I Mech E Conference on "Heat and Fluid Flow in Nuclear and Process Plant Safety", London 1983.

C97/83

The use of a computational method to assess the safety and quality of ventilation in industrial buildings

T W BROYD, BSc, PhD, R B DEAN, BSc, PhD, CEng and S G OLDFIELD, BSc, PhD, DIC
Atkins Research and Development, Epsom, Surrey
A MOULT, MSc, PhD, CEng
The Institut Cerac S.A., Switzerland

SYNOPSIS Industrial buildings, particularly those containing nuclear and process plant, often require high standards of ventilation in order to cope with unusual features of the operations or process which take place within the buildings. Four examples of recent studies carried out by the present authors are given in this paper: storage of coal in a covered stockyard, a chlor-alkali plant, a clean room, and the turbine hall of a nuclear power station. In each of these examples, quite detailed information was required about air flows, temperatures and gas concentrations, and it was decided to use the two-equation k,ε model of turbulence to help predict these variables. This is solved with equations for momentum and continuity by finite differences. It is concluded that complex computer programs of this kind can provide valuable assistance in support of the more traditional hand calculations using BS codes and CIBS guides. However, careful engineering judgement must be exercised in the use of the programs and in the interpretation of the results.

1 INTRODUCTION

The increase in capital and running costs of forced ventilation systems in industrial buildings, together with the power supply limitations in cases such as offshore platforms, have highlighted the need for more efficient HVAC systems. For example, potential energy savings of over 50% have been identified (see references 1,2,3 and 4) in cases where 'over design' has been employed to compensate for lack of more accurate design procedures. Furthermore, a reduction in the use of HVAC systems in recent years has led to a greater reliance on natural ventilation. Whilst this is a desirable trend from the point of view of energy conservation, it is important that the design methods used are accurate enough to ensure that the necessary quality and safety criteria are being met. This is particularly so where critical air temperatures or gas concentrations are concerned as, for example, in the wellhead modules of offshore platforms, the storage of nuclear fuel, and in many types of industrial processes.

Section 2 of this paper discusses the various specialised techniques which may be used as design aids to assess the quality and safety performance of ventilation systems in industrial buildings. Section 3 focuses on one of these design aids, computational modelling, which may be employed to enhance the accuracy of the more conventional HVAC design procedures. Section 4 gives examples of the application of computational modelling in some design studies recently carried out by Atkins Research and Development.

The following examples have been chosen because they demonstrate how the requirements of energy, quality and safety place demands on the accuracy of the analysis procedure:

(a) Safety (+ energy) criteria: concentrations of methane released from a stack of coal in a building.
(b) Safety criteria: discharge of chlorine from a fractured pipe in a building at a chlor-alkali plant.
(c) Energy (+ quality) criteria: air movements within a clean room.
(d) Quality criteria: temperatures and air movements within the turbine hall of a nuclear power station.

The difficulties of obtaining reliable results from scale models are also discussed and it is concluded that computational methods have an increasingly important role to play in the design of safe ventilation systems for industrial buildings.

2 NOTATION

Ar	Archimedes number
C_1, C_2, C_D	Constants in turbulence model
C_p	Specific heat at constant pressure (kJ/kg K)
D	Characteristic length (m)
g	Gravitational acceleration (m/s^2)
G_k	Turbulence generation term (kg/ms^3)
k, ε	Turbulence quantities (m^2/s^2, m^2/s^3)
P	Pressure (N/m^2)
Pr, σ	Prandtl number
Re	Reynolds number
S_ϕ	Source term in differential equation
$(T-T_o)$	Temperature difference from reference (K)
t	Time (s)
u,v,w	Velocity (m/s)
x,y,z	Cartesian co-ordinates
x, r, θ	Polar co-ordinates
β	Volume expansion coefficient (K^{-1})
Γ_ϕ	Diffusion coefficient (kg/sm)
∂	Partial differential operator
μ	Dynamic Viscosity (Ns/m^2)

ρ	Density (kg/m^3)
ν	Kinematic Viscosity (m^2/s)
ϕ	General property

Suffices

o	Reference condition
eff	Effective
k	Turbulent kinetic energy
L	Laminar
T	Turbulent
ϕ	General property

3 DESIGN AIDS

The four main methods which can be used in the design of HVAC systems for industrial buildings are:

3.1 Design codes and practices;
3.2 Full-scale measurements;
3.3 Scale model measurements;
3.4 Computational modelling.

3.1 Design codes and practices

Existing design practices and guide lines employ a wide range of simple empirical formulae using bulk heat and mass exchange between rooms, and assuming uniform conditions (see references 5,6 and 7). In many cases these are sufficient but, in certain applications, more detail of the flow conditions and temperature distributions within rooms may be needed to design effectively and with confidence, in order that the desired quality and safety of ventilation is achieved. In addition, increased confidence in design can lead to significant energy savings in installation and running costs, as suggested in the clean room investigation described in Section 5.

3.2 Full-scale measurements

Full-scale measurements are potentially the most accurate means of obtaining data. However, since specialised HVAC systems are generally 'one-off' designs, these measurements can only be used for retrospective design changes. Despite the cost and difficulties in obtaining full-scale measurements under controlled conditions, data of this type are vital to computational models for validation purposes.

3.3 Scale model measurements

The use of scale models can alleviate some of the practical problems inherent in full-scale measurements. Wind tunnel tests are frequently used in situations where it is suspected that there may be an unusual surface pressure distribution on a building, owing to the proximity of other buildings or geographical features. A good example of this is the modular design of the topside layout of offshore platforms (Fig.1). In this case, there is an additional problem which is that the hot exhaust gases from the turbine generators installed on the deck may enter the HVAC inlets, unless care is taken to locate the turbine exhausts and HVAC inlets in optimum positions with respect to the wind direction. The work of Plate et al (reference 8) describes typical wind tunnel tests carried out on a ventilating system of a building.

