Numerical Prediction of Air Distribution in Rooms with Ventilain of the Mixing Type Using the Standard K, Model

Skovgaard, M.; Nielsen, Peter V.

Publication date:
1990

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

Link to publication from Aalborg University

Citation for published version (APA):
Skovgaard, M., & Nielsen, P. V. (1990). Numerical Prediction of Air Distribution in Rooms with Ventilain of the Mixing Type Using the Standard K, Model. Institut for Bygningsteknik, Aalborg Universitet. Indoor Environmental Technology Vol. R9042 No. 13

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

INDOOR ENVIRONMENTAL TECHNOLOGY

PAPER NO. 13

Presented at "Nordisk Indeklima - arbejdsmiljø og energikonference", Copenhagen, August 1990

M. SKOVGAARD & P. V. NIELSEN
NUMERICAL PREDICTION OF AIR DISTRIBUTION IN ROOMS WITH VENTILATION OF THE MIXING TYPE USING THE STANDARD K, ε MODEL

DECEMBER 1990 ISSN 0902-7513 R9042

INSTITUTTET FOR BYGNINGSTEKNIK

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

INDOOR ENVIRONMENTAL TECHNOLOGY

PAPER NO. 13

Presented at "Nordisk Indeklima - arbejdsmiljø og energikonference", Copenhagen, August 1990

M. SKOVGAARD & P. V. NIELSEN

NUMERICAL PREDICTION OF AIR DISTRIBUTION IN ROOMS WITH VENTILATION OF THE MIXING TYPE USING THE STANDARD K, ε MODEL

DECEMBER 1990 **ISSN 0902-7513 R9042**

NUMERICAL PREDICTION OF AIR DISTRIBUTION IN ROOMS WITH VENTILATION OF THE MIXING TYPE USING THE STANDARD K,ϵ MODEL

Contribution to:

Nordisk indeklima - arbejdsmiljø og energikonference
d. 27.-28. aug. 1990

By

M. Skovgaard and P.V. Nielsen
The University of Aalborg

SUMMARY

In the summer 1988 13 countries under the International Energy Agency established a cooperative work - Annex 20. This Annex is divided into two subgroups where Denmark contributes in subtask 1.

This contribution is giving a historical background for the establishment and the contents of Annex 20. The purpose of Annex 20 is to carry out simulations and full-scale experiments and to compare these investigations. This is done to support a more common use of advanced numerical calculations in engineering flow calculations within the field of ventilation.

In the project it is possible for the countries to make investigations which are of particular interest for them in order to increase the understanding of the complicated flow patterns in ventilated buildings.

One of these topics which is of interest to the University of Aalborg is the low Reynolds number effects. Another area of interest is the boundary conditions around more complicated inlet devices. These two topics are commented in relation to some simulations and experiments.

SUMMARY IN DANISH

I sommeren 1988 indledte 13 lande et samarbejde i det Internationale Energi Agenturs regi. Dette samarbejde - Annex 20 er opdelt i to undergrupper, hvor Danmark deltager i gruppe 1.

Indlægget her giver en historisk baggrund for oprettelsen og indholdet af IEA-Annex 20 projektet, hvis formål er at udføre sammenlignende simuleringer, sammenlignende fuldskalaforsøg samt sammenligne de numeriske beregninger med fuldskala test.

Samtidig med disse test er det dog muligt for de enkelte lande at dyrke egne interesseområder for dermed at hæve forståelsen for de komplicerede strømningsmønstre, der hersker i ventilerede lokaler.

Aalborg Universitetscenter er interesseret i lavturbulente effekter samt randværdier for mere komplicerede armaturer. Disse to områder er kommenteret i forhold til målinger og indledende simuleringer.

BACKGROUND

Up through the 1970's the computational fluid dynamics (CFD) started to be applied to problems related to ventilation techniques. These early two-dimensional predictions showed in comparison with scale model experiments that it was possible to produce pictures of the flow field in large computational domains with small openings and relatively low velocities (e.g. *P.V. Nielsen et. al 1978*). However, these two-dimensional tests were not directly applicable in engineering ventilation technique because the attempt to calculate a complicated three-dimensional case with a 2D approach is a too big restriction due to the three-dimensional inlet conditions and often a strongly three-dimensional flow in the room.

