CALCULATION OF THE VELOCITY AND CONCENTRATION FIELDS IN A CLEAN ROOM

by

Lars Davidson
Department of Applied Thermodynamics and Fluid Mechanics
Chalmers University of Technology
S-412 96 Gothenburg, SWEDEN

ABSTRACT

The turbulent flow in a two-dimensional isothermally ventilated clean room with vertical airflow has been numerically simulated. A finite volume computer program has been used. The turbulence was modelled by using the standard k-ϵ turbulence model.

The air is supplied through the whole width of the ceiling and it is extracted through the floor. A machine is placed in the room; it is desirable to keep its work table as clean from contaminant as possible. The contaminant is generated by a line source, simulating an operator (not included in the calculations) working at the table.

The flowfield is calculated. The dynamically passive contaminant is then introduced via a source near the work table. It is investigated whether the concentration of contaminant in the working section is reduced if a local exhaust is introduced at the surface of the work table.

The calculations show that it is preferable to have a local exhaust at the work table.

NOMENCLATURE

c	concentration of contaminant
$\langle c_{ex,1} \rangle$	mean exit concentration in Case 1
$C_{1\epsilon}$, $C_{2\epsilon}$, C_μ	constants in turbulence models
G	turbulence generating source term in the k- and ϵ-equations
k	turbulent kinetic energy
p	pressure
Q	total volumetric flow
S_ϕ	source term of general variable
t	time

U, V	mean velocity in x and y-direction, respectively
U_i	mean velocity in x_i - direction
w_{out}	exit velocity below the working zone (Case 2; see Fig. 1)
x	horisontal cartesian co-ordinate (see Fig. 1)
x_i	cartesian co-ordinate in i-direction
y	vertical cartesian co-ordinate (see Fig. 1)

<u>Greek symbols</u>

Δt	time step
Γ_ϕ	exchange coefficient of dependent variable
ϵ	dissipation of turbulent kinetic energy
ϵ_w	local ventilation efficiency for the region near the work table [see Eq. (2)]
μ, μ_{eff}, μ_t	dynamic viscosity (laminar, effective and turbulent, respectively)
Ψ	stream function
ρ	density
σ_c, σ_k, σ_ϵ	turbulent Prandtl number for concentration, turbulent kinetic energy and dissipation of turbulent kinetic energy, respectively
ϕ	dependent variable

<u>Subscripts</u>

in	inlet

1. INTRODUCTION

The demands on air quality are very stringent in many industries (the electric, for example), which means that production has to be carried out in special, clean rooms. It is therefore of great interest to study the air flow pattern and the concentration of contaminant in this type of room. In the present work a numerical study is carried out in order to investigate whether air quality can be improved by the introduction of a local exhaust.

Numerical simulation of air movements in ventilated rooms using finite-difference methods have become rather common. Much work has been carried out where good agreement between calculated results and experiments have been obtained, see e.g., Nielsen (1973), Holmberg <u>et</u>

al. (1975), Nielsen et al. (1978), Hanel and Scholz (1978), Hjerthager (1979), Gosman et al. (1980), Larsson (1977), Davidson (1986) and Davidson and Olsson (1987a, 1987b, 1987c). This work shows that the flow pattern can be well predicted by using numerical simulation.