This type of model test can give useful predictions of bulk average quantities, such as air change rates per hour, but it is impracticable to obtain accurate distributions of velocities and temperatures within the enclosed space. Furthermore, it is difficult to deal satisfactorily with the situation of very low wind speeds where temperature-driven ventilation dominates. Under these circumstances, model tests are extremely difficult to carry out and interpret. This is because it is impossible to achieve full dimensional similarity of the Prandtl, Reynolds and Archimedes numbers (reference 9) in both model and prototype:

$$Pr = \mu C_p/k \qquad (1)$$

$$Re = uD/\nu \qquad (2)$$

$$Ar = \frac{g\beta D(T-T_o)}{u^2} \qquad (3)$$

It can be seen that satisfying Prandtl and Reynolds numbers makes it impractical to satisfy Archimedes number, as this would require very high model flow temperatures (of the order model scale cubed).

3.4 Computational modelling

The use of computer programs in all aspects of engineering design is now widespread. The energy budget of buildings, HVAC network operation, and air infiltration and distribution are a few examples of topics in the area of ventilation which can now be handled by the use of computer programs (reference 10). Furthermore, considerable effort is in progress around the world to develop new and more sophisticated programs (reference 11). Model tests and full-scale measurements will play an important rôle in this.

There are distinct advantages in the use of computational methods, such as flexibility and repeatability. Achieving a useful level of accuracy at an acceptable cost is also an important consideration, particularly where more sophisticated numerical methods are being employed, such as in the calculation of laminar and turbulent convective flows. It is on this particular aspect of ventilation and its treatment in design studies that the remainder of this paper now concentrates.

4 COMPUTATIONAL APPROACH

Until relatively recently it was possible to simulate only very simple flow problems by computational methods, owing to limitations in computer power and size. This situation has changed dramatically over the last ten or fifteen years. The growth in computer power has encouraged development of sophisticated methods of computational analysis, to the extent that it is now possible to simulate complex, three-dimensional turbulent flow problems to useful levels of accuracy. However, considerable caution must be exercised in the use of advanced numerical techniques and turbulence models, and skilled engineering judgement will continue to play a very important role in the interpretation of the results.

4.1 The mathematical formulation

Suitable numerical techniques have been developed which may be used to solve the three momentum equations, mass conservation, the energy equation and equations for concentrations of any number of either reacting or inert chemical species. A turbulence model may also be included, consisting of one or more transport equations.

Within the above framework, the independent variables are the three components of a co-ordinate system (x,y,z for cartesian co-ordinate systems, x,r,θ for cylindrical polar co-ordinate systems) and the time, t. Time-averaged dependent variables include the three velocity components u, v and w, the pressure p, the mass fraction m_j of chemical species j and the stagnation enthalpy h (or temperature T). Additional variables are found in the turbulence model which, for the two-equation model presently adopted (described in Section 4.2), are the turbulence kinetic energy k and its rate of dissipation ε.

The elliptic differential equations for the transport of the above variables in a cartesian co-ordinate system can be conveniently presented in a general form (see reference 12):

$$\frac{\partial}{\partial t}(\rho\phi) + \frac{\partial}{\partial x}(\rho u\phi) + \frac{\partial}{\partial y}(\rho v\phi) + \frac{\partial}{\partial z}(\rho w\phi) - \frac{\partial}{\partial x}\left(\Gamma_\phi\frac{\partial\phi}{\partial x}\right) - \frac{\partial}{\partial y}\left(\Gamma_\phi\frac{\partial\phi}{\partial y}\right) - \frac{\partial}{\partial z}\left(\Gamma_\phi\frac{\partial\phi}{\partial z}\right) = S_\phi \qquad (4)$$

where ϕ stands for the general variable, Γ_ϕ is the appropriate exchange coefficient for the variable ϕ, S_ϕ is the source/sink term of ϕ (which also includes any other terms which cannot find a place on the left hand side of the equation), and ρ is the density. The advantage of writing the equation in the form given by (4) is that additional equations can be incorporated very easily into the solution sequences of a computer program.

The variables Γ_ϕ and S_ϕ depend on the problem being solved and are given in Table 1, which summarises the equations in the form in which they are solved.

The terms involving the velocity divergence div v in the source terms of the momentum equations have been ignored. For constant density flows div v is identically zero. For most other flows handled by the present procedure, div v is expected to be small compared to other terms and is omitted for simplicity.

4.2 The turbulence model

A widely tested turbulence closure model is the high Reynolds number (k,ε) two-equational model of Harlow and Nakayama (reference 13). This model requires the solution of two differential equations, for the two turbulence characteristics, the kinetic energy of turbulence k, and its dissipation rate, ε.

The governing differential equations are presented in Table 1 where G_k is the generation term for the kinetic energy of turbulence and is given by:

$$G_k = 2\mu_T\left\{\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2 + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)^2\right\} \qquad (5)$$

In this model the turbulent viscosity, μ_T, is related to k and ε via:

$$\mu_T = C_D\rho k^2/\varepsilon \qquad (6)$$

and the effective viscosity is given by:

$$\mu_{eff} = \mu_T + \mu_L \qquad (7)$$

The local effective exchange coefficient, $\Gamma_{eff,\phi}$ for the transport of the scalar property ϕ, is calculated from:

$$\Gamma_{eff,\phi} = \frac{\mu_L}{\sigma_{L,\phi}} + \frac{\mu_T}{\sigma_{T,\phi}} = \frac{\mu_{eff}}{\sigma_{eff,\phi}} \qquad (8)$$

where μ_L and $\sigma_{L,\phi}$ are the molecular viscosity and the laminar Prandtl or Schmidt number, respectively, and μ_T and $\sigma_{T,\phi}$ are their turbulent counterparts, to take account of the effect of turbulence on mixing. This turbulence model includes four constants, their recommended values being: $C_D = 0.09$, $C_1 = 1.43$, $C_2 = 1.92$, $\sigma_{k,eff} = 0.9$. The value of $\sigma_{\varepsilon,eff}$ is calculated from the relation $k^2/(C_1^2-C_2)C_D^{\frac{1}{2}}$ where κ is the Von Karman constant (= 0.4) (see Jones, 1971).

4.3 Reduction to steady, two dimensional flow

The computing effort and present-day cost for a fully unsteady, three-dimensional fluid flow problem is often unacceptably high. Furthermore, it is doubtful whether, even with a fine grid in 3D, results of sufficient accuracy can be obtained with current turbulence models. In many cases, however, it is adequate for engineering purposes to reduce a particular problem to one or two spatial dimensions, and also use either a single steady-state computer run, or a number of discrete, steady-state runs to simulate an unsteady flow problem. This is particularly advantageous for cases where a qualitative assessment of design modifications is required, since a number of two-dimensional calculations may be considered for the same cost as a single three-dimensional analysis. However, we emphasise that this approach means that the results can only be interpreted in a qualitative sense or, at best, in a 'banded' quantitative sense.

