AIR MOVEMENT & VENTILATION CONTROL WITHIN BUILDINGS

12th AIVC Conference, Ottawa, Canada 24-27 September, 1991

.

POSTER 44

Modelling Complex Inlet Geometries in CFD - Applied to Air Flow in Ventilated Rooms.

M. Skovgaard, P.V. Nielsen

University of Aalborg Denmark

MODELLING COMPLEX INLET GEOMETRIES IN CFD -APPLIED TO AIR FLOW IN VENTILATED ROOMS

By

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

SUMMARY

Modern inlet devices applied in the field of ventilation of rooms are getting more complex in terms of geometry in order to fulfil the demand for thermal comfort of the occupants in the room and in order to decrease the energy consumption. This expresses the need for more precise calculation of the flow field. In order to apply CFD for this purpose it is essential to be able to model the inlet conditions precisely and effectively, in a way which is comprehensible to the manufacturer of inlet devices and in a way which can be coped by the computer resources.

In this paper a universal method is presented and tested. The method is based upon three dimensional - and radial wall jet theory and upon diffuser specific experimental data.

Simulations are held up against a more basic method and full scale measurements. The inlet model is evaluated in terms of result, computational effort and applicability. Promising results are obtained.

		Subscript	2
Α	Area	-	
a	Coefficient in difference equation, area	D	Downstream point
C ₁	Constant in the turbulence model	E	East point
C ₂	Constant in the turbulence model	i,j,k	Indicators of direction
Cn	Constant in the turbulence model	N	North point
C,,	Constant in the turbulence model	0	Inlet
E "	Wall roughness function in the	P	Centre point
	logarithmic law	ŕ	Radial
F	Force	rm	Recirculation maximum
I	Turbulence intensity	S	Shear, south point
k	Turbulent kinetic energy	t	Turbulent
K	Factor in wall jet formula	U	Upstream point
n	Air change rate, normal to surface	W	West point
Р	Pressure		-
PD	Percentage Dissatisfied people due to draught	Greek	
Re	Reynolds number $(\rho U/\mu)$		
S	Source term	ð	Kronecker delta, wall jet width, area
Tu	Turbulence intensity	ε	Energy dissipation
u,v,w	Velocity fluctuations	θ	Angle
U,V,W	Mean velocities	ĸ	Von Karman constant
U ⁺	Dimensionless velocity parallel to the	μ	Viscosity (dynamic)
	surface (U_n/Ur)	ρ	Density
x,y,z	Directions	τ	Shear stress
y	Normal distance from wall	σ	Constant in the turbulence model (the turbulent
y+	Dimensionless wall distance $(U_r y_n \rho / \mu)$		Prandtl number)
-	<i>c</i> , <i>p</i> , <i>r</i> , <i>p</i> , <i>r</i> , <i>r</i> , <i>r</i> , <i>p</i> , <i>r</i> , <i>r</i> , <i>r</i> , <i>p</i> , <i>r</i>	φ	Generalized variable

LIST OF SYMBOLS

INTRODUCTION

Numerical prediction of air flow patterns in mechanically ventilated rooms has been a research object for almost two decades. Up through 1970 and -80 Computational Fluid Dynamics (CFD) showed that it was possible to predict the flow field in large domains with relatively small openings (see e.g. *Nielsen 1976*, *Nielsen et. al. 1978* or *Gosman et. al. 1980*).

In recent years the field of ventilation engineering has started to look upon CFD as a design and analysing tool, because CFD offers a radical change in available analytical tools, by which the engineer can predict the impact of a certain design of an air condition system on the indoor climate and the energy management of real buildings. But in contrast to most test carried out with CFD, real buildings and air condition systems are often very complex in terms of building - and air supply device geometry (fig. 1), which gives arise to very complex flow phenomena such as transitional and non-developed flow regimes etc.

The aim of the work reported in the present paper is to look into the influences of a modern complex air supply terminal used in the mixing type of ventilation. The ideas about the model, however, may just as well be applied in the field of displacement ventilation.