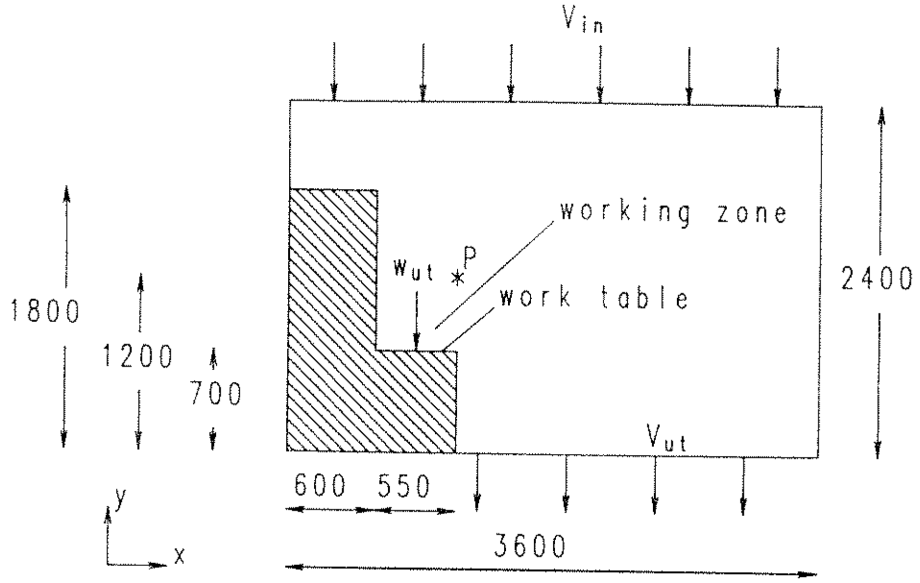


Figure 1. Configuration.

In this study velocity and concentration fields are calculated for a two-dimensional, isothermally ventilated clean room. The configuration is shown in Fig. 1. In the left part of the room is a machine (shaded area) at which an operator (not included) is supposed to work. The region above the lower part of the shaded rectangle is the working zone. Air is supplied through the whole width of the ceiling and is extracted through the whole width (non-shaded) of the floor. The velocity field and the concentration field are calculated for two cases:

Case 1: $w_{out}=0$

Case 2: $w_{out}=V_{in}$

where w_{out} denotes the velocity at which the air is extracted below the working section, see Fig. 1. It is desireable to keep the working zone as free from contaminant as possible. The main object of this study is to investigate whether it is advantageous to use a local exhaust at the work table (Case 2), or not (Case 1).

The velocity field is first calculated for each case. A line source of contaminant (simulating an operator who emits particles) is then introduced at point P (see Fig. 1), and the concentration field is calculated. Since the contaminant is considered to be dynamically passive, the concentration field can be calculated using the calculated velocity field.

2 SOLUTION PROCEDURE

A slightly modified version of the computer program by Davidson and Hedberg (1986) has been used. The program solves equations of the type

$$\frac{\partial}{\partial t}(\rho\phi) + \frac{\partial}{\partial x_i}\left(\rho U_i\phi - \Gamma_\phi \frac{\partial\phi}{\partial x_i} \right) = S_\phi \qquad (1)$$

by expressing them in finite difference form. The finite difference equations are solved by a procedure which is based on the SIMPLER procedure by Patankar (Patankar 1980, 1981). The five main features are staggered grids for the velocities; formulation of the difference equations in implicit, conservative form, using hybrid upwind/central differencing; rewriting of the continuity equation into an equation for pressure correction which is used to correct the velocities; a separate Poisson equation for pressure; and iterative solution of the equations. A transient formulation was adopted as a convenient means of introducing relaxation into the iterative solution. When, as in this study, only the steady-state solution is of interest, the time step Δt is used as a free parameter through which the convergence rate may be optimized.

In the present calculations the dependent variable in Eq. (1) takes the following forms: U, V, c, k, ϵ and 1 (continuity equation). The corresponding coefficients, Γ_ϕ, and sources, S_ϕ, are defined in Table 1.

Table 1. Definition of Γ_ϕ and S_ϕ for conservation equations

Equation	ϕ	Γ_ϕ	S_ϕ
Continuity	1	0	0
Momentum	U	μ_{eff}	$-\partial p/\partial x$
Momentum	V	μ_{eff}	$-\partial p/\partial y$
Concentration	c	μ_{eff}/σ_c	0
Turbulence energy	k	μ_{eff}/σ_k	$G - \rho\epsilon$
Turb. dissipation	ϵ	$\mu_{eff}/\sigma_\epsilon$	$\epsilon/k(C_{1\epsilon}G-C_{2\epsilon}\rho\epsilon)$

Notes:

1. $G = \mu_{eff} \frac{\partial U_i}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) \qquad \mu_{eff} = \mu + \mu_t = \mu + C_\mu \rho k^2/\epsilon$

2. Turbulence constants (see Rodi, 1980)
$C_\mu = 0.09$; $C_{1\epsilon} = 1.44$; $C_{2\epsilon} = 1.92$; $\sigma_k = 1.0$; $\sigma_\epsilon = 1.3$
$\sigma_c = 0.7$