In the next Section, various examples are given of the two-dimensional approach applied to the ventilation of industrial buildings. The way in which the mathematical model was assembled in each case is described and the manner in which the results have been interpreted for engineering purposes is explained. Each of the cases quoted were parts of recent studies carried out for clients by Atkins Research and Development.

5 EXAMPLES OF VENTILATION STUDIES BY COMPUTATIONAL ANALYSIS

In all four examples given below, the Atkins finite difference program CAFE has been used to predict the air flows, gas concentrations and temperatures in each of the process buildings concerned. CAFE (Computer Aided Flow Evaluation) solves the equations given in Table 1 and eqn. 4 and employs the k,ε model of turbulence, also given in Table 1 (see reference 14).

Only a sample of the results from CAFE has been presented in each case to illustrate the type of output data that can be obtained. The details of the input data and range of conditions studied are not given, as these are confidential to the clients concerned.

5.1 Concentrations of methane released from a stack of coal in a building

In this example, the main criterion was Safety but Energy was also an important consideration.

At the Gascoigne Wood site of the UK National Coal Board (NCB) Selby coal mine project, coal will be brought to the surface by conveyor belt and stacked temporarily in a 43,000t capacity covered stockyard building, one end of which is closed and the other end entirely open to the atmosphere. This end, and the ducts proposed for the side walls and roof, provide ventilation for the building. Since coal contains a small quantity of methane gas which is emitted at a decreasing rate over a period of time, adequate ventilation must be available to ensure that explosive concentrations of the gas do not occur.

The behaviour of the gaseous impurity in air depends on the density of the gas, on its concentration in the mixture and on the circulation of the air. Normally, on a windy day, it is expected that there will be sufficient circulation due to the air entering via the inlet ventilation ducts, but on a calm day this may not be the case. Indeed, it is likely that the worst situation is when there is no wind and no movement of machinery, so that the stack of coal is releasing methane into a 'quiescent' atmosphere.

A study was therefore required to determine the extent of air movement within the covered stockyard building and in the surge bunker on a calm day, and whether in fact sufficient natural circulation may be set up by the temperature of the wall and coal stack to prevent locally high concentrations or 'pockets' of methane from developing.

The covered stockyard building is 312m long, 61m wide and 22m high. It was therefore decided that a two-dimensional analysis in a vertical plane of 1m width would give useful results for the predictions of methane concentrations.

The first stage of the analysis was to estimate the release rate of methane from the coal stack. An empirical formula supplied by the NCB was used for this purpose. The next stage was to determine the solar heat gain and loss of the building during a 24hr period. The Atkins energy analysis program ATKOOL was run for several types of day such as sunny/winter and cloudy/summer and, knowing the thermal properties of the structure, the roof and wall temperatures were calculated. This information was used to estimate the air change rate by natural convection due to the temperature gradient only, i.e. in the absence of wind. The CIBS/IHVE guidelines were used for this purpose.

The final stage of the analysis was to predict the natural circulation patterns and contours of methane concentration within the building. CAFE was run with the previously calculated wall temperatures and air inflow rates as boundary conditions. A sample set of results for the covered stockyard buiding is shown in Figure 2.

By repeating this analysis for different ventilation inlet and outlet areas, it was possible to converge on the optimum vent sizes which would ensure sufficient ventilation by natural convection to prevent dangerous levels of methane from being reached. The dimensions obtained from this study were accepted by the NCB and have since been incorporated in the design of the covered stockyard building.

5.2 Discharge of chlorine from a fractured pipe in a building at a chlor-alkali plant

In this example, the main criterion was Safety.

Hazard Analysis and Risk Assessment are becoming a very important part of the design of nuclear and process plant. In this study, an accident scenario was proposed in which a pipe containing a flow of chlorine gas at ambient temperature fractures in a chlorine process building. This was closed on all sides except for a door to the east of the building. Ventilation was provided by two extractor fans built into the roof to provide twelve air volume changes per hour. It was assumed that the maximum chlorine production rate of 8.9kg/minute would represent an upper limit on the amount of chlorine which might escape from the pipe.

The information that was required was an estimate of the time taken for the concentrations of chlorine emerging from the extractor fans in the roof to reach a dangerous level. In order to determine this, it was necessary to predict how the gas escaping from the fractured pipe would mix with the air in the building. The CAFE program was used so that the negative buoyancy effects, caused by the dense chlorine, would be taken into account implicitly by the equations.

At time t = 0, a 130 kg mass of chlorine was introduced in a 0.9m thick layer on the floor, with mass concentration $C_m = 0.624$. The flow patterns and concentration (C_m) contours for the time-dependent flow within the building are shown in Figure 3 for the time intervals shown. These indicate that the mixing is fairly rapid, with the $C_m = 2\frac{1}{2}\%$ contour (corresponding approximately to $C_v = 1\%$) almost reaching the outlet vent at the top of the building after 100 seconds.

The time histories of the concentration of gas at the roof vent and also of the percentage of the initial mass of chlorine remaining within the building, are shown in Figure 4. The results suggest that the rate of flow of chlorine after

100 seconds is of order 0.1 kg/second, implying that the density of the emerging gas is of order 3% higher than that of air. Because the density differences are small, a Gaussian plume model was used to represent the outside dispersion of chlorine away from the Process Building.

5.3 Air velocities within a clean-room

In this example, the main criterion was Energy conservation but Quality and Safety were both very important aspects.

Clean rooms are being increasingly used in the nuclear and process industries for operations requiring a particle-free environment. Although filters (usually in the ceiling void) eliminate most particles of dust before air enters a clean-room, design requirements generally stipulate that a laminar air flow should exist in a direction parallel to all 'clean' components.