The two-dimensional approach contributed in a very important way to the modelling of inlet conditions - the box method (*P.V. Nielsen 1976*) - which combines the numerical prediction of the velocity field with prescribed empirical velocity near the inlet. This method saves computational time because it economizes the use of finite volumes near the inlet.

Since the first two-dimensional simulations there have been an extensive mathematical and engineering research in the field of CFD in order to develop three-dimensional programmes, better models and to improve the accuracy of the programmes (e.g. *Benocci, C. and Skovgaard, M. 1989* or *Gosman, A.D. et. al. 1980*). This research has also led to a commercialization of CFD with several programmes available on the marked.

Also in the field of ventilation technique - aspiration for a better indoor climate and a better energy management - there is an extensive research in CFD as a design and analysing tool. This is done because the numerical simulation immediately has two benefits. It is able to give very detailed information even in complicated geometries and it is often cheaper to make "numerical experiments" compared to full-scale tests.

The above mentioned reasons are the background for establishment of Annex 20 under the International Energy Agency (IEA).

IEA - ANNEX 20

The IEA - Annex 20 with the title "Air Flow Pattern within Buildings" was established to see how well CFD works in prediction of room air flow patterns. It is also the intention to develop the codes enabling the models to predict some of the problems especially attached to ventilation problems.

The purpose of Annex 20 subtask 1 is:

- to evaluate the performance of air flow models in predicting air velocity, temp. and contaminant distributions,
- to evaluate the applicability of models as design tools,
- to produce guidelines for their use.

The participating countries are :

Belgium, Canada, Finland, the Netherlands, Sweden, Switzerland, UK, USA, Germany, France, Denmark, (Norway and Italy).

The approach is to make identical full-scale experiments in identical test rooms with identical inlet devices at different sites. The measured data will be compared and a database established.

Simultaneously numerical simulations for the measured configurations will be carried out. The accuracy of the predictions will be assessed by an independent council.

The flow considered is:

- jet driven , steady, three-dimensional flow (forced convection, low Archimedes' number).
- buoyancy driven, steady, three-dimensional flow (free convection).

The primary interest of the University of Aalborg is isothermal steady forced convection flow because of the specific interest in the flow around the inlet and the low Reynolds phenomena. These topics are of great importance as we shall see in the next paragraph when flow pattern has to be predicted at low air velocities. Unfortunately the most used models are not able to predict these phenomena which we shall see later.

NUMERICAL PREDICTION OF FLOW PATTERNS IN A ROOM WITH VENTILATION OF THE MIXING TYPE

The numerical predictions are obtained by a numerical solution of the time averaged Navier - Stokes equations and the continuity equation extended with a turbulence model which normally is the k-ϵ model. These equations can be written in the following general form (in Cartesian coordinates):

$$\frac{\partial}{\partial x_j}(\rho u_j \phi) = \frac{\partial}{\partial x_j}\left(\Gamma_\phi \frac{\partial \phi}{\partial x_j}\right) + S_\phi \qquad j \in [1..3] \tag{1}$$

$$\frac{\partial}{\partial x_j}(\rho u_j) = 0 \tag{2}$$

For definition of Γ_ϕ, S_ϕ see *Patankar S.V, 1980.* ϕ may be any of the three velocity components u,v or w as well k and ϵ.

The solution domain is separated into finite volumes and the equations can be discretized to a set of linear homogeneous equations. The solution of the unlinear set of equations is done by a pressure relaxation method and the equations are solved by a TDMA - ADI solution technique.

Computational details

The computational domain is divided into a non-uniform mesh 15*20*11 giving 3300 grid points (fig.1). Staggered grid for the dependent variables u,v and w and hybrid differentiation scheme are used.