Boundary conditions

The inlet velocity at the ceiling was set (V_{in}=0.45 m/s); the turbulent quantities were estimated, and the inlet concentration was zero. Conventional wall-functions (Rodi, 1980; Davidson, 1986) were used for velocities, k and ϵ at all walls; zero flux (impermeable walls) was applied for the concentration. Zero stream wise gradient was imposed for all variables at the floor outlet. For case 2 the V-velocity was set to w_{out} ($=V_{in}$) at the outlet below the working zone.

3. RESULTS

The velocity fields are presented in Figs. 2 and 3. The vector plots (Fig. 2) show that the recirculation is greater in the working zone for Case 1 than for Case 2. The reason for this is that most of the fluid approaching the work table in Case 1 must turn 180^o and continue upwards along the vertical wall of the machine, thus forcing the downward directed fluid further away from this wall. In Case 2 a fraction of the flow approaching the work table is exhausted through the outlet at the work table; the amount of fluid forced upwards along the vertical wall is thus smaller in this case and the downwards directed fluid is consequently not forced as far from the wall as in Case 1.

The contours of the stream function are shown in Fig. 3. The stream function, Ψ, is defined so that it is zero at the left vertical wall above the machine, and at the contour of the machine; $\Psi=1$ at the right vertical wall. Between the contours $\Psi=0.1$ and 0.2, for example, 10% of the total incoming flow, Q, is moving in the direction of the tangent of the contours. A negative value of Ψ means that the flow enclosed by this contour is (mostly) directed upwards. When we study Fig. 3 we see, as indicated in the discussion above, that in the working zone more fluid is moving upwards in Case 1 than in Case 2. The sharp break (discontinuity) in the contours in Fig. 3b (Case 2) at the level of the work table, is due to mass flow being extracted at the work table.

The predicted concentration field in the room, which is the most interesting field in the present study, is presented in Fig. 4 (note that the concentration field has been scaled with the mean exit concentration in Case 1, $\langle c_{ex,1} \rangle$, in both Figs. 4a and 4b in order to enable comparison). It is clearly seen that the concentration of contaminant in the working zone is higher in Case 1 than in Case 2. Contaminant is transported from the source at point P towards the work table. In Case 1 a larger fraction of this amount of contaminant is forced to recirculate, above the work table, than in Case 2; this explains the higher values of concentration for the former case.

The concentration fields above the work table are presented in Fig. 5 in a magnified plot. We see that, although the concentration level is much lower for Case 2 than for Case 1 in the larger part of this region, this is not true near the outer edge of the work table.

In studies by Larsson <u>et al</u>. (1987a,b), who carried out experimental investigations in a configuration similar to that in the present study, a local ventilation efficiency for the near region above the work table was defined as

$$\langle \epsilon_w \rangle = \frac{\langle c_{ex,1} \rangle}{\langle c_w \rangle} \tag{2}$$

Subscript w denotes the region $0.6 < x < 1.15$, $0.7 < y < 0.8$. For Case 1 and Case 2 the values $\langle \epsilon_w \rangle = 0.85$ and 0.18, respectively, were obtained. The local ventilation efficiency is thus <u>higher</u> for Case 1 than for Case 2. The reason for this is that the concentration in Case 2 is very high near the edge of the work table. If the region for ϵ_w is redefined so that $0.6 < x < 1.0$, $0.7 < y < 0.8$, the values $\epsilon_w = 1.18$ and 10.6 are obtained for Case 1 and Case 2, respectively; these figures seem better in agreement with Figs. 4 and 5.

