NUMERICAL MATHEMATICAL MODELING OF AIR FLOW IN VENTILATED ROOMS

by

Professor Bjørn F. Magnussen
Division of Thermodynamics
Norwegian Institute of Technology
N-7034 Trondheim, Norway

Presented at ROOMVENT'90
International Conference in Oslo, Juni 13-15, 1990
Engineering Aero- and Thermodynamics of Ventilated Rooms.

NORGES TEKNISKE HØGSKOLE TRONDHEIM

NUMERICAL MATHEMATICAL MODELING OF AIR-FLOW IN VENTILATED ROOMS

by

Bjørn F. Magnussen
Division of Thermodynamics
Norwegian Institute of Technology
N-7034 Trondheim

SUMMARY

Mathematical models combined with experimental and theoretical investigations are necessary, and often efficient, tools for prediction of the effects of design and operational changes.

Numerical mathematical modeling combined with efficient computer codes and computers offers the opportunity to treat the Air-flow problem in ventilated rooms in a general way on the basics of the fundamentals in thermo- fluid-dynamics.
Such methods therefore bridge the gap between this area and other areas of fluid dynamics.

Besides the technology connected to the solution of the non linear partial differential equations, the modeling of physical important processes like turbulence play an important part. Because the technology usually implies that the turbulence in wall boundary layers is not included in the solution of the total flow field rather simple turbulence models can be applied with good results.

A model of this kind is the so-called k-ε model.

There exists today on the market computer codes that with consideable success can be applied for the treatment of ventilation problems for rooms, which can predict important features of the flow with the necessary accuracy.

1. INTRODUCTION

1.1 The Problem

Comfort and safety are important problems to consider in relation to ventilation of rooms. The comfort problem is related to draft which again is connected to air-flow and temperature distribution in the room, as well as to the spreading of odorous species in the room.

The air-flow in rooms is influenced by the inflow and outflow of the air as well as the internal flow in the room set up by the buoyancy forces.

Buoyancy forces are created by the interaction between air density differences and the gravitational field.

Internal heating and heat transfer from walls, people and objects in the room as well as the geometric configuration of the room have a strong influence on the buoyancy forces as well as on the flow field being set up.

Fire development and the spreading of smoke represent important safety aspects. The physics behind these phenomena is complicated, although very many of the underlying physical subprocesses are the same.

The opportunity is therefore offered to treat these very important problems with the same technology as for the comfort ventilation.

The following will discuss the numerical mathematical modeling of these processes, for ventilated rooms.

1.2 Mathematical Models

Mathematical models combined with experimental and theoretical investigations are necessary, and often efficient, tools for prediction of the effects of design and operational changes.

Mathematical modeling covers a great spectrum of methods from simple formula-type of expressions relating output to input variables, to more complex models, incorporating a number of submodels for elementary physical and chemical processes.

The operational domain of a mathematical model is limited to the domain where there exists experimental evidence of its necessary accuracy. The domain of a more complex mathematical model is determined by the domain for each of the incorporated submodels. The submodels often model elementary physical processes observed in a great variety of natural and technical environments. Mathematical models based on such submodels therefore generally have a wide operational domain.

The only general requirement for a mathematical model is that it should predict the output variables within its operational domain with the necessary accuracy. However, the mathematical models which models the elementary physical processes most accurately may be considered more valuable as they offer the opportunity not only to discuss practical results but also their physical implications.

2 FLOW MODELLING BASICS

2.1 Governing Equations

Fluid flow is governed by the conservation equations for mass, momentum and energy. These equations are quasi linear partial differential equations (PDE).

Conservation of mass is expressed by the continuity equation, and reads:

$$\frac{\partial \rho}{\partial t} = \frac{\partial(\rho u_i)}{\partial x_i} \tag{1}$$

Conservation of momentum is given by the Naviar Stokes equations, here written in tensor notation:

$$\frac{\partial(\rho u_j)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_i} = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_i}\left(\mu_{eff}\frac{\partial u_j}{\partial x_i}\right) + F_j \tag{2}$$

where F_j are body forces, i.e. buoyancy, and μ_{eff} is the effective viscosity.

The energy equation may be expressed as

$$\frac{\partial(\rho h)}{\partial t} + \frac{\partial(\rho u_i h)}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\mu_{eff}}{\sigma_h}\frac{\partial h}{\partial x_i}\right) + S_h \tag{3}$$

where S_h is an energy source.

2.2 General Transport Equation

Equations 1,2 and 3 may all be written in the following form:

$$\underset{I}{\frac{\partial(\rho\phi)}{\partial t}}+\underset{II}{\frac{\partial(\rho u_i\phi)}{\partial x_i}}=\underset{III}{\frac{\partial}{\partial x_i}(\Gamma_\phi\frac{\partial\phi}{\partial x_i})}+\underset{IV}{S_\phi} \tag{5}$$

where Φ is the transported variable.