A half section through a typical clean-room is shown in Figure 5. Filtered air is blown in through seven inlet filters in the ceiling, and extracted close to ground level. Streamline plots of a two-dimensional computer simulation of flow within the room are shown in Figure 6. The same flow predictions are shown in vector form on Figure 7, on which measured data points from the clean-room in question are also shown. A good model/prototype agreement was obtained, except in the bottom right hand corner of the Figure, where the measured velocities are close to the zero error of the hot wire anemometry equipment used for data collection.

Figure 8 shows computed flow results (in the form of streamlines of flow) with two of the filter panels removed, i.e. with a 28% reduction in inlet air flow. The flow directions at 1m above floor level are remarkably similar to those with a full filter coverage. This implies that there may be significant opportunities for energy saving through reductions in filter panel inlet area. Further experimental and computational work is to be carried out to investigate this possibility.

The design brief for this clean room was to provide Class 100 conditions to Am. Fed. Std. 209B at a point 1 metre above the floor by laminar flow. The room has produced particle counts of less than 10/ft^3 for particles greater than 0.5 micron and is therefore well within the requirements of the design brief.

5.4 Air quality within a power station turbine hall

In this final exmaple, the main criterion was Quality of the working environment.

The circulation of air within a nuclear power station turbine hall of overall dimensions 180m long x 35m high x 50m wide was analysed. A diagrammatic section through the turbine hall is shown in Figure 9. Inlet ducts along the lower boundary and outlet fans along the top boundary provide ventilation for the building. The turbine machinery (Blocks 1-6 in Figure 9) provide sources of heat to the hall, and duct-work around this equipment has been represented by five vertical vents (vents a-e in Figure 9). A turbine hall of rectangular section, that is with a wall along line Z-Z in Figure 9, was already in use when the predictive work was done, from which one could obtain boundary and calibration data. The purpose of the work was to see whether the alteration of building configuration was likely to cause high air velocities and an associated uncomfortable working environment within the turbine hall.

By using a two-dimensional format, the turbine hall was modelled as though its characteristics were independent of one of the horizontal co-ordinates, and vents were used to simulate the flow around the turbines in between the ductwork and other obstructions.

A number of CAFE runs were carried out and the results indicated that air velocities within the long extension would be uncomfortably high. A partition wall 'A' was then introduced in the mathematical model which reduced the velocities at working levels to under 0.3 m/s at a temperature of about 20°C. This wall is now under construction within the turbine hall and it is hoped that some full-scale measurements will be taken, when it is completed, that will verify the predictions.

6 DISCUSSION AND CONCLUSIONS

The four examples given in this paper have demonstrated the way in which computational analysis can be used to assist the traditional hand calculation methods for the design of certain types of ventilation conditions in industrial buildings. Hence, a computer program, such as CAFE does not replace the BS codes and CIBS and IHVE guidelines, etc; what it does is to provide an opportunity for more detailed assessment of air movements, temperatures and gas concentrations within the enclosed space for situation in which this information may need to be known to greater accuracy. The standard code and guideline calculations can only give bulk average values but these may be used to provide input data for CAFE; as for example in the case of the stack of coal in the covered stockyard, where natural ventilation flow rates were estimated using the IHVE guidelines.

It should be noted that all the above studies of ventilation were carried out using a two-dimensional computational analysis. This was an adequate procedure to adopt for the type of information required in each case, provided great care is taken in the interpretation of the results. If air flow movements etc. are required under circumstances in which three-dimensional effects cannot be ignored, then a fully three-dimensional computational analysis becomes necessary. The alternative would be model tests or full-scale measurements, each of which are subject to their own difficulties, as explained in the paper. New numerical techniques and more powerful and cost effective computing facilities are making three-dimensional computations of complex turbulent air flows in buildings a real possibility. However, considerable caution must be exercised in their use as the quality of the results still depend on the coarseness of the finite difference (or finite element) mesh employed and in the skill of the user in applying it, as well as on the validity of the turbulence model adopted in order to close the time-averaged Navier-Stokes equations.

The main conclusion to emerge from these and other similar studies carried out by the authors for clients in industry is that the existing hand calculation methods for determining forced and natural ventilation flows in buildings are inadequate for cases where fairly accurate determinations of air flows, temperatures and gas concentrations are required. Model tests and full-scale measurements can be extremely valuable in these situations but it is clear that computational methods have an increasingly important role to play in the design of safe ventilation systems for industrial buildings.

REFERENCES

(1) HADLEY J. Energy Conservation and Indoor Air Quality. ASHRAE Journal, Mar. 1981, Vol. 23.

(2) LEWIS A. Sweden's Suffocating Success. Consulting Engineer, Nov. 1981, Vol. 45, p. 45.

(3) MARSH R.C. and THOMAS B.Q. Energy Savings in Laminar Flow Clear Air System. Solid State Technology, July 1981, p. 83.

(4) TAYLOR B.A. Natural Ventilation and the PSA Estate. CIBS, Dec. 1980, p. 55.

(5) BRITISH STANDARD.Code of Practice for Mechanical Ventilation and Air Conditioning in Buildings. BS 5720, 1979.

(6) BRITISH STANDARD. Code of Practice for Ventilation Principles and Designing for Natural Ventilation, BS 5925, 1980.

(7) CIBS Guides/formally IHVE Design Guides. Various, in particular: Section A4, Air Infiltration.

(8) PLATE E.J. and LOESER J. Wind-tunnel Evaluation of Ventilation Problems. J. of Wind Eng. and Ind. Aero., 81, Vol. 8, Nos. 1&2, p. 123.

(9) EUSER H., HOOGENDOORN C.J. and OOIJEN H. Van. Airflow in a Room as Induced by Natural Convection Streams. Int. Conf. on heat and Mass Transfer in Buildings, Dubroonic, 1977.

(10) ASHRAE. Bibliography on Available Computer Programs in the General Area of Heating, Refrigerating, Air Condition and Ventilation. ASHRAE Research Project, 1975, GRP-153.

(11) AIC Survey. 1981 Survey of Current Research into Air Infiltration in Buildings. Dec. 1981, AIC TN 7.

(12) PATANKAR S.V. Numerical Heat Transfer and Fluid Flow. McGraw-Hill, 1980.

(13) JONES W.P. Laminarisation in Strongly Accelerated Boundary Layers. PhD Thesis, 1971, University of London.