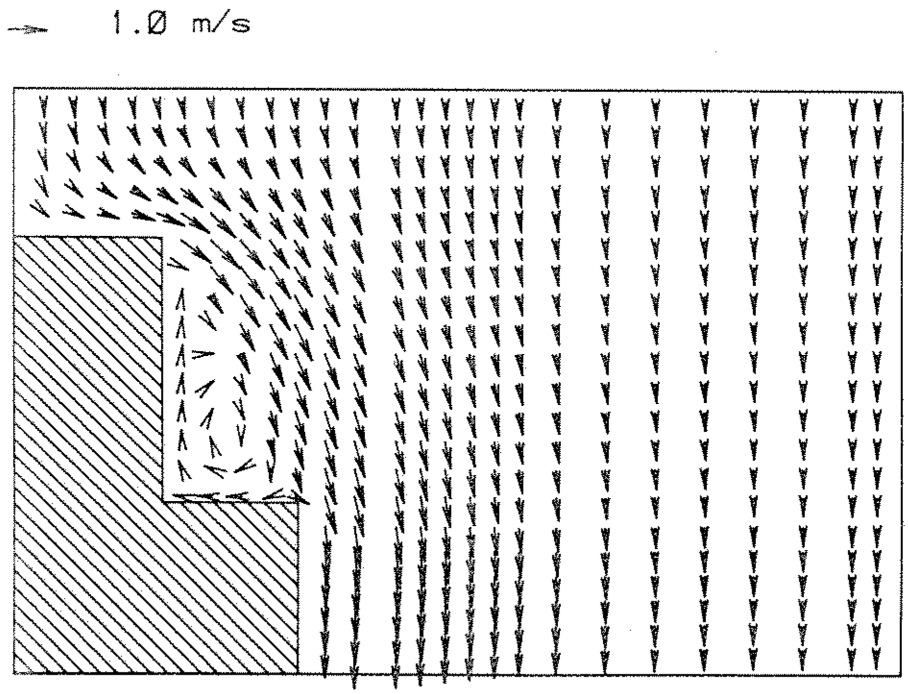
<u>3.1 Computational Details</u>

A grid with 55 x 54 nodes was used. The calculated results are considered to be grid independent. This was checked in two ways:

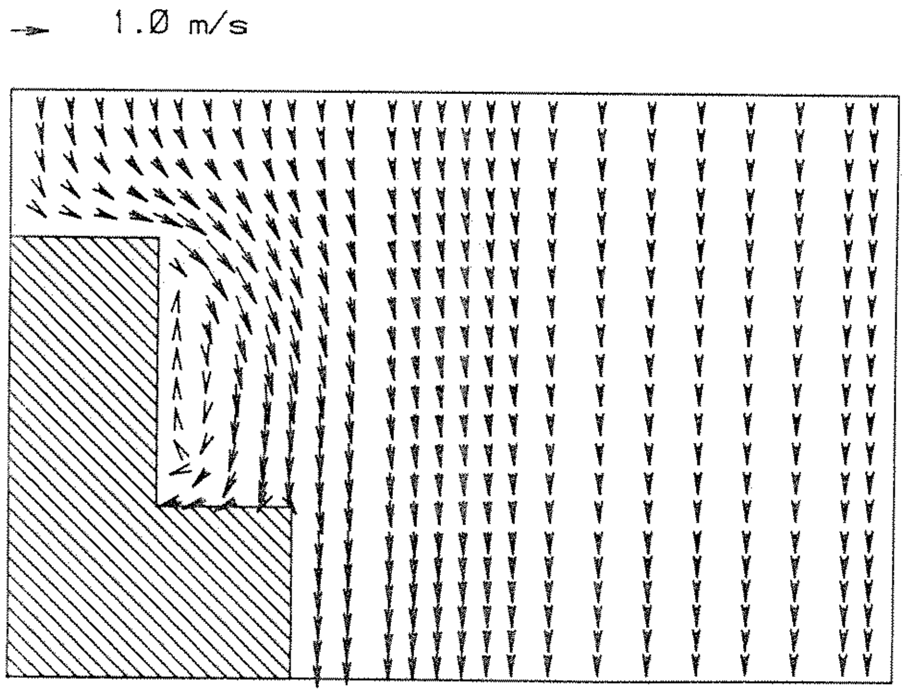
i) the velocity fields were calculated using a 42 x 39-node grid; these results were more or less identical to those obtained with the 55 x 54-node grid.

ii) the velocity fields were also calculated (with the 55 x 54-node grid) using a skew-upwind differencing scheme (Raithby, 1976); the results obtained differed very little from those obtained with the conventional hybrid central/upwind differencing scheme. The skew-upwind scheme reduces numerical diffusion. The scheme was used only in the momentum equations. The k- and ϵ-equations are rather insensitive to the choice of differencing scheme since they are very source dominated (Leschziner, Rodi 1981).