This equation consists of four distinct terms, shortly explained as:

I) Transient term, accumulation of Φ during the time step dt.
II) Convective term, transport of Φ by convection.
III) Diffusive term, transport of Φ by diffusion.
Γ_Φ is a diffusion coefficient.
IV) Source term, local production or dissipation of Φ

The transport equation thus expresses a balance between these phenomena.

3 NUMERICAL MODELLING

3.1 General

For some special cases, it is possible to find analytical solutions to the equation system 1 to 3. This is, however, the exception. In general, we have to use other techniques to obtain quantative information on the flow field in question.

In recent decades the rapid development of computers have caused an increasing interest in numerical methods for solution of the PDE. Numerical methods have shown to be very useful in many fields of dynamics. They are used both as a supplement to experimental methods, and as an alternative to them.

3.2 Finite difference

There are several methods for numerical simulation of fluid flow available:
Finite element, Boundary element, Finite difference etc.

In the following the discussion will be limited to use the Finite difference technique, which shortly may be explained as follows. A finite number of grid points are distributed over the calculation domain. These grid points are surrounded by control volumes, see Figure 1. The equations are descretized over the control volumes. In this way each PDE will be represented by a set of non linear algebraic equations, one equation for each grid point for each variable.

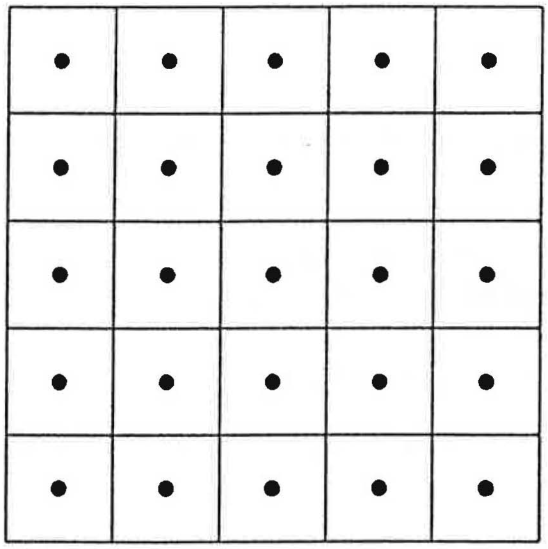


Figure 1: Grid points and control volumes

The resulting algebraic equations are solved on a digital computer by use of a solution algorithm. One such algorithm is the TDMA, i.e. Tri Diagonal Matrix Algorithm.

The non linear nature of the algebraic equations calls for an iterative solution procedure.

3.3 Staggered grid

Staggered or non-staggered grid may be employed. Staggered grid is often employed to obtain a consistent connection between the pressure gradient and the velocities. This implies that the velocities are calculated at the boundaries of the volumes in the original grid.

The grid for the velocity components are then staggered relative to the grid for the scalar variables; which are all calculated in the original grid points. Hence the name staggered grid. This is shown in Figure 2.

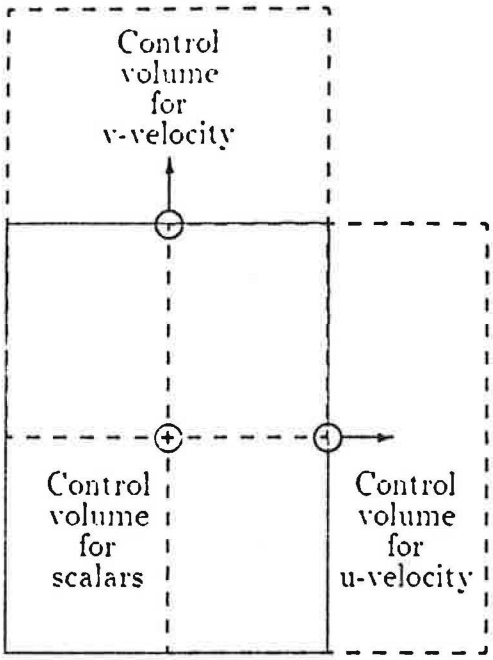


Figure 2: Grid points and control volumes

3.4 Discretization

The general transport equation Eq. 5 is discretized over the control volume. The results is a set of algebraic equations of the form:

$$A_p^{\phi}\Phi_p=\sum_k A_k^{\phi}\Phi_k+S_c \tag{6}$$

where k=all neighbours.

To avoid negative coefficients in eq. 6, which could cause physical unrealistic results, upwind differencing is employed.

3.5 Pressure correction

Several algorithms may be applied to solve the algebraic equation system. Probably the most frequently applied is the SIMPLE-algorithm proposed by Patankar and Spalding [1].

This algorithm introduces the so called pressure correction. An equation for the pressure equation is obtained by substitution of the momentum equation into the continuity equations. These equations are solved several times for each time or iterative step, and the pressure- and velocityfields are updated according to the pressure correction, until satisfactory continuity is obtained for each step.