The boundary conditions are a prescribed velocity plug profile determined by the required air change rate at inlet and by overall mass flux balance at outlet. The no-slip condition at all the walls is indirectly introduced by wall functions which either can be logaritmic or linear (Couette type) depending on the local y^+ at first grid node. The flow is assumed symmetrical and all the gradients were forced zero on the symmetry plane.

Approx. 2000 iterations of 3 s/it. are required to obtain a fully converged result on a Sony work station with a RISC 3000 processor.

Results and discussion

The room dimensions are identical to the room chosen for full-scale experiments in Annex 20. The location of the inlet and outlet can be seen in fig. 1.

Two different test cases are simulated at different air change rates (fig. 2).

One of the major forces of CFD is that it gives a very detailed knowledge of the qualitative velocity field in a room. From fig. 3 the three-dimensional nature of the flow is evident and it is very interesting to observe that the maximun velocity in the "return flow" is not located in the centre but instead very close to the side walls. It is also interesting to see that the location of the outlet creates a stagnation point near the floor in the occupied zone. The Coanda effect is also observed.

To evaluate the result more specific it is chosen to see how the simulation in the present form (standard turbulence model) performs versus some of the well-known and extensively used simplified relations for wall jets and maximum velocities in the occupied zone (*SBI 128, 1981*):

$$\frac{u_x}{u_0} = K_a \frac{\sqrt{a_0}}{x} \tag{3}$$

$$u_{rm} = c_1 n \tag{4}$$

$$u_{rm} = c_2 \sqrt{I_0} \tag{5}$$

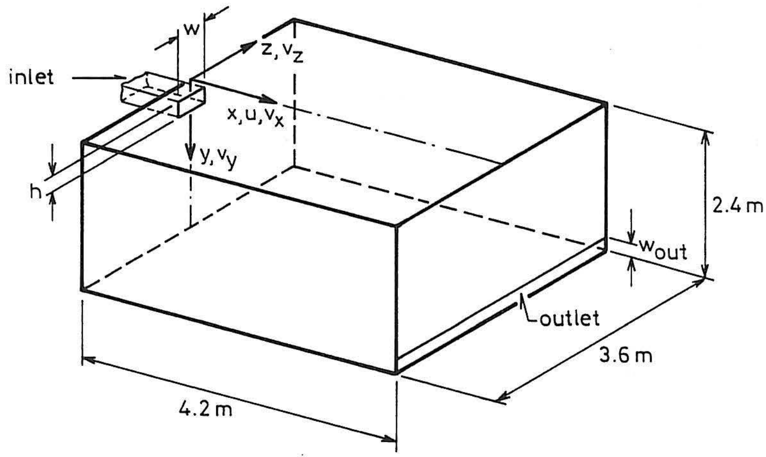
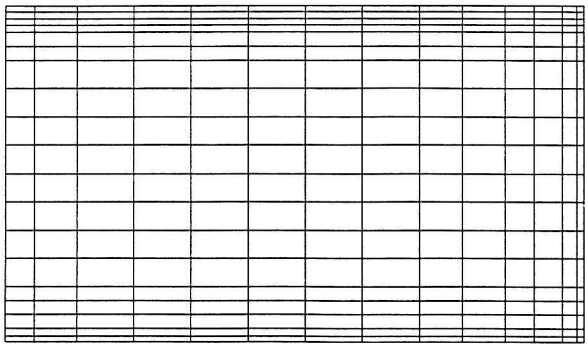


Figure 1. The geometry of the test case.

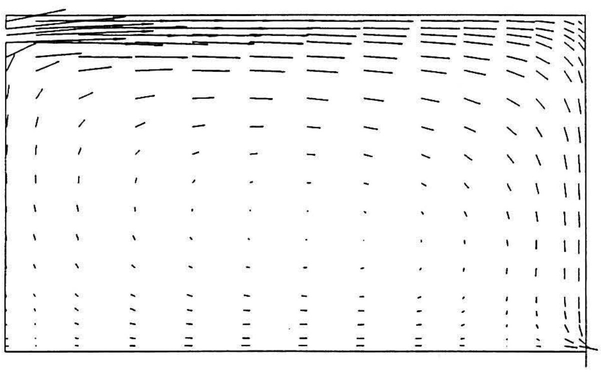
$x_{tot}{}^*y_{tot}{}^*z_{tot}$	w	h	q_0	u_0	
4.2*2.4*3.6	0.3	0.35	0.011	0.104	Case 1.
-"-	0.3	0.35	0.035	0.332	
-"-	0.3	0.35	0.061	0.579	
-"-	0.3	0.35	0.11	1.060	
-"-	0.3	0.23	0.039	0.564	Case 2.
-"-	0.3	0.23	0.047	0.693	
-"-	0.3	0.23	0.092	1.340	