(14) MOULT A and DEAN R.B. CAFE - A Computer Program to Calculate the Flow Environment. CAD 80, 4th International Conference on Computers in Design Engineering, Brighton, 1980.

Table 1 Summary of the equations solved (Cartesian co-ordinate system)

Equation	ϕ	Γ_ϕ	S_ϕ
Continuity	1	0	0
x-momentum	u	μ	$-\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}(\mu\frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(\mu\frac{\partial v}{\partial x}) + \frac{\partial}{\partial z}(\mu\frac{\partial w}{\partial z})$
y-momentum	v	μ	$-\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}(\mu\frac{\partial u}{\partial y}) + \frac{\partial}{\partial y}(\mu\frac{\partial v}{\partial y}) + \frac{\partial}{\partial z}(\mu\frac{\partial w}{\partial y})$
z-momentum	w	μ	$-\frac{\partial p}{\partial z} + \frac{\partial}{\partial x}(\mu\frac{\partial u}{\partial z}) + \frac{\partial}{\partial y}(\mu\frac{\partial v}{\partial z}) + \frac{\partial}{\partial z}(\mu\frac{\partial w}{\partial z})$ + body forces (buoyancy, etc.)
Turbulence energy	k	$\mu_{eff}/\sigma_{k,eff}$	$G_k - \rho\varepsilon$
Energy dissipation rate	ε	$\mu_{eff}/\sigma_{\varepsilon,eff}$	$(C_1G_k - C_2\rho\varepsilon)\varepsilon/k$
Stagnation enthalpy	$\bar{h}$	μ/σ_h	0 (assuming no radiation)
Concentration	m_j	μ/σ_j	0

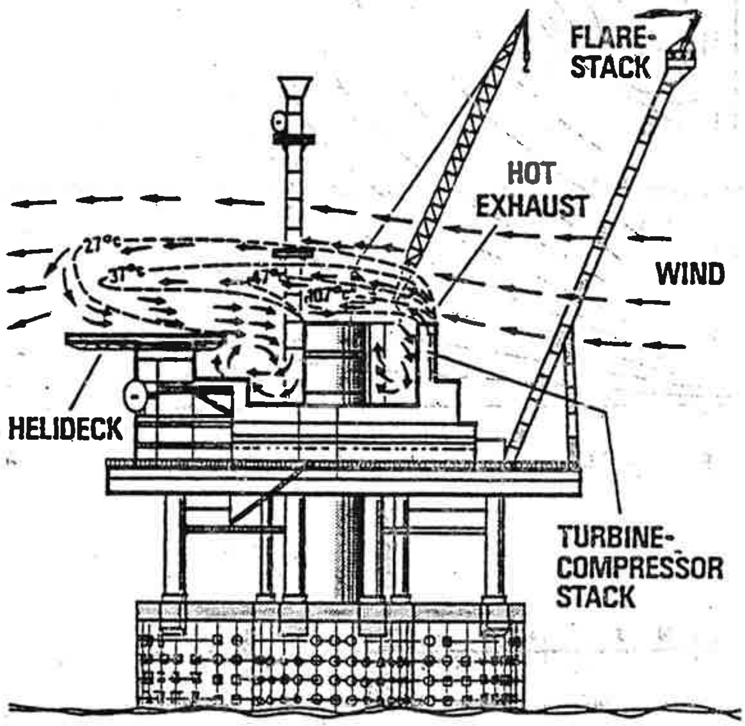

Fig 1 Typical offshore platform showing the flow characteristics over the topside and the mixing of hot turbine exhaust gas near HVAC inlets

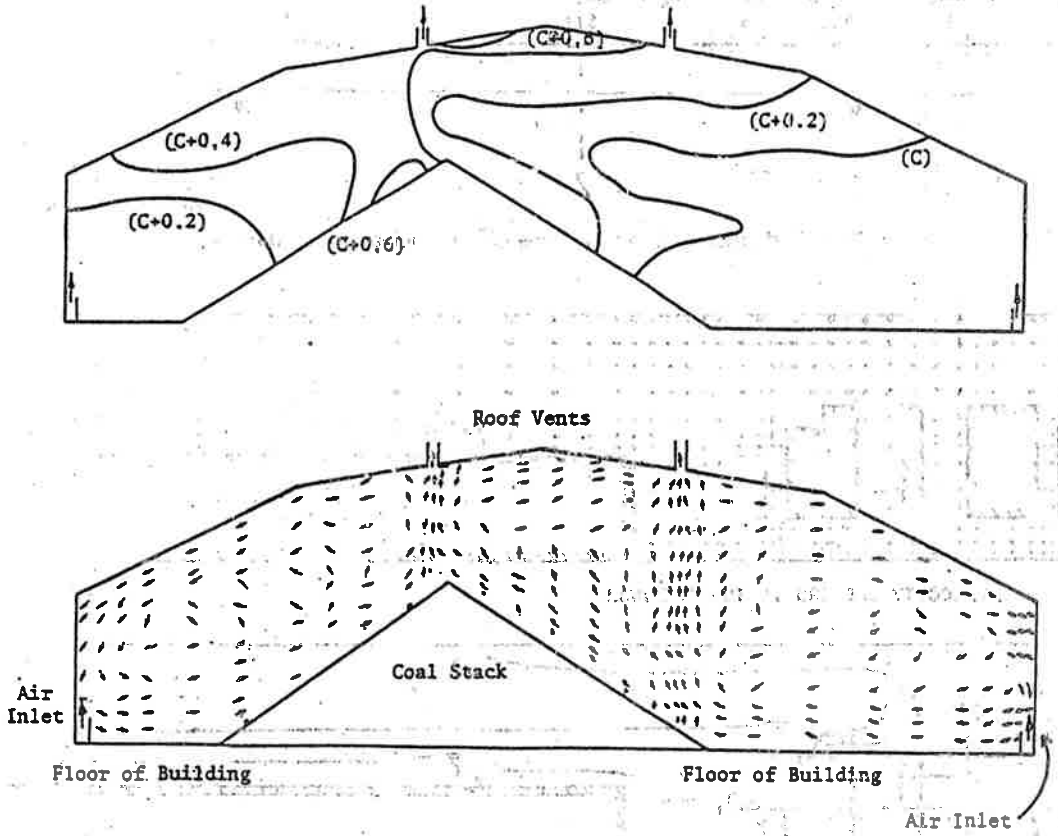

Fig 2 Computational analysis of methane release and air movement in the covered stockyard building at Selby