All calculations were performed on a VAX-750 machine. The CPU-time required for calculating the velocity fields was about one and a half hours, and 200 iterations were used; the corresponding figures for the concentration calculations were two minutes and 20 iterations. Time steps Δt of the order of half a second were used.

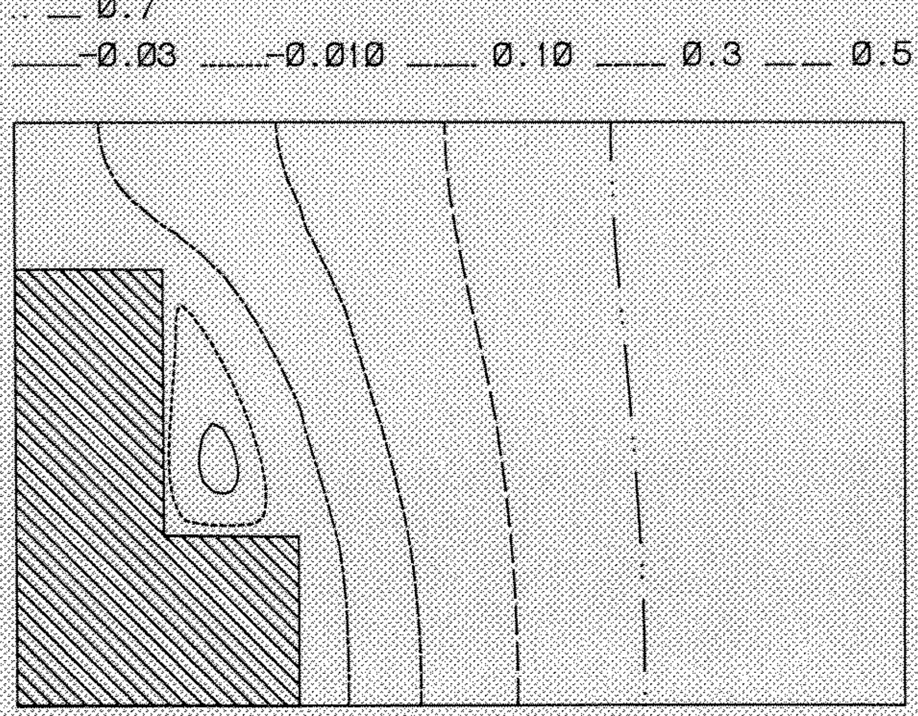


a)