Figure 2. Initial data for the two test cases.

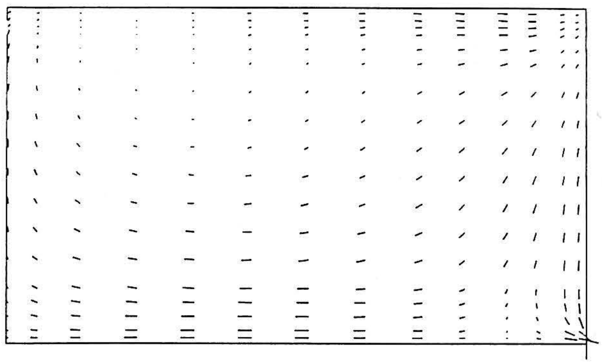
Mesh in xy - plane.



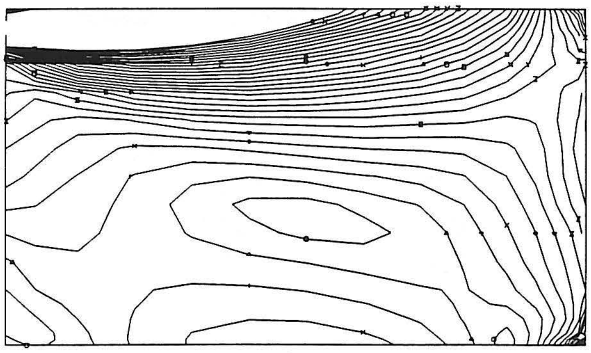
Velocity vectors in the symmetry plane.



Velocity vectors near the side wall in the u_{rm} velocity plane.



Velocity levels in the centre plane.



Velocity levels in the u_{rm} plane.

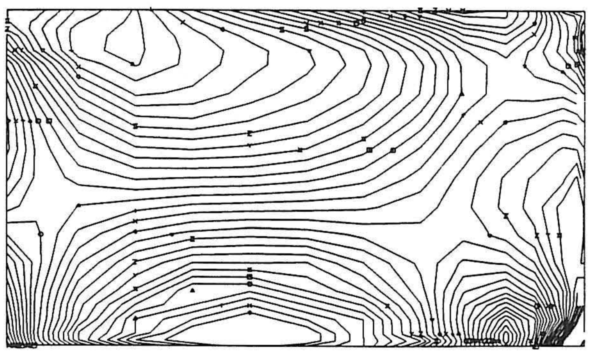


Figure 3. The non-uniform mesh distribution in the xy plane and the velocity distribution in the centre plane and in the plane (z=1.7) in which we find the maximun velocity in the occupied zone.

$$u_{rm} = u_0 f_1 k_a \frac{\sqrt{a_0}}{L}, \text{ where } f_1 = \frac{u_{rm}}{u_L} \quad (6)$$

These comparisons show how the calculations perform both in relation to engineering practice and in relation to recent measurements (*M. Skovgaard et. al. 1990*).