Figure 1. Different designs of air inlet devices.

It is well known that the velocity level in a room ventilated by a mixing type of ventilation is strongly influenced by the supply conditions (*Nielsen 1976*) and that it is

the momentum flow of the inlet which has the major impact of the flow pattern in the room (except maybe for very low inlet velocities). It is also known that the momentum flow in a wall jet created by the jet from the inlet diffuser is lower than the inlet momentum flow (*Mc Ree et. al. 1967*). It is therefore very important that the model of the inlet device is able to produce that wall jet momentum either directly by a model of the inlet device or by use of empirical data. In *Nielsen 1976* a method is presented which fulfils these basic requirements by use of diffuser specific data. This fact makes also the method quite universal. The model presented in the following utilizes the same basic ideas.



Figure 2. Typical flow from a wall mounted diffuser.

Fig. 2 shows a typical mounting of a inlet device in a wall. The flow from the inlet creates a wall jet type of flow in a certain distance from the inlet, either because the inlet flow is directed upwards or because it is influenced by the Coanda effect. An attempt to model the inlet directly would in most cases require too much computational effort to be realistic. Another more comprehensible way would be to prescribe the actual wall jet conditions in a volume where the flow has adopted this character. Such a method implies that device specific data are available.

MATHEMATICAL MODEL

The mathematical model of the flow is described by the following equations.

The continuity or mass conservation equation

$$\frac{\partial}{\partial x_i} (\rho U_i) = 0$$

(1)

The momentum equations

$$\frac{\partial}{\partial x_i} \left(\rho U_i U_j \right) = - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i} \left(-\rho \overline{u_i u_j} \right) + \frac{\partial}{\partial x_i} \left(\mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right)$$
(2)

where U_i is the time mean velocity of the direction x_i and u_i is the fluctuating velocity in the x_i direction.

To solve the above set of equations it is necessary to represent the fluctuating velocity by a set of turbulence equations. There are several of such models available, but the most suitable model for practical engineering is the $k-\epsilon$ model, which is a 2 equation semi-empirical model for turbulent kinetic energy and its dissipation. The $k-\epsilon$ model takes up the eddy viscosity concept by describing the Reynolds stresses in the following way

$$-\rho \overline{u_i u_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij}$$

By substituting the eddy viscosity concept into (2) we obtain the following equation for the mean flow

$$\partial (\rho U_i U_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i} \left((\mu_t + \mu) \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right) - 2\rho k \delta_{ij}$$
(3)

where
$$\mu_t = C_\mu \rho \frac{k^2}{\epsilon}$$
 (4)

The closure equations for k and ϵ will be transport equations of the following form

$$\frac{\partial(\rho U_{i}k)}{\partial x_{i}} = \frac{\partial}{\partial x_{i}} \left(\frac{\mu_{t}}{\sigma_{k}} \frac{\partial k}{\partial x_{i}} \right) + \mu_{t} \frac{\partial U_{i}}{\partial x_{j}} \left(\frac{\partial U_{i}}{\partial x_{j}} + \frac{\partial U_{j}}{\partial x_{i}} \right) - C_{D} \rho \varepsilon$$
(5)

$$\frac{\partial(\rho U_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{\mu_t}{\sigma_{\varepsilon}} \frac{\partial \varepsilon}{\partial x_i} \right) + C_1 \frac{\varepsilon}{k} \mu_t \frac{\partial U_i}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{\varepsilon}{k} C_2 \rho \varepsilon$$
(6)

where k and ϵ has the definition

$$k = \frac{1}{2} \overline{u_i} \overline{u_j}$$

$$\varepsilon = \frac{\mu}{\rho} \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i}$$

