilation systems for livestock buildings. A very interesting feature is the possibility to make detailed calculations of environmental conditions in the occupied zone, such as velocity and temperature distribution and contaminant concentrations.

Finally, it should be mentioned that one three-dimensional simulation of non-isothermal air flow may take several hours of computing time on a normal workstation, but this problem is expected to be reduced in the future with more efficient computer codes and faster computers.

REFERENCES

Christensen, K.S.:

Numerical Prediction of Airflow in a Room with Ceiling-Mounted Obstacles. Proceedings, Roomvent'92, Third International Conference on Air Distribution in Rooms, Aalborg 1992.

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.

Choi, H.L., L.D. Albright, M.B. Timmons and Z. Warhaft:

An Application of the K-Epsilon Turbulence Model to Predict Air Distribution in a Slot-Ventilated Enclosure. Transactions of the ASAE, Vol. 31 (6), November-December 1988.

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.

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.

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.

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.

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.

Nielsen, P.V.:

Flow in Air Conditioned Rooms. Ph.D. dissertation, Danish Techincal University, 1974.

Nielsen, P.V.:

Air Diffusion in Rooms with Ceiling-mounted Obstacles and Two-Dimensional Isothermal Flow. 16th international congress of refrigeration, Paris 1983.

Nielsen, P.V.:

Specification of a Two-Dimensional Test Case. International Energy Agency, Annex 20, 1990.

Patankar, Suhas V.:

Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corporation 1980.

Rodi, Wolfgang:

Turbulence Models and their Application in Hydraulics - a State of the Art Review. IAHR 1984.

Schlichting, Hermann:

Boundary-Layer Theory. McGraw-Hill 1979.