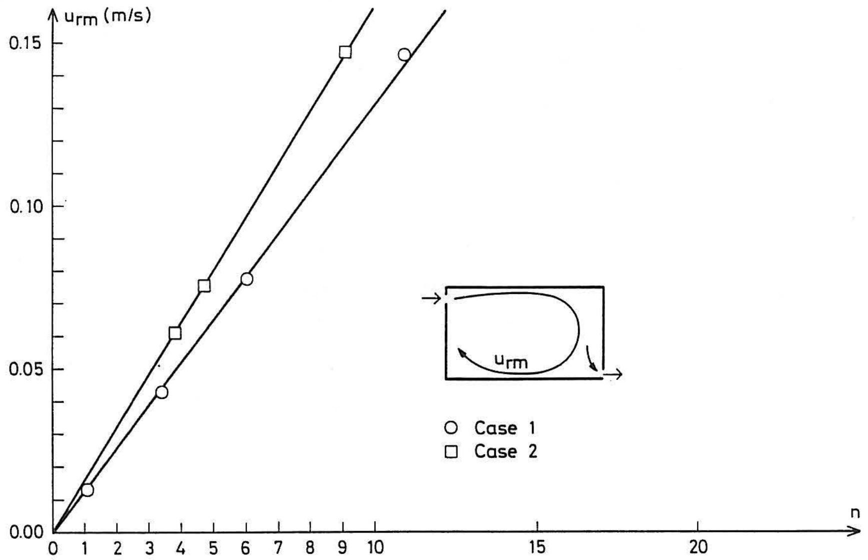


Figure 4. The maximum velocity in the occupied zone as a function of the air change rate for case 1 and 2.

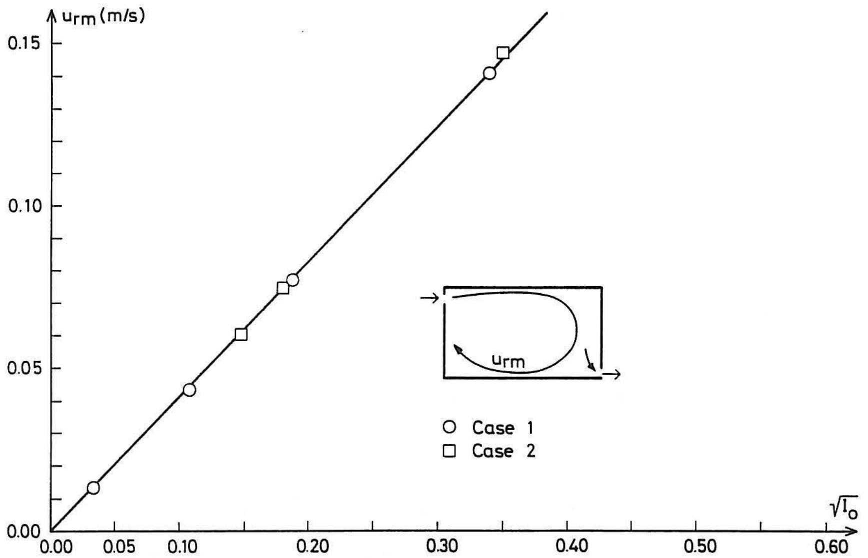


Figure 5. The maximun velocity in the occupied zone versus the momentum flow in the inlet.

Equation (4) is a well-known relation for fully turbulent flow conditions. In fig 4. simulated data for u_{rm} are plottet against the air change rate. It is seen that u_{rm} is proportional to the air change rate. The slope of the curve which corresponds to c_1 is different for the two cases which means that the coefficient c_1 is at least a function of the inlet geometry.

If we instead investigate equation (5) we can from fig. 5 see that c_2 is not a function of the inlet geometry. This result is very interesting because it means that we are able to predict the magnitude of the maximum velocity (not the location of) u_{rm} just by information of the momentum flow through the inlet. So, if we know the connection between the inflow momentum and u_{rm} for one single case (e.g through full-scale or numerical experiments) we will be able to give a qualified guess at the maximum velocity in the occupied zone even for another inlet device of the same type (same K_a-factor).

The two coefficients c_1 and c_2 will of course be functions of the room geometry and inlet velocity (see recent studies by *Skovgaard, M. et. al. 1990*). The supply velocity is not in the fully turbulent domain in many modern inlet devices which means that the momentum flow is different from what was expected, assuming fully turbulent inflow conditions. This low Reynolds number biased flow cannot be taken into account at the moment but at the University of Aalborg an ongoing research project is performed based on the prescribed velocity method to incorporate the low Reynolds number effects from the inlet into the numerical predictions.