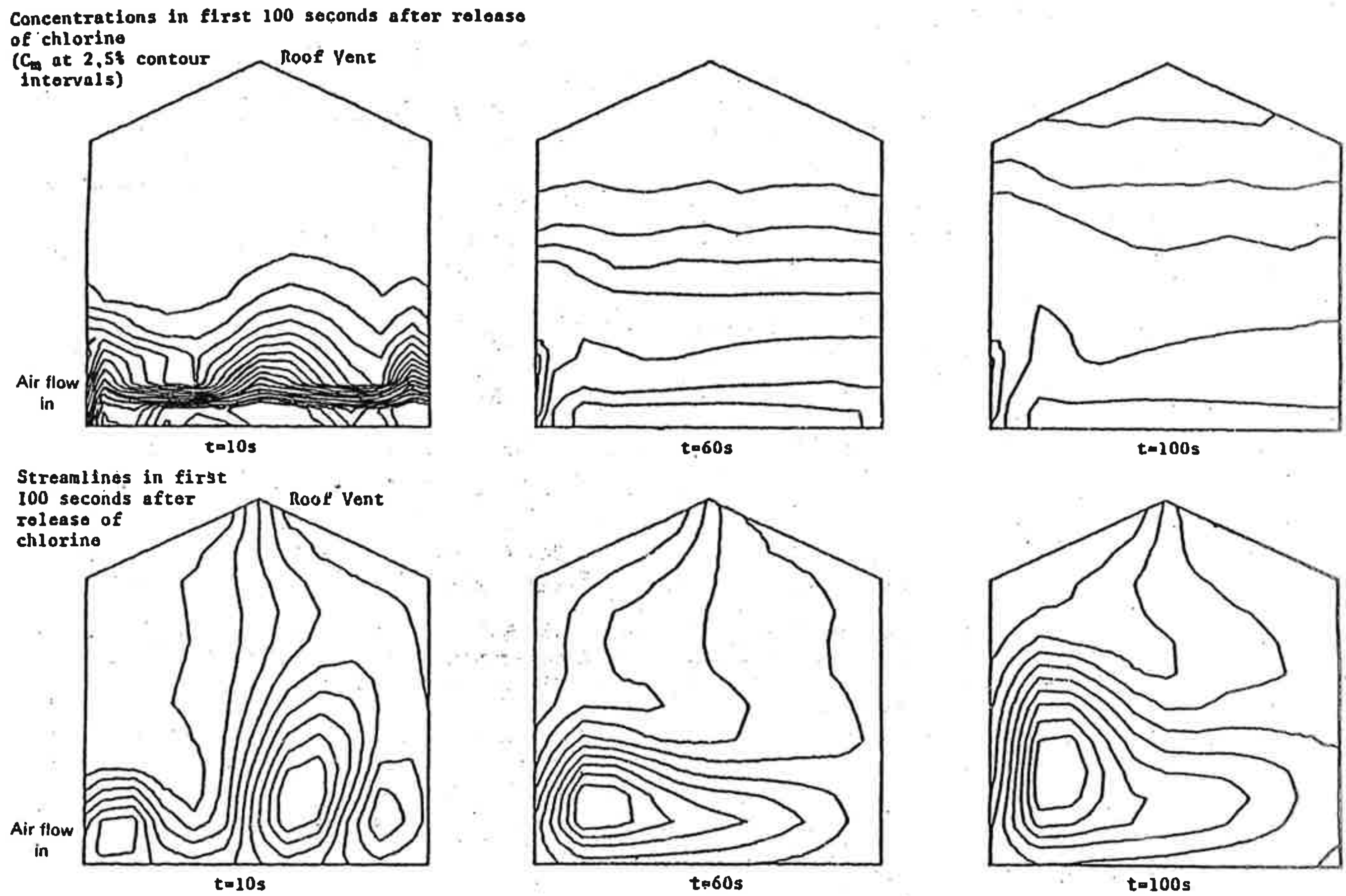

Fig 3 Mixing of chlorine in process building at chlor-alkali plant

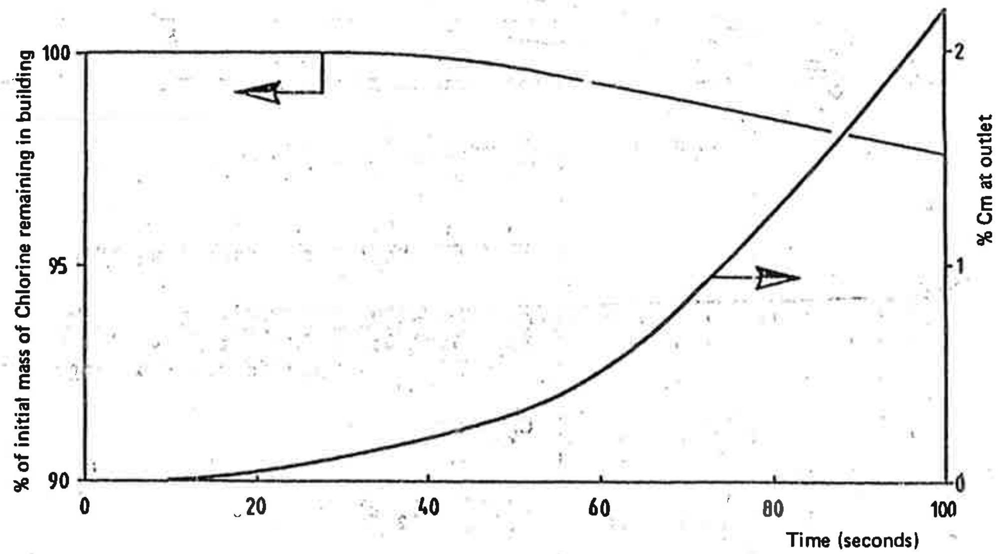

Fig 4 Characteristics of chlorine escape from process building

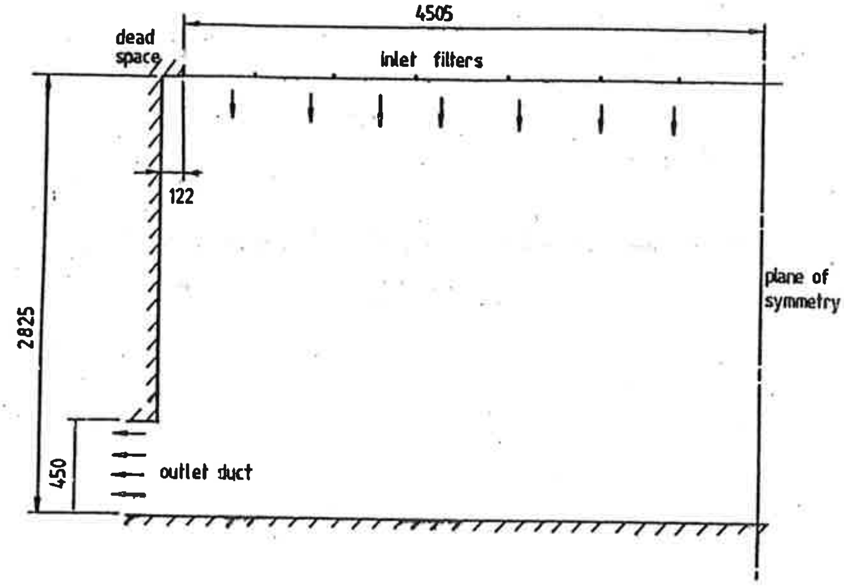