3.6 Stationary or Transient calculations

Stationary solutions are obtained by iterative procedures. However, the time steps are critical parameters when executing transient calculations for problems like fire development.

The time steps in the calculations are normally dimensioned according to a Courant number of the order of unity. In order to increase the time steps, and thereby reduce the computer time, the rapid variations due to pressure waves may be neglected. This implies that the velocity of sound is assumed not to be the characteristic velocity in the Cournat number. The characteristic velocity will then be the velocity at which the flame propagates relative to a fixed frame of reference, i.e. the sum of the mean flow velocity and the turbulent flame propagation velocity.

$$U_c=\bar{u}+U_B \qquad (7)$$

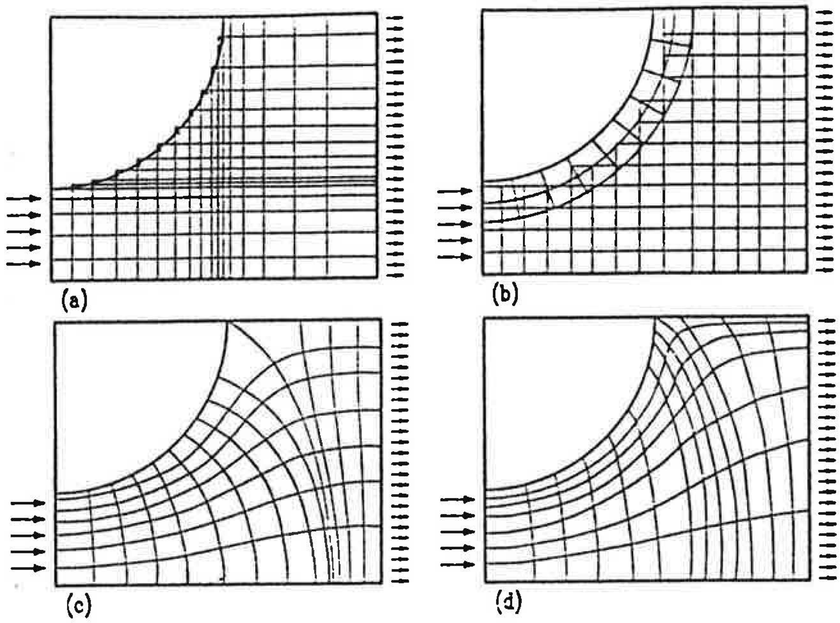
In a typical shear flow situation U_B is given by [2]

$$U_B=2u' \qquad (8)$$

where u' is the turbulent fluctuation velocity.

3.7 Orthogonal and non-orthogonal grid

Different computer codes have different options for the selection of grids. For complex geometries it may be necessary to use socalled "boddy fitted coordinates", however for most cases a cartesian grid is sufficient.



(a) Regular grid with stepwise boundary
(b) Composed grid
(c) Curvilinear orthogonal grid
(d) Curvilinear non-orthogonal grid

Fig. 3. Different grids.

4 PHYSICAL SUBMODELS

4.1 General

Special physical models have to be included in the calculations to take care of different physical phenomena.

4.2 Turbulence modelling

Turbulence can be modelled in very many different ways. The models may vary in complexity from simple eddy viscosity formula type of models, through algebraic stress models to complex Reynolds-stress differential equations.

The most importante duty of a turbulence model is to model the shear stress in the momentum equations. Very often in ventilation flows the pressure has a more dominating influence on the flow than the shear stresses. Under such cases the turbulence model selection is not very importante.

Very often the so-called k-ε model is applied. This is a model where the turbulence kinetic energy, k, and its rate of dissipation, ε, is modeled. This model is very useful. Much experience exists regarding its weak and strong features.

Our own experience is that this model can be extensively used for ventilation air flows. The weaknesses of the k-ε model is most pronounced when the flow is strongly non-isotropic, or when the Reynolds number of the flow is low.

Both these phenomena are most frequently observed, close to walls. However, the wall effects are usually incorporated into the computational domain through wall function methods, and therefor do not enter into the operational domain of the turbulence model.

The following describes the treatment of the k-ε model.

k and ε are determined from the following equations:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho u_i k)}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\frac{\mu_{eff}}{\sigma_k}\frac{\partial k}{\partial x_i}\right)+P-\rho\varepsilon \tag{9}$$

and

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho u_i\varepsilon)}{\partial x_j}=\frac{\partial}{\partial x_i}\left(\frac{\mu_{eff}}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_i}\right)+C_1\rho\frac{\varepsilon}{k}-C_2\rho\frac{\varepsilon^2}{k} \tag{10}$$

P is the turbulent kinetic energy produced by the main motion, and may be expressed as:

$$P=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_j}{\partial x_i} \tag{11}$$

The constants appearing in the turbulence model may be are taken from [3]

The effective viscosity is composed of the molecular and the turbulence viscosity.

$$\mu_{eff}=\mu_l+\mu_t \tag{12}$$

where the turbulence viscosity is expressed.

$$\mu_t=C_D\rho\frac{k^2}{\varepsilon} \tag{13}$$

Sometimes additional terms are included to account for low Reynolds number effects. These terms are often applicable only for the flow very close to walls.