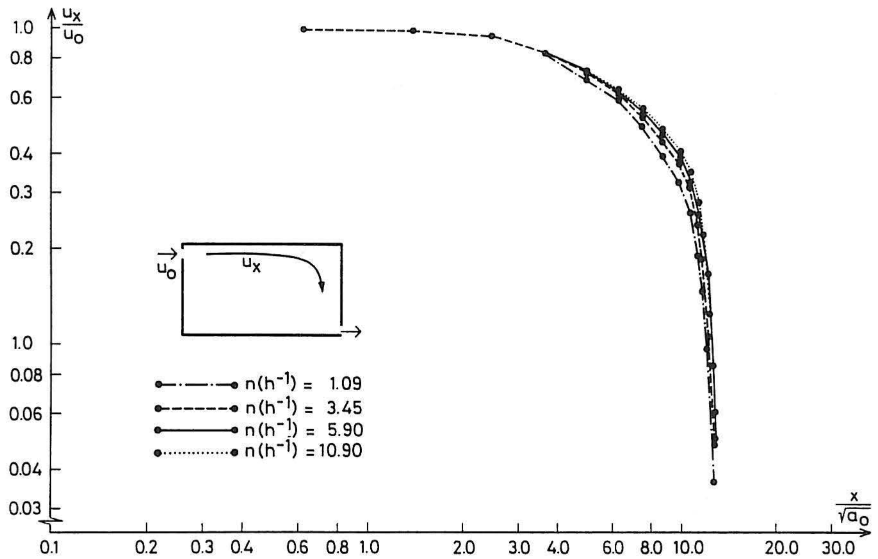


Figure 6. The decay of the center line velocity in the wall jet for case 1.

Looking at the velocity in the jet region below the ceiling (fig. 6-7), we can see that there are a zone with small decay, a zone with velocity decay corresponding to a three-dimensional wall jet (3) and a region where the decay is under influence of the end wall. The first zone which is the core of the jet should have been without any decay of velocity at all. In this region the decay is due to numerical diffusion which in pratice is working like additional (not physical) shear stresses in the fluid. In order to avoid this numerical diffusion one must have a large number of cells in the region near the inlet which increase the calculation time disproportionately much from an engineering point of view. A general form of a prescribed velocity method would be able to overcome this drawback.

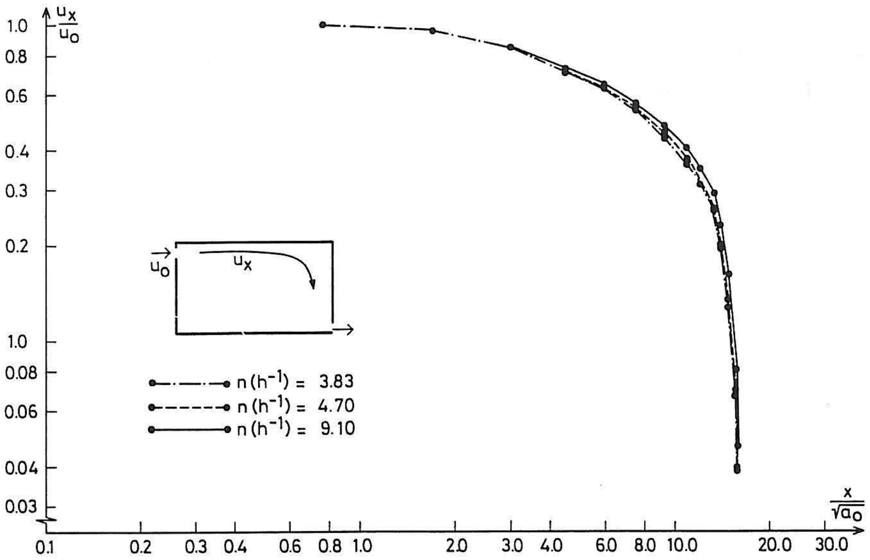


Figure 7. The decay of the center line velocity in the wall jet for case 2.