For fully turbulent flow the following set of constants is assigned

$$C_{\mu}$$
=0.09; C_1 =1.44; C_2 =1.92; σ_k =1.0; σ_e =1.3; C_D =1.0

NUMERICAL PROCEDURE

The computational domain is divided into a number of cells by a non-uniform, staggered and rectangular mesh in order to produce finer grid close to the walls and other areas where gradients may be expected to be large. The previous set of equations (3 + 5 + 6) are dicretised by the finite volume technique (FV) (*Patankar* 1980) and cast into following general form:

$$a_{\mathrm{P}}\phi_{\mathrm{P}} = a_{\mathrm{E}}\phi_{\mathrm{E}} + a_{\mathrm{W}}\phi_{\mathrm{W}} + a_{\mathrm{N}}\phi_{\mathrm{N}} + a_{\mathrm{S}}\phi_{\mathrm{S}} + a_{\mathrm{D}}\phi_{\mathrm{D}} + a_{\mathrm{U}}\phi_{\mathrm{U}} + S_{\phi}$$

$$S_{\phi} = S_{\mathrm{C}} + S_{\mathrm{P}}\phi_{\mathrm{P}}$$
(7)

The pressure is linked via the pressure correction technique and the turbulent viscosity is calculated by (4).

When a surrounding set of boundary conditions is provided a solution can be obtained. To solve the equations (7) the TMDA technique with an ADI - like procedure in the cross room planes is used.

BOUNDARY CONDITIONS

To be able to solve the discretised set of equations a full set of boundary conditions (BC) must be given because of the elliptic nature of the governing equations. However, the use of the hybrid scheme may have the effect that not all BC's are effective.

Test case

The test room is shown in fig. 3. The inlet device is of the HESCO-type (KS4W205K370) where the flow can be adjusted to any kind of three dimensional flow. For this purpose all nozzles are adjusted to an angle of 40° upwards.



Figure 3. Sketch of the test case. a) The room geometry. b) Close up of the inlet device.

(i) Boundary conditions at surfaces

Because of the validity range of the set of equations (fully turbulent region) the no-slip boundary conditions have to be introduced indirectly by wall functions in the source term at the first gridnode (subscript p). This approach has also the advantage that it spares some gridnodes in the near wall region. However, this method can be difficult to use in areas where the maximum velocity is found close to the surface (e.g. the wall jet region).

The boundary conditions are given by the shear force at the wall (subscript s) and the velocity parallel to the surface in the first gridnode (subscript p)

$$F_{\rm s} = \delta x_{\rm s} \tau_{\rm s} = -\delta x \mu \frac{\partial U}{\partial n}$$

If $y^+ < 11.63$:

$$F_{\rm s} = -\mu \frac{(U_{\rm p} - U_{\rm s})}{y_{\rm p}} \delta x$$
(8)

If $y^+ > 11.63$:

$$F_{\rm s} = -\frac{\rho C_{\rm D} C_{\mu}^{\frac{1}{4}} k_{\rm p}^{\frac{1}{2}} (U_{\rm p} - U_{\rm s})}{U^{*}} \, \delta x \tag{9}$$

$$U^{*} = \frac{1}{\kappa} \ln(Ey^{*})$$
; $k_{\rm P} = \frac{U_{\tau}^{2}}{\sqrt{C_{\mu}}}$; $\kappa = 0.4187$; $E = 9.793$

 F_s is subtracted as a source term in the near wall cell in the point p with ϕ_p equal to

 u_p and $u_s = 0$. The limit 11.63 is probably not valid in three dimensional boundary layer flow and one can argue that it is already too low in the one dimensional boundary layer but it is, nevertheless, used in this two layer approach.

Turbulent kinetic energy and dissipation:

$$\frac{\partial k}{\partial n} = 0$$

$$\varepsilon = \frac{C_{\mu}^{\frac{3}{4}}k^{\frac{2}{3}}}{y_{p}\kappa}$$

(ii) Boundary conditions in the return opening

Outlet U-velocity is set to fulfil the overall continuity so

$$U_{\text{out,uniform}} = \frac{\int_{x_1}^{x_2} \int_{y_1}^{y_2} U_0(x,y) dA}{A_{\text{out}}}$$

All gradients in the outflow plane are zero as well as the pressure. Upwind boundary is assumed so exact values are not required.