Fig 5 Geometry of clean room (baseline design, not to scale and all dimensions are in mm)

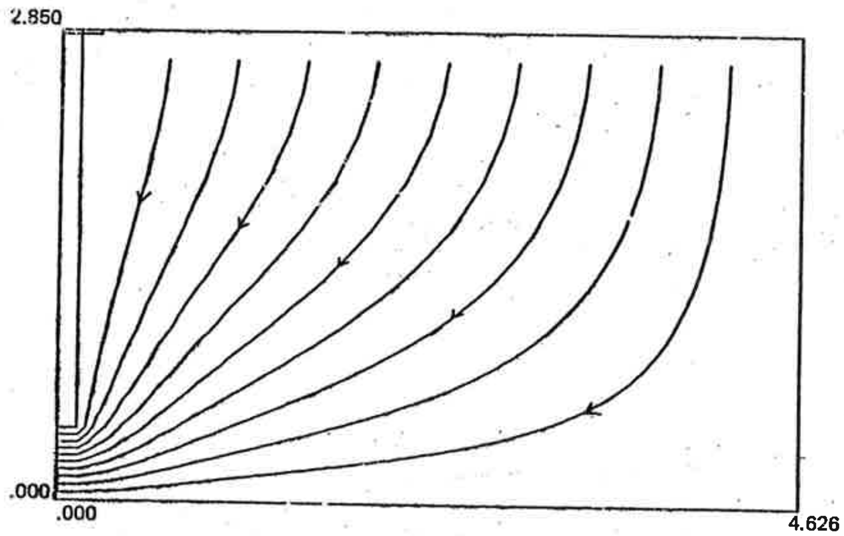

Fig 6 Predictions of streamlines in clean room

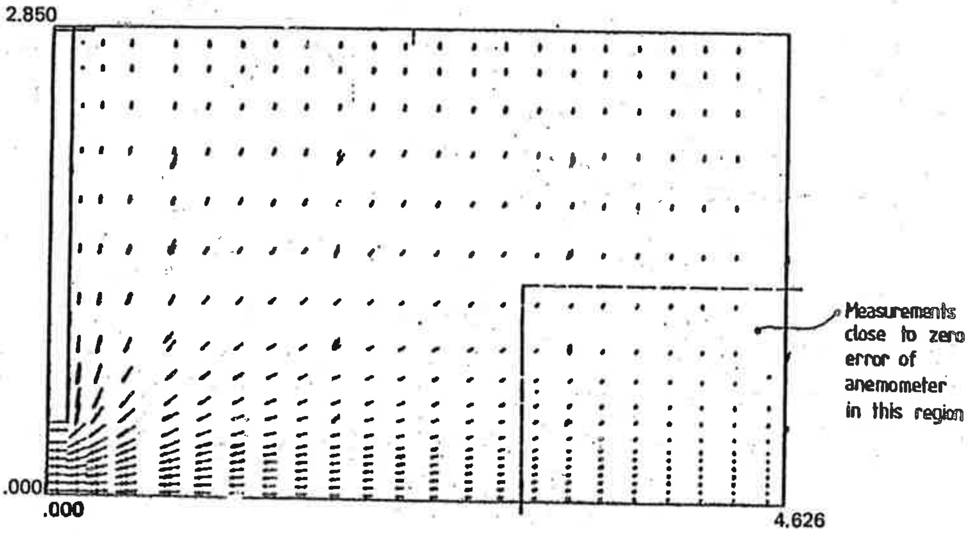

Fig 7 Predictions of velocity in clean room and comparison with measured velocities