Equation (6) is a relation between the velocity at the end wall according to (3) and the maximum velocity in the occupied zone taking the inlet conditions into account. Therefore, it is very interesting to study the function f_1. f_1 is a function of the flow dimensions (two - or three-dimensional) (*SBI 128 1981*). This function gives a more universal description of the maximum velocity in the occupied zone than the momentum method (5) because it is not necessary to know other relations than f_1. The use of (5) is limited to the same room geometry and the same structure of the flow.

u_0	u_{rm}	a_0	L	k_a	f_1	
0.104	0.013	0.104	4.2	3.7	**0.44**	Case 1.
0.332	0.043	0.104	4.2	3.9	**0.43**	
0.579	0.077	0.104	4.2	4.0	**0.43**	
1.06	0.146	0.104	4.2	4.1	**0.44**	
0.564	0.037	0.068	4.2	4.1	**0.42**	Case 2.
0.693	0.046	0.068	4.2	4.2	**0.42**	
1.34	0.092	0.068	4.2	4.4	**0.41**	

Figure 8. Calculation of the function in relation (6) for case 1 and 2. In calculation of the $K_a$value the origin of the wall jet is assumed to be located at the inlet (fig. 6-7).

In fig. 8 and 9 the values of f_1 are calculated and as expected the values are almost constant.

The two test cases are very similar and it can be a little difficult to throw the benefit of (6) into relief but, nevertheless, it is certainly a matter which has to be examined more detailed.

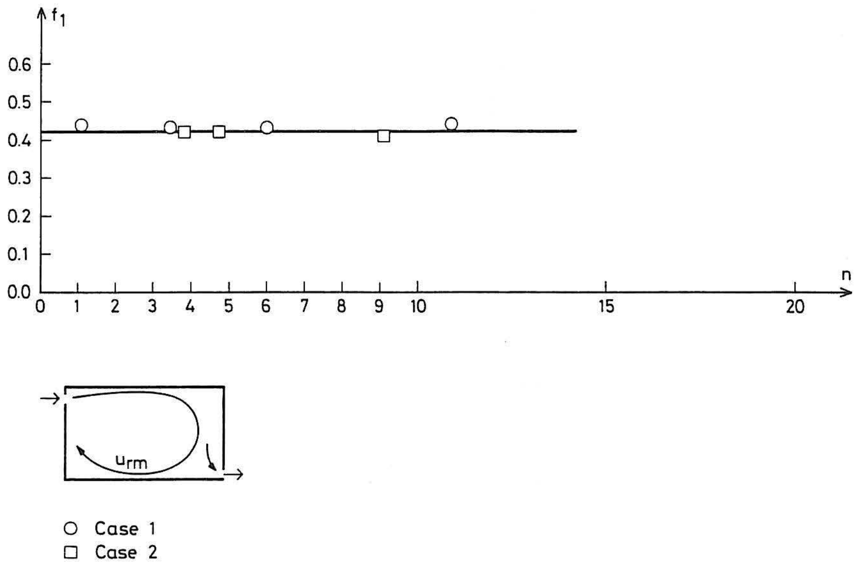


Figure 9. The function f_1 for case 1 and 2.

CONCLUSION

This paper has dealt with evaluation of CFD applied to some aspect in the field of mixing ventilation. The used code is developed at the University of Aalborg by the authors based on the time averaged momentum equations, the continuity equation and the standard k,ϵ model. This code will be referred to as the standard CFD code.

- The performance of the standard CFD code which presumes fully turbulent flow conditions gives additional knowledge (e.g. gives the location of the maximum velocity) and supports the prediction by the simplified models.

- The standard CFD code is at present a very good additional analysing tool to obtain detailed information of flow phenomena in ventilated rooms of different geometries. Nevertheless, this requires some effort and some computational time.

- If low Reynolds' number effects are present, (which is often the case) such as Reynolds' number dependent effective inlet area or low Reynolds' number effects in the room, special actions have to be taken. These special actions could be special implementations or another model than the standard k,ϵ model for description of the turbulence.