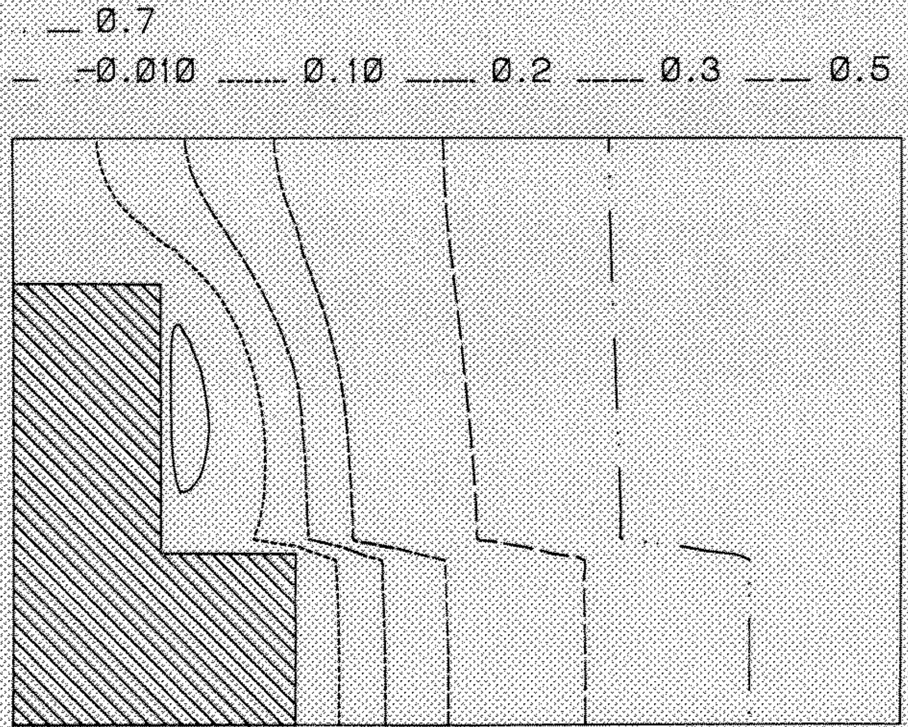


b)

Figure 2. Velocity vectors. a) Case 1. b) Case 2.

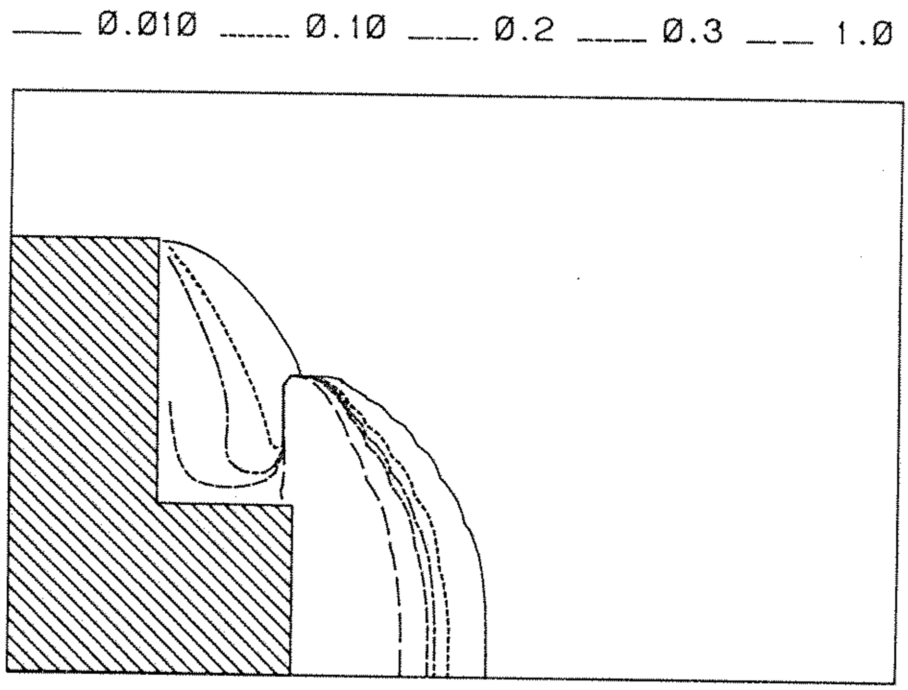


a)