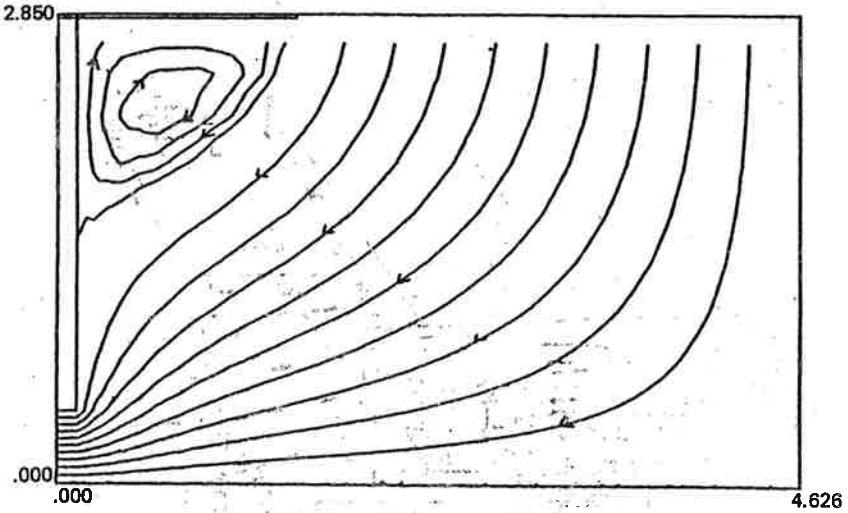

Fig 8 Predictions of streamlines with two filter panels removed

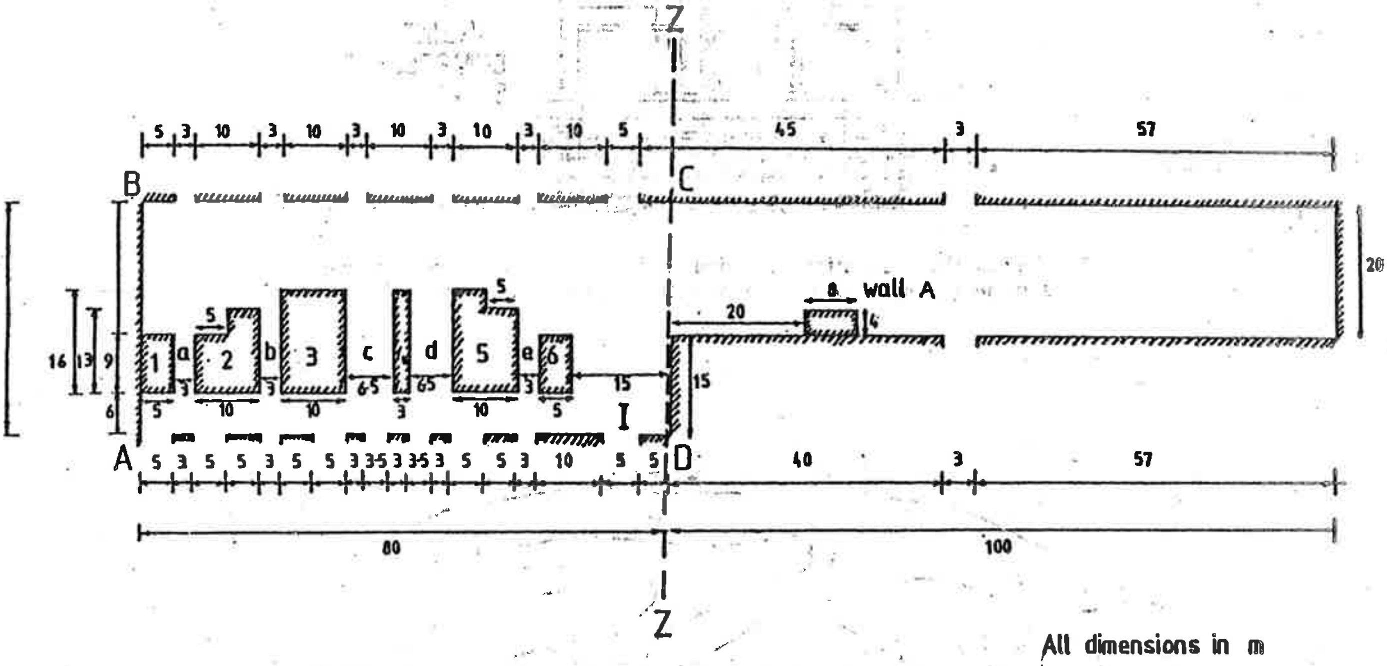

Fig 9 Idealised cross-section of turbine hall of nuclear power station

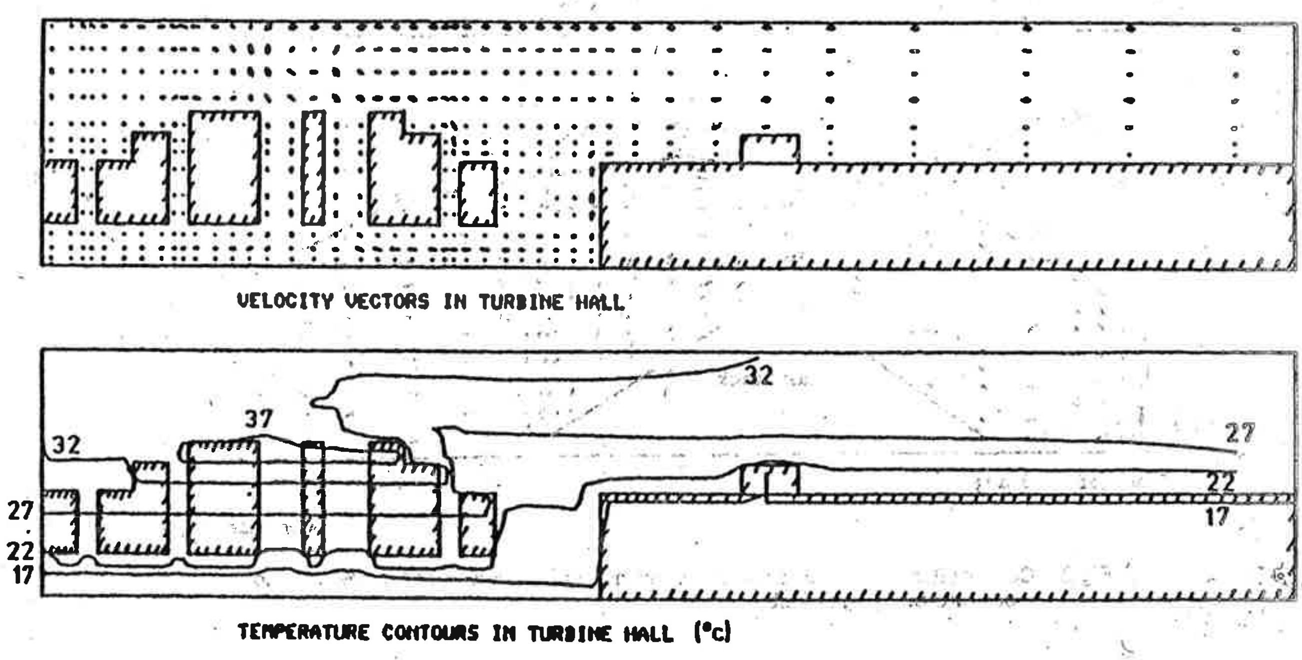

Fig 10 Predictions of velocity vectors and temperature contours in turbine hall