- When the simplified models are used (such as the momentum method (5) or the direct method (6)) to design a ventilation system, CFD could be used at an early stage to obtain knowledge of the relations or to make a schedule for a full-scale investigation.

LIST OF SYMBOLS

a	: area
c	: coefficient
f_1	: function to characterize the relation u_{rm}/u_L
h	: height
I	: momentum flow
K_a	: parameter to characterize the wall jet
S	: source term
u	: velocity
w	: width
x,y,z	: Cartesian coordinate
y^+	: Dimensionless wall distance in the boundary layer
q	: volume flux

Greek.

Γ	: fluid property
ρ	: density
ϕ	: variabel

Indices.

j	: direction
L	: length of the room
rm	: maximum in the occupied zone
tot	: total
0	: inlet
1,2..	: arbitrary number
ϕ	: refers to variable ϕ

TABLE OF LITERATURE.

Benocci, C. and Skovgaard, M. "Prediction of Turbulent Flow Over a Backward Facing Step". Proc. of 6th Int. Conf. of Laminar and Turbulent Flows. Swansea, UK 1989.

Gosman, A.D.; Nielsen, P.V.; Restivo, A. and Whitelaw, J.H. "The Flow Properties of Rooms with Small Ventilation Openings. Transactions of the ASME, vol. 102 1980

Nielsen, P.V. "Flow in Air Conditioned Rooms - Model Experiments and Numerical Solution of the Flow Equations." Revised edition of Ph.D. thesis. 1976.

Nielsen, P.V.; Restivo, A. and Whitelaw, J.H. "The Velocity Characteristics of Ventilated Rooms." ASME Journal of Fluids Engineering Vol. 100, 1978 p. 291.

Patankar, S.V. "Numerical Heat Transfer and Fluid Flow." Hemisphere USA. ISBN 0-89116-522-3, 1980.

SBI - Rapport 128. "Luftstrømning i ventilerede arbejdslokaler." Statens Byggeforskningsinstitut. ISBN 87-563-0394-7. 1981

Skovgaard, M.; Hyldgaard, C.E.; Nielsen, P.V. "High and Low Reynolds Number Measurements in a Room with an Impinging Isothermal Jet." ROOMVENT '90, Oslo, 1990.

PAPERS ON INDOOR ENVIRONMENTAL TECHNOLOGY

PAPER NO. 1: C. E. Hyldgård: *Aerodynamic Control of Exhaust.* ISSN 0902-7513 R8712.

PAPER NO. 2: Per Heiselberg, Peter V. Nielsen: *The Contaminant Distribution in a Ventilated Room with Different Air Terminal Devices.* ISSN 0902-7513 R8713.

PAPER NO. 3: Peter V. Nielsen, L. Evensen, Peter Grabau, J. H. Thulesen-Dahl: *Air Distribution in Rooms with Ceiling-Mounted Obstacles and Three-Dimensional Flow.* ISSN 0902-7513 R8714.

PAPER NO. 4: Peter V. Nielsen, Åke T. A. Mõller: *Measurements on Buoyant Wall Jet Flows in Air-Conditioned Rooms.* ISSN 0902-7513 R8715.

PAPER NO. 5: Peter V. Nielsen: *Numerical Prediction of Air Distribution in Rooms. Status and Potentials.* ISSN 0902-7513 R8823.

PAPER NO. 6: Peter V. Nielsen, Åke T. Mõller: *Measurements on Buoyant Jet Flows from a Ceiling-Mounted Slot Diffuser.* ISSN 0902-7513 R8832.

PAPER NO. 7: Peter Kofoed, Peter V. Nielsen: *Thermal Plumes in Ventilated Rooms - An Experimental Research Work.* ISSN 0902-7513 R8833.

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.

PAPERS ON INDOOR ENVIRONMENTAL TECHNOLOGY

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.

Department of Building Technology and Structural Engineering
The University of Aalborg, Sohngaardsholmsvej 57. DK 9000 Aalborg
Telephone: 45 98 14 23 33 Telefax: 45 98 14 82 43