When the Reynolds number is low the k-ε model without low Reynolds number terms may give too high turbulence viscosity because the viscous dissipation will be too low.

4.3 Combustion modelling

Combustion is an exothermic chemical reaction. The concentrations of the participating species are calculated by solving transport equations of the same type as equation 6. Transport equations are solved for the mass fraction of fuel and oxygen, and the other mass fractions are calculated from algebraic equations relating them to the mass fraction of fuel and oxygen.

The most important part of the combustion modelling is the reaction rate of the fuel. This reaction rate can be found by use of the Eddy Dissipation Concept of Magnussen [EDC].

4.4 Radiation

Knowing that the radiant heat transfer is carried over long distances, and even through vacuum, as opposed to conduction and convection which are local modes of heat transfer, it is easily realized that determination of the radiant fluxes is a complicated task.

The radiation travelling along a path within a medium, is attenuated by absorption and scattering, and is enhanced by emission and radiation scattered from other directions.

If scattering effects are neglected, it is possible to deduce a differential equation governing the radiation intensity along a path through an absorbing and emitting medium [1].

5 TREATMENT OF BOUNDARIES.

The computational domain may be bounded by solid walls, inlets and outlet, and interiors.

5.1 Solid wall

Due to the complicated flow structure in the near wall regions of turbulent flows, it is common practice not to calculate the details of the flow in this region, but to match the calculation to the "universal logarithmic law of the wall".

The wall function method is applied for all variables like the turbulence variables, k, and ε, as well as for the energy equation.

5.2 Inlet and outlet openings

Openings can be forced inlets or outlets or free boundaries.

For forced ventilation constant velocities are incerted for the inlet and outlet openings. The outlet velocities must be corrected for density changes in the room.

The flow through the free boundaries is calculated by the solving the basic equations 1, 2 and 3, also for the grid points on the boundaries. The pressure surrounding the module is calculated from the hydrodynamic static pressure field and the wind outside the module.

5.3 Interiors

Interiors may be modeled by blocking and porosity techniques. Heating and cooling effects of the interiors will appear as sources and sinks in the energy equation.

6 SOLUTION PROCEDURE

When applying the pressure correction method the solution procedure transformers into:

1. Assume or have a pressure and velocity field

2. Solve momentum equations

3. Solve for pressure correction

4. Correct velocities

5. Check continuity equation

6. If necessary return to point 3.

7. Solve for other variables (Energy, Turbulence, etc.)

8. Got to point 1 and solve for a new iteration or time step.

7 ACCURACY, NUMERICAL ERRORS COMPUTATIONAL TIME.

For most engineering purposes the accuracy will be sufficient if the grid is not too coarse and if the grid is aligned with the main flow direction. It is important to bear in mind that the continuity equation is always accurately solved. Therefor a release of a certain quantity of a certain specie can always be retraced in the computational domain. Turbulence and numerical diffusion may have effected the spreading. This, however, may not be serious for the enterpretation of the practical consequences of the release.

Computational times as low as 6 minutes have been observed for approximately 50 000 gridpoints on a CRAY XMP-28