(iii) Inlet boundary conditions

The inlet boundary is particularly complex because we must give an exact image of the real conditions and real conditions often mean a very complicated design of the inlet device. The complex inlet device used in present test case is chosen to give a complicated flow pattern around the inlet and therefore is a realistic test of the CFD method. The present inlet device is difficult to model directly because the many small nozzles are distributed, over a fairly large area and are directed upwards in an angle of 40°. It is therefore decided to try two approaches to represent the inlet flow conditions.

The two methods tested in the following are: a basic momentum preserving method and a development - or extension of the method outlined by Gosman et. al. 1980, and further explaned by Nielsen 1989 and Skovgaard et. al. 1990. The latter method is developed to be general - and to be more comprehensible to the manufacturer. The ideas behind the method are not only useful in most set-ups of the mixing type but also in many displacement systems.

The basic method:

The assumptions for this method are that the momentum flow in the jet created around the inlet should be presented by a more simple model. The distributed nozzles are simulated by a single opening having the same effective inlet area, the same aspect ratio (h/w) and the flow rate equal to the measured and therefore the same momentum flow as the actual diffuser. The above mentioned assumptions give following B.C.'s.

$$a_{inlet} = 0.18*0.062 = 0.011 \text{ m}^2$$

$$a_o = f(u_o)$$
for n = 1 : 0.008 m²
for n = 3 : 0.00855 m²
for n = 6 : 0.009 m²

$$U_o = 3.6 \text{ m/s} : U_{inlet} = 2.71 \text{ m/s}, V_{inlet} = 2.27, W_{inlet} = 0.0 \text{ (n = 3)}$$

$$k_{inlet} = 1.5*I^2 U_{inlet}^{-2}, I = 0.1$$

$$\epsilon_{\text{inlet}} = C_{\mu}^{3/4} k^{3/2} / l$$

The "prescribed velocity" PV method:

When the PV method is used the velocity profiles for u and v are prescribed in a full volume in front of the diffuser at a location where the wall jet type of flow is established and therefore has a parabolic nature. This method requires data for the behaviour of the real flow. Test were carried out on the chosen inlet geometry by *Skovgaard et. al. 1990.* The tests showed that the maximum velocity of the wall jet is described by

(10)

$$U_{\rm r} = K(\theta) U_0 \frac{\sqrt{a_0}}{x + x_0}$$

where the K-factor for this particular inlet is interpolated from experimental data (Skovgaard et. al. 1990) (fig. 4).

$$K(\theta) = 4.2 - 0.975\theta - 8.206\theta^2 + 7.828\theta^3 - 2.088\theta^4$$
(11)

 θ is in rad.



Figure 4. $K(\theta)$ interpolated from eksperimental data.

U and V are calculated from

 $U = \cos \theta U_r$ $V = \sin \theta U_r$

U and V profiles are self similar up to a distance of $y/\delta_{y_2} = 1.0$ under the ceiling

(Nielsen 1989). The following wall jet profile is assumed (Verhoff 1963)

$$f\left(\frac{y}{\delta_{\frac{1}{2}}}\right) = 1.4794\left(\frac{y}{\delta_{\frac{1}{2}}}\right)^{\frac{1}{7}}\left(1 - erf\left(0.6775\left(\frac{y}{\delta_{\frac{1}{2}}}\right)\right)\right)$$

and δ is taken from Skovgaard et. al. 1990:

$$\delta_{\frac{1}{2}}(x) = 0.08(x+0.45)$$

RESULTS.

Simulated results from the basic- and the PV model will in the following be compared with measured data for air change rates 1, 3 and 6 h^{-1} .

The overall flow patterns for the cases are shown in fig. 5 and 6.

Predictions in the basic model show that the radial jet below the ceiling, fig. 5g, has a component of very high velocity directed against the corners. This flow pattern may be the result of the impingement at the ceiling and it has the effect that a high velocity level is obtained in the occupied zone.