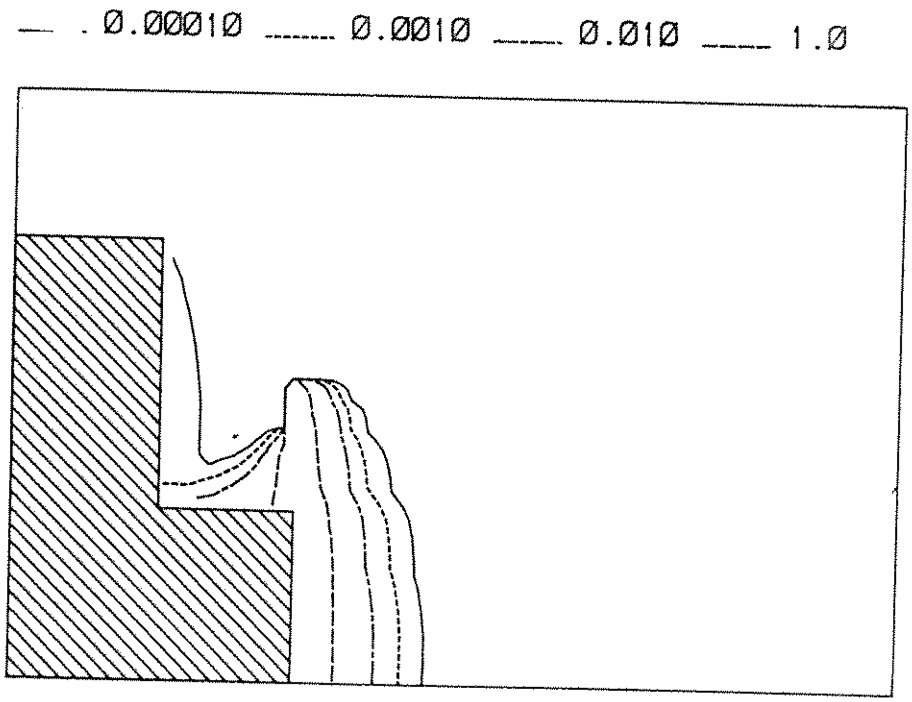


b)

Figure 3. Contours of stream function. a) Case 1. b) Case 2.

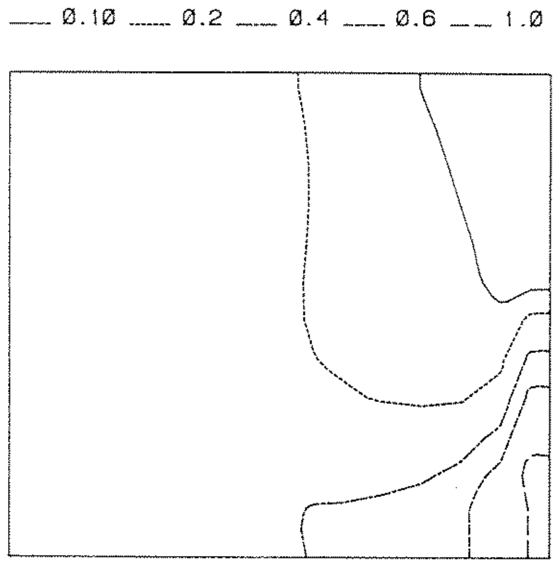


a)

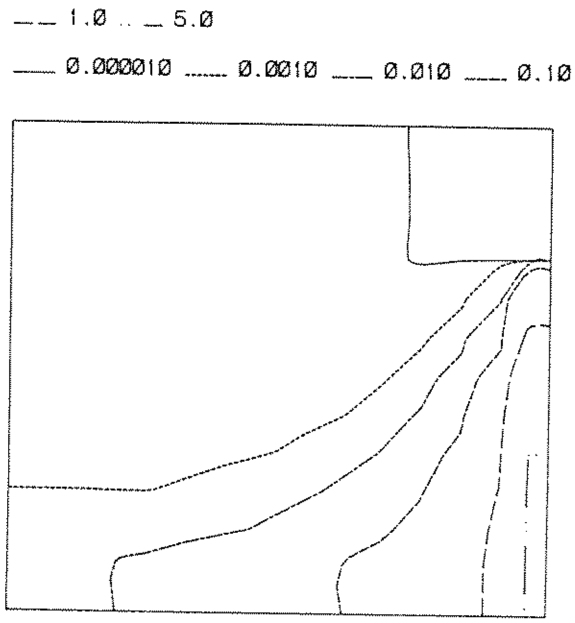


b)

Figure 4. Contours of concentration scaled with mean exit concentration for Case 1, $\langle c_{ex,1} \rangle$. a) Case 1. b) Case 2.



a)



b)

Figure 5. Contours of concentration scaled with $\langle c_{ex,1} \rangle$. a) Case 1. b) Case 2. The working section has been magnified in this figure. The lower boundary of the figure coincides with the whole width of the work table. The upper right corner is situatated at point P (see Fig. 1).

CONCLUSIONS

The air flow in a clean two-dimensional room, with a machine at which an operator (not included) is supposed to work, has been numerically simulated. Two cases have been investigated: a local exhaust was introduced at the work table in one case, while in the other no exhaust was used. The main object was to calculate the concentration of contaminant generated near the work table, in order to study whether it is advantageous (i.e. low concentration of contaminant near the work table) to introduce a local exhaust. The present study shows that it is indeed preferable to have a local exhaust at the work table; the concentration of contaminant is reduced (apart from the region near the edge of the work table) by an order of magnitude.