8 CODES

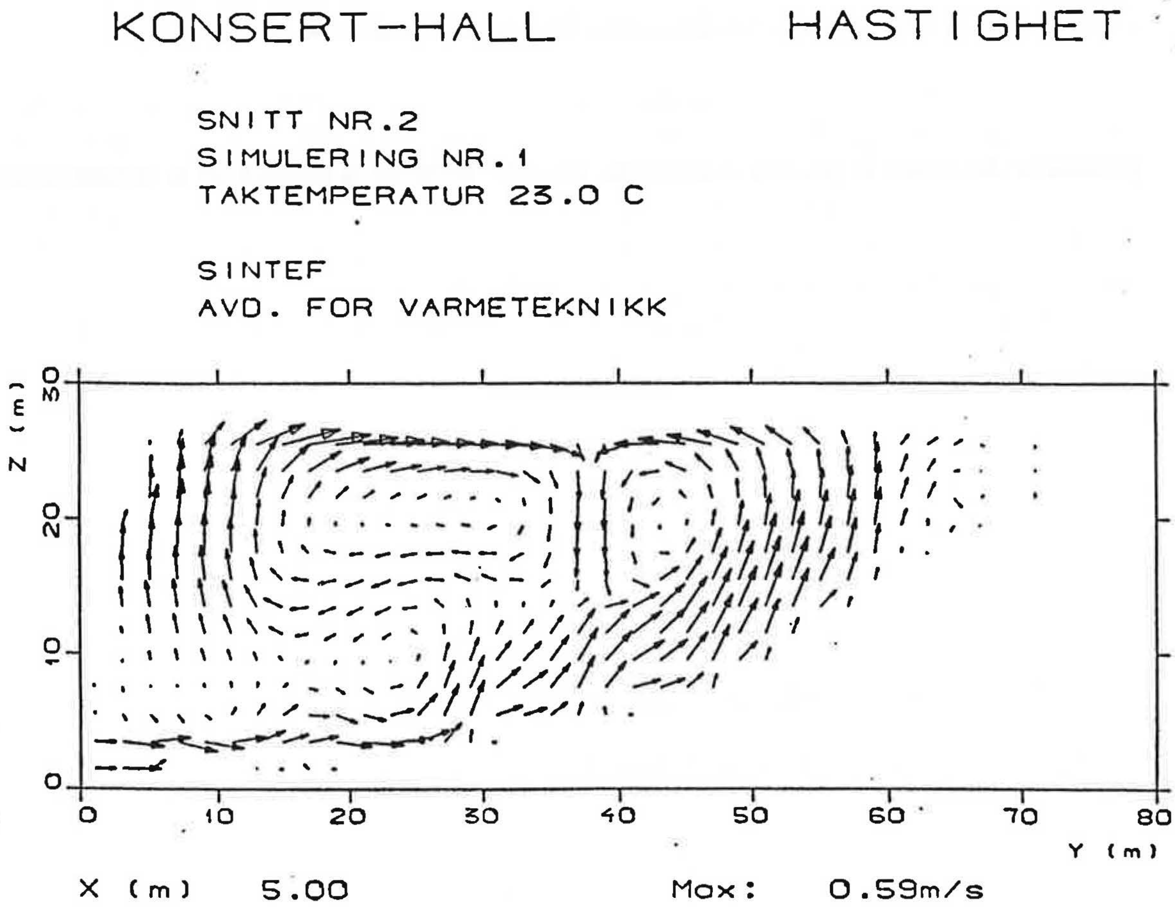
There are many codes available on the market like

PHOENIX, FLUENT, FLOW-3D KAMELEON II, SPIDER etc.

9 PRACTICAL RESULTS

Figures 4 and 5 demonstrate some computational results for a simulation by KAMELEON of the Oslo City Hall.

Figures 6 and 7 demonstrates results for fire and smoke dispersion studies performed on the basis of KAMELEON for a ventilated offshore modul.