The velocity distribution below the ceiling in the case of the PV model has a characteristic "peak" along the centre line, fig. 6g. This is, in the experiments, observed as a area with parallel flow. Measurements by *Heikkinen 1991* have also shown this combination af two/three dimensional flow and radial flow. The maximum velocity in the occupied zone is close to the measured level, which shows the practical relevance of the PV model.

The area with the maximum velocity, U_{rm} , in the occupied zone is located very close to the wall opposite the supply opening (*Skovgaard et. al. 1990*). The predictions are not able to reproduce this location.

The results in fig.7 relate to the measured data by *Skovgaard et. al. 1990*. The figure depicts the decay of the centre line velocity. x_0 is measured to 0.45 m behind the inlet. $a_0^{\frac{14}{5}}$ is a function of the inlet Reynolds number and is again taken from the measured data.

It is seen that discrepancies are found in both simulations due to the very complex flow structure in the real case. In the basic model case, the peak velocity decreases too rapidly caused by the flow going outwards towards the corners. If we instead focus on the PV model it is seen that although discrepancies are present the decay is simulated with higher accuracy.

The predictions indicate that a good description of the boundary conditions is a necessary requirement for the prediction of fully turbulent flow pattern with acceptable accuracy.



Figure 5. Air flow patterns from the basic model $(n = 3h^{-1})$. a) Velocity vectors in the centreline. b) Velocity vector in a plane 0.04m below the ceiling. c)-e) Speed contours in the planes z = 0.02, 1.0 and 1.7m. f) Iso - kinetic energy in plane z = 0.02. g)-h) Speed contours in y = 2.36 and y = 0.05.

The width of the jet in the centre plane, which is depicted in fig. 8, has a close connection to the results shown in fig. 5, 6 and 7. It is seen that the spread of the jet simulated by the PV boundary conditions is close to the measured values while the basic case shows some deviation.



Figure 6. Air flow patterns from the PV model $(n = 3h^{-1})$. a) Velocity vectors in the centreline. b) Velocity vector in a plane 0.04m below the ceiling. c)-e) Speed contours in the planes z = 0.02, 1.0 and 1.7m. f) Iso - kinetic energy in plane z = 0.02. g)-h) Speed contours in y = 0.05 and y = 2.36.

One of the main purposes of a design propos is to predict the maximum velocity in the occupied zone. Fig. 9 shows the measured and the predicted maximum velocity U_{rm} . The figure shows that the PV method for description of the supply opening is giving the best estimate of U_{rm} . But it can also be seen that the performance is poor for n = 1 in both models because the low Reynolds number effect has a large impact on the flow at low velocities.



Figure 7. Measured and simulated decay of peak centreline velocity ($n = 3h^{-1}$). o - measured, dashed line - basic model and line - PV model (the PV volume in indicated by squares).



Figure 8. Spread of the wall jet in the centre plane. (o - measured, dashed line - basic model and line - PV model.)

Several authors have made studies of this phenomenon (see e.g. Skovgaard et. al. 1990, Murakami 1983, Chen 1979 or Restivo 1979) but it has not led to any prediction of it because the k,ϵ model supposes a fully turbulent flow. The low Reynolds number effect i fig. 9 arises partly from the supply device and partly from the flow in the room (Skovgaard et. al. 1990). The basic model can incorporate the change in the effective area, a_o and the PV model can in addition to this also include variations in the prescribed volume up to a distance of x=1.35m. The effects are not very obvious in any of the predictions. This is partly because the change in $K(\theta)$ as a function of air change rate isn't taken into account in the PV model (a mean of the measured K factors is used). But it is also because low Reynolds phenomena that are occurring inside the flow and therefore cannot be described by the boundary conditions. Further investigations in this specific area have to be done.

Figure 9. $U_{\rm rm}$ as a function of air change rate. squares - experiments, triangles - basic model, o - PV model.