ACKNOWLEDGEMENTS

The author wishes to thank Ove Strindehag at Fläkt AB, Jönköping, Sweden, who initiated this work. The Swedish Council for Building Research and Fläkt AB have sponsored this work.

REFERENCES

Davidson, L. (1986), "Turbulence Modelling and Calculation of Ventilation Parameters in Ventilated Rooms", Rept. 86/10, Dept. of Applied Thermodynamics and Fluid Mechanics, Chalmers Univ. of Tech., Göteborg.

Davidson, L. and Hedberg, P. (1986), "A General Computer Program for Transient, Three-Dimensional, Turbulent, Recirculating Flows", Rept. 86/13, Dept. of Applied Thermodynamics and Fluid Mechanics, Chalmers Univ. of Tech., Göteborg.

Davidson, L. and Olsson, E. (1987a), "A Numerical Investigation of the Local Age and the Local Purging Flow Rate in Two-Dimensional Rooms", Proc. Room Vent-87, Stockholm, Sweden.

Davidson, L. and Olsson, E. (1987b), " Calculation of Age and Local Purging Flow Rate in Rooms", Bldg. Environ., Vol. 22, p. 111.

Davidson, L. and Olsson, E. (1987c), "Calculation of some Parabolic and Elliptic Flows Using a New One-Equation Turbulence Model", p. 411, Vol. 1, Part 1, 5th Int. Conf. on Numerical Methods in Laminar and Turbulent Flow, Eds. C. Taylor, W. G. Habashi and M. M. Hafez, Montreal.

Gosman, A. D., Nielsen, P. V., Restivo, A. and Whitelaw, J. H. (1980), "The Flow Properties of Rooms With Small Ventilation Openings", ASME J. Fluids Engng., Vol. 102, p. 316.

Hanel, B. und Scholz, R. (1978), "Exprimentelle und Numerische Untersuchungen Ebener, Isotermer Strömungen in Räumen bei

Unterschiedlichen Bedingungen des Zuluftstrahls", Luft- und Kältetechnik 2.

Hjerthager,B. H. (1979), "Flow, Heat Transfer and Combustion in Three-Dimensional Rectangular Enclosures", Thesis, Trondheim, NTH.

Holmberg, R., Larsson, M. and Sundkvist, S. G. (1975), "Calculation of Velocity Distribution in a Ventilated Room", (in Swedish), VVS, No. 5, p. 59.

Larsson, M. (1977), "Predictions of Buoyancy Influenced Flow in Ventilated Industrial Halls", Proceedings of Heat Transfer in Buildings, Dubrovnik.

Larsson, B., Neikter, K. and Strindehag, O. (1987a), "Air Distribution and Ventilation Efficiency in Clean Rooms", Proc. Room Vent-87, Stockholm.

Larsson, B., Neikter, K. and Strindehag, O. (1987b), "Ventilation Efficiency in Clean Rooms", 4th Int. Conf. on Indoor Air Quality and Climate, Berlin.

Leschziner, M. A. and Rodi, W. (1981), "Calculation of Annular and Twin Parallel Jets Using Various Discretization Schemes and Turbulence-Model Variations", ASME Journal Engng., Vol. 103, p.352.

Nielsen, P. V. (1973), "Berechnung der Luftbewegung in einem zwangsbelufteten Raum", Gesundheitsingenieur 94, Heft 10, pp. 299.

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

Patankar, S. V. (1980), "Numerical Heat Transfer and Fluid Flow", McGraw-Hill, New York.

Patankar, S. V. (1981), "A Calculation Procedure for Two-Dimensional Elliptic Situations", Num. Heat Transfer, Vol. 4, p. 409.

Raithby, G. D. (1976), "Skew-Upwind Differencing Schemes for Problems Involving Fluid Flow", Comp. Methods in Appl. Mech. and Engng., Vol. 9, p. 153.

Rodi, W. (1980), "Turbulence Models and Their Application in Hydraulics", International Association of Hydraulic Research, Monograph, Delft, The Netherlands.