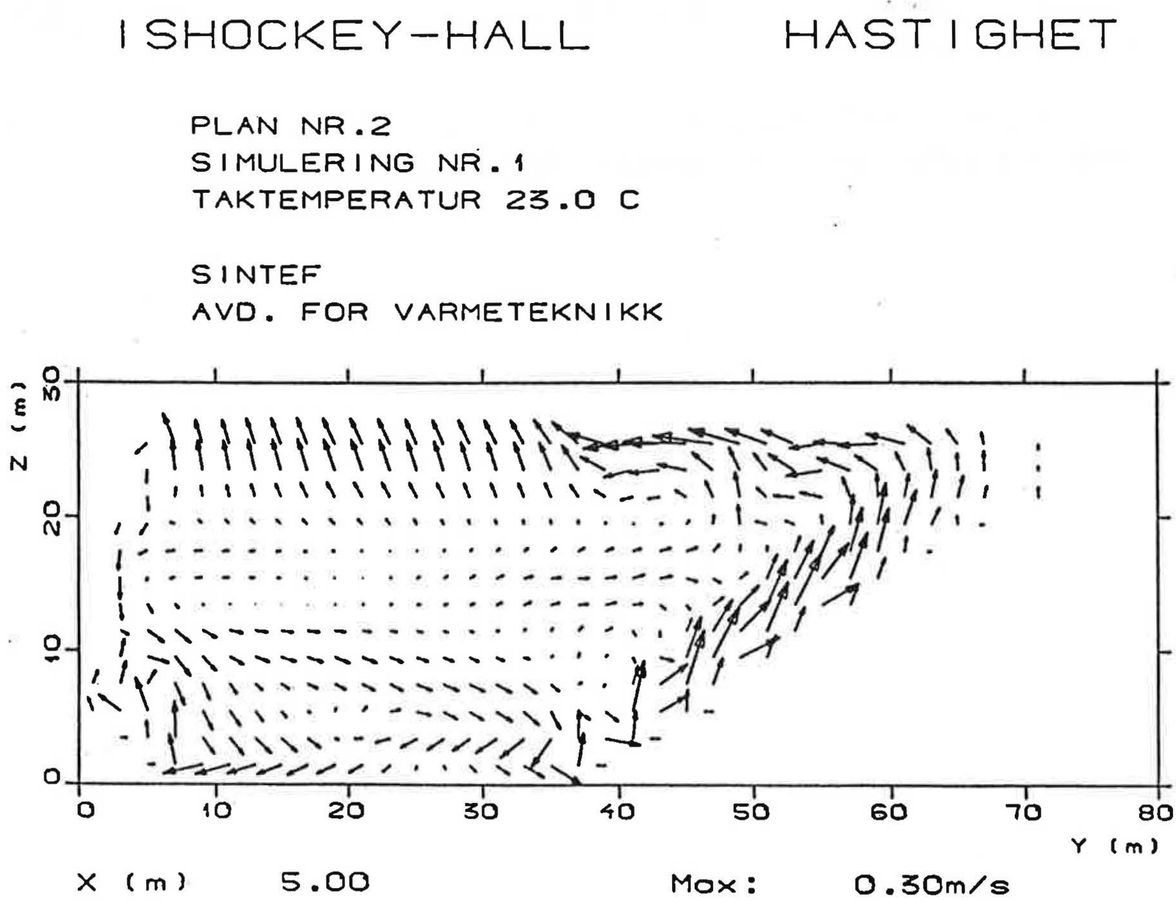


Fig. 4. Flow in Oslo City Hall

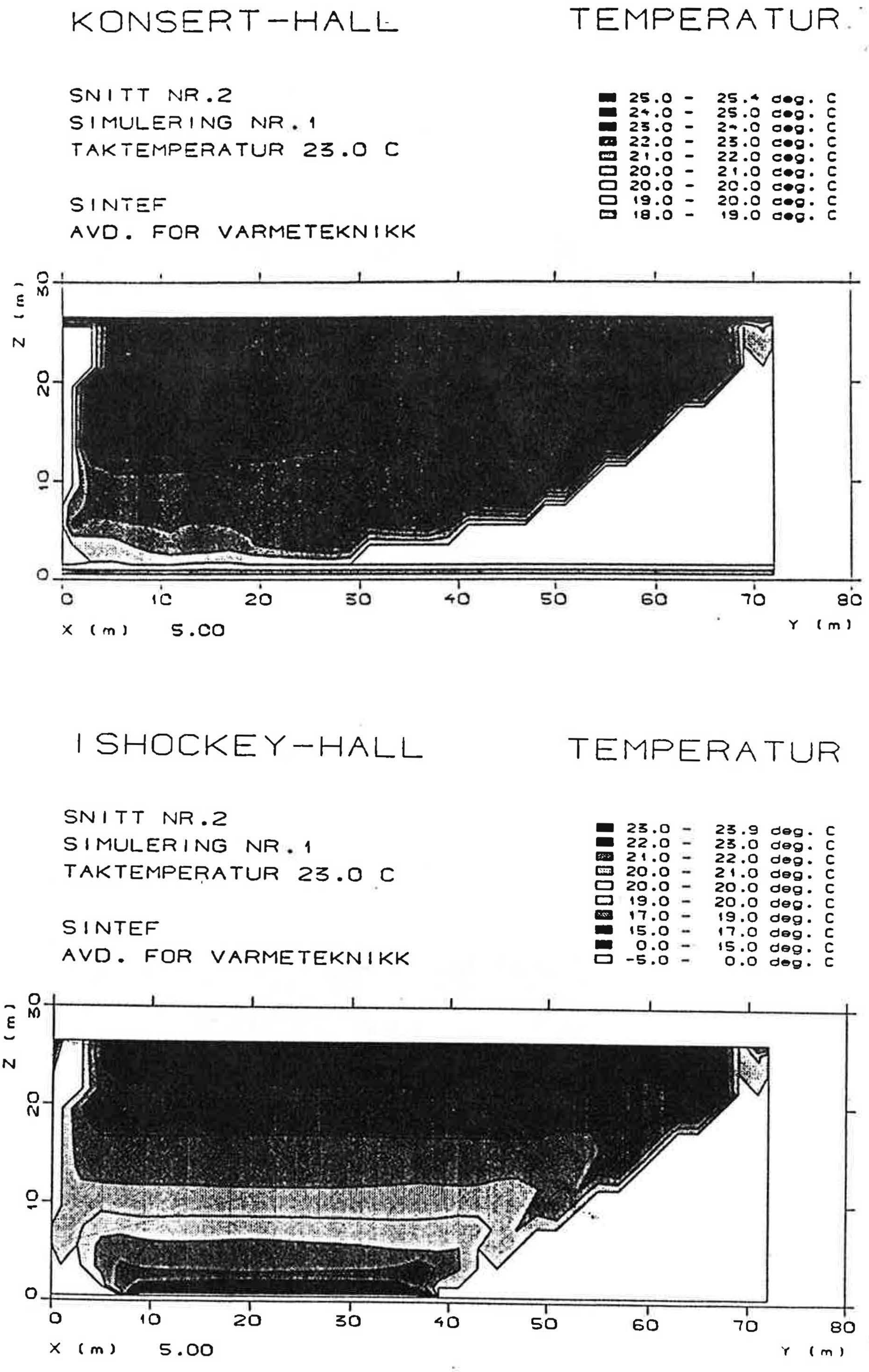


Fig. 5. Temperatures in Oslo City Hall

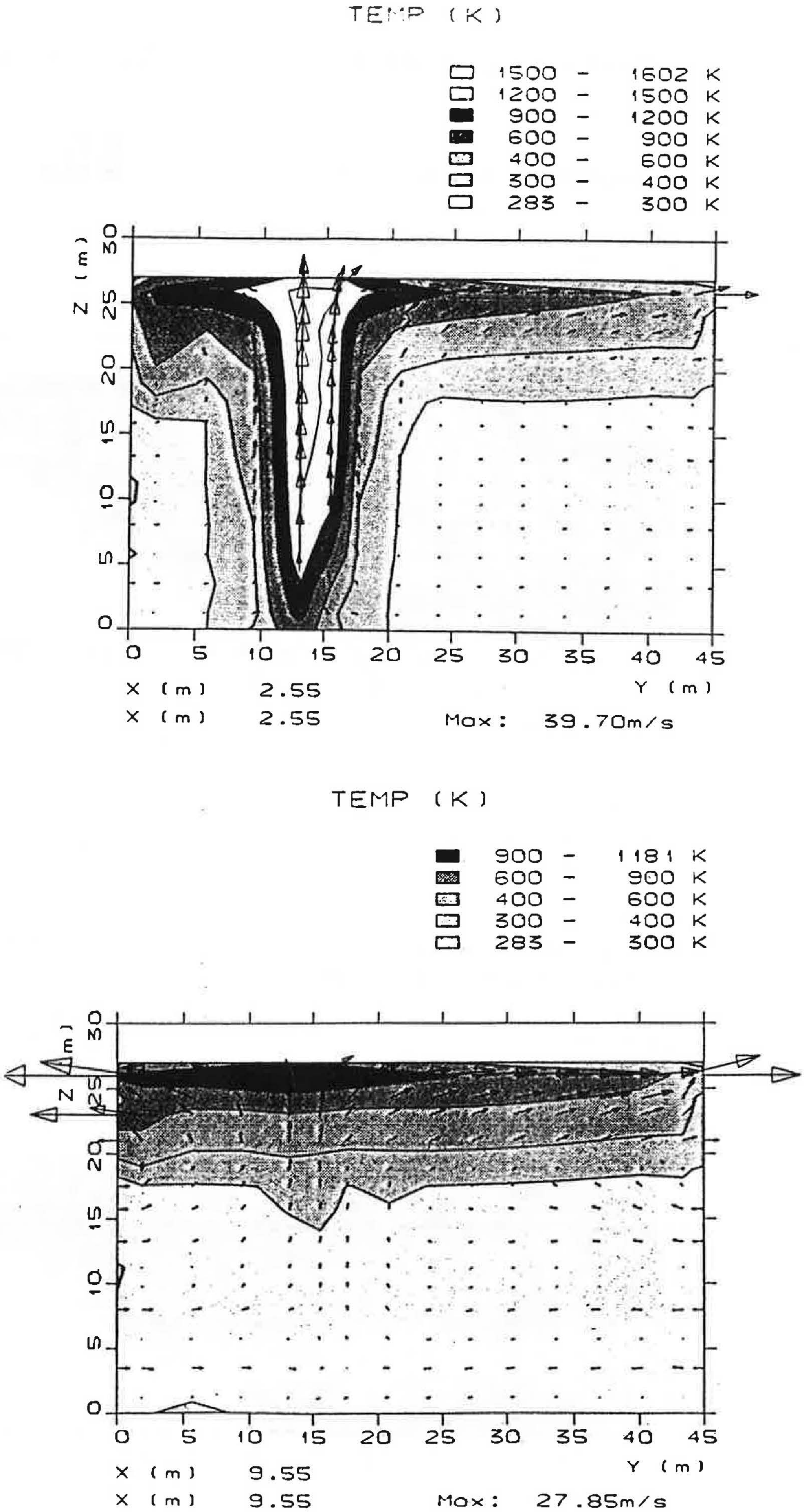


Fig. 6. Temperatures, gas-fire in offshore module

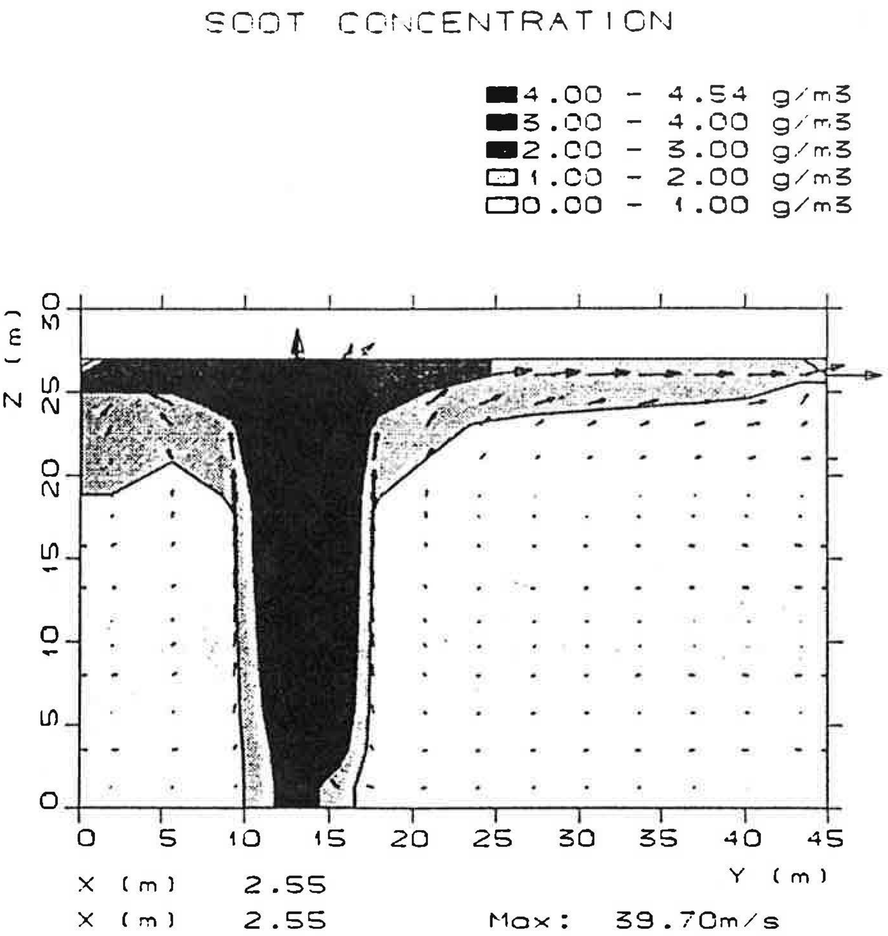


Fig. 7 Soot concentration in off-shore module

10 CONCLUSION

Technology is available for simulation of air-flow in rooms with the necessary accuracy, both for ventilation purposes and even for fire and smoke dispersion cases. There is still need for improvements on the fundamental as on the applied side.

11 ACKNOWLEDGEMENT

The valuable assistance by Jens Holen, Bård Grimsmo, Ole Melhus and Anita Juberg is greatfully acknowledged.

REFERENCES

[1] Patankar, S.V. and Spalding, D.B. "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensjonal Parabolic Flows", Int. J. Heat Mass Transfer, Vol. 15, pp 1787-1806, 1972.