As previously mentioned one of the major forces with CFD analysis of flow fields is the very detailed knowledge of the velocity, turbulence and thermal parameters in the room. These data can be used in a comfort analysis of the room or the occupied zone. In fig 10 such an application is shown where the PD index (percent dissatisfied) is calculated (*Fanger et. el. 1989*).

The calculation is given as an example and the same equation is therefore used for the full room and for the foot level, although it is known that the draft tolerance is higher in foot level than in the average of the body. The following comfort equation is used

$$PD = (34-t_{air})(U-0.05)^{0.62}(0.37UTu+3.14)$$

where

$$Tu = \frac{\sqrt{k}}{1.1U}$$

Figure 10. Calculated PD index. a) z = 0.02. b) Foot height approx. 5 cm above the floor.

CONCLUSION

CFD simulations of air flow patterns in ventilated spaces give detailed information of comfort parameters and are therefore useful as a analysing and design tool.

A good representation of the boundary conditions is a necessary requirement for prediction of fully turbulent flow. Two inlet models have been tested. One which aims to model the supply device directly (basic) and one which models the resulting flow pattern in a volume in front of the diffuser (PV). The latter method, which requires diffuser specific data, has clearly the best performance. The first method fail to meet one basic requirement namely to reproduce the overall flow pattern. The second mathod gives also he lowest computational cost for the simulated case.

The PV model is still the best choice if low Reynolds number effects are present because it can incorporate low Reynolds number effects from the inlet device and from the resulting flow up to the border of the volume in a certain distance from the inlet.

The simulation indicates that low Reynolds number effects not only arise from the inlet device and the boundary layer but from the flow in the room. This pert of the low Reynolds number effects can therefore not be taken into account by solely modifying the boundary conditions.

LIST OF REFERENCES

Chen, Q., Suter, P. and Moser, A.

Influence of Air diffusion. Energy systems laboratory series. Federal Inst. of Tech, ETH, Zürich Switzerland 1990.

Fanger, P.O., Melikov, A.K., Hanzawa, H. and Ring, J. Turbulence and draft. ASHRAE journal, 1989.

Gosman, A. D., Nielsen, P. V., Restivo, A. and Whitelaw, J. H. The Flow Properties of Rooms with Small Ventilation Openings. ASME Journal of Fluids Eng. Vol 102 1980.

Heikkinen, J. Appendix to IEA, RID no. 1.13: Specifications of Test case B. 1989.

Heikkinen, J. Private communication. 1991.

Murakami, S., Tanaka, T. and Kato, S.

Numerical Simulation of Air Flow and Gas Diffusion in Room Model -Correspondance between Simulation and Model Experiments. University of Tokyo, 1983.

Nallasamy, M. Turbulence models and their application to the prediction of intenal flows, a review. Computers and fluids, Vol. 14 no. 2. 1987.

Nielsen, P.V. Flow in Air Conditioned Rooms - Model Experiments and Numerical Solutions of the Flow Equations. Revised ed. of Ph.D. - Thesis 1976.

Nielsen, P.V, Restivo, A. and Whitelaw, J. H. The velocity Characteristics of Ventilated Rooms. ASME Journal of Fluid Eng. Vol. 100, 1978 p.291.

Nielsen, P.V. Representation of Boundary Conditions at Supply Openings. International Energy Agency, Annex 20, ISSN 0902 - 7513 R8902, 1989.

Mc Ree, D.I. and Moses, H.L. The Effect of Aspect Ratio and Offset on Nozzle Flow and Jet Reattachment. 1967 FLUIDICS symposium ASME.

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

Restivo, A.M.O. Turbulent flow in ventilated rooms. Imp. Coll. of Science and Tech, Mech. Eng. Dept. Ph.D. - Thesis, 1979.

Skovgaard, M., Hyldgård, C.E. and Nielsen, P.V. High and Low Reynolds Number Measurements in a Room with an Impinging Jet. Roomvent '90, Oslo, 1990.

Verhoff, A. Report N626, Princeton University, Dept. of Aeronautional Engineering, May 1963.