[2] Byggstøyl, S. and Mangussen, B.F. "A model for Flame Extinction in Turbulent Flow", 4th Symp. on Turbulent Shear Flows, Karlsruhe, 1983.

[3] Launder, B.E. and Spalding D.B. "The Numerical Calculation of Turbulent Flows", Computer Methods in Applied Mechanics and Engineering, No. 3 pp 269-289, 1972.

[4] Magnussen, B.F. "On the Structure of Turbulence and a Generalized Eddy Dissipation Concept for Chemical Reaction in Turbulent Flow", Paper presented at 19th AIAA Science Meeting, St.Louis, Missouri, USA, Jan. 12-15, 1981.

[5] DeMarco, A.G. and Lockwood, F.C. "A New Flux Model for the Calculation of Radiation in Furnaces", Italian Flame Day, La Rivista dei Combustibility, No. 5-6, Vol. 29., pp 174-196, 1975.

[6] Shah, N.G. "The Computation of Radiation Heat Transfer", Ph.D. Thesis, University of London, Faculty of Engineering, 1979.

[7] Berge, G. "Numerisk programsystem basert på fleirdimensional ortogonal koordianatar for simulering av forbrenning, masse- og varmetransport", Dr.ing. Thesis, University of Trondheim, The Norwegian Institute of Technology, Division of Thermodynamics. (1982).

[8] Magnussen, B.F., and Hjertager, B.H. "On Mathematical Modeling of Turbulent Combustion with special Emphasis on Soot Formation and Combustion", The Sixteenth (International) Symposium on Combustion, 1976.

[9] Orlof, L. Modak, A.T. and Markstein, G.H. "Radiation from Smoke Layers", 17th Symposium (Int.) on Combustion, The Combustion Institute, pp. 1029-1038, 1978.

[10] Leckner, B, "Spectral and Total Emissivity of Water Wapour and Carbon Dioxide", Combustion and Flame, 19, pp. 33-48, 1972.

[11] Magnussen, B.F.,"Modeling of NO_x and Soot Formation by the Eddy Dissipation Concept", 1 st Topic Oriented Technical Meeting, Amsterdam, Holland, 17-19 oct. 1989.