#4146

ROOM
-VENT
'90

Room ventilation simulations

B.A. Børresen
T.H. Toften
Techno Consult AS
Sandvika, Norway

D1

52

ROOM VENTILATION SIMULATIONS

B.A. Borresen, T.H. Toften
Techno Consult AS
Sandvika, Norway

SUMMARY

New and challenging concepts of ventilation are a result of more advanced numerical simulations. Improvements mean better and often simpler ventilation systems. The environment and safety is improved and the result is usually lower installation, running and maintenance costs.

This paper summarises some of the fields where we are involved. Our emphasis is lain on showing the need for a variete of numerical simulation tools - from specially designed programs to more generally and advanced CFD-programs (Computerised Fluid Dynamics).

The presentation starts with dynamic responses in solar loaded ceiling girders and floor conditioning in large glazed spaces, goes via ventilation of offshore modules and ends with air flow simulations of impuls fans mounted in the ceiling of vehicle tunnels.

Introduction

New and challenging concepts of ventilation are a result of more advanced numerical simulations. Improvements mean better and often simpler ventilation systems. The environment and safety is improved and the result is usually lower installation, running and maintenance costs.

This paper summarises some of the fields where we are involved. Our emphasis is lain on showing the need for a variete of numerical simulation tools - from specially designed programs to more generally and advanced CFD-programs (Computerised Fluid Dynamics).

The presentation starts with dynamic responses in solar loaded ceiling girders and floor conditioning in large glazed spaces, goes via ventilation of offshore modules and ends with air flow simulations of impuls fans mounted in the ceiling of vehicle tunnels.

Bibliotheca Alexandrina

To illustrate the implications of room venting in large glazed spaces, some aspects of Bibliotheca Alexandrina in Egypt are chosen. The Norwegian architect company, Snøhetta, won the UNESCO sponsored international design contest among 1300 competitors.

The 70,000 m2 library building has a circular shape with a diameter of 160 m, as shown in figure 1. The roof is 16° sloped toward North-North-West and is partly glazed through sections of 14.4 X 9.6 m2.

The roof is partly below the ground level. The inside open library space of about 12,000 m2 is designed in terraces up a slope of 16°. The free height to the ceiling is therefor between 15 and 20 m. Figure 2 shows the staged floor and the ceiling. Integrated into the building are also closed access library rooms for book storages, auditoriums, laboratories, workshops and underground parking.

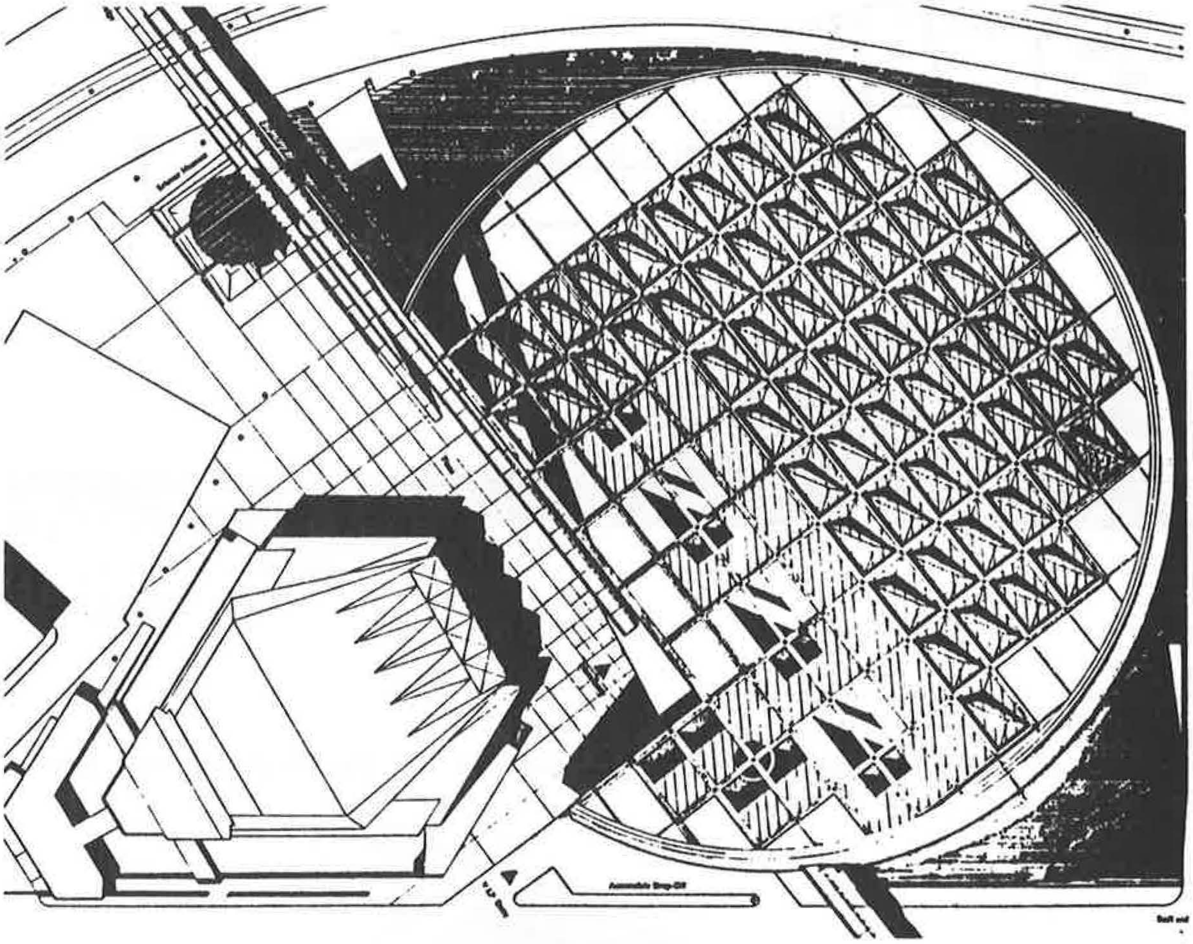

Figure 1 Bibliotheca Alexandrina has a 160 m diameter ground floor area. Its sloping appearance resembles the rising sun.

The roof baffle system.

Each section of the ceiling has a glazed triangle and baffles are placed in triangles below the glazing as shown in figure 3. The baffles reduce direct sunlight into the open library space, and give indirect solar lighting via the ceiling. The baffles are designed to give best possible view to the outside.

The surface temperatures of and the temperature differences within the ceiling girders in the roof sections are important to find. This due to thermal stresses bending and elongating the beams. Radiant heat flux from the ceiling surfaces will also result in higher temperatures at the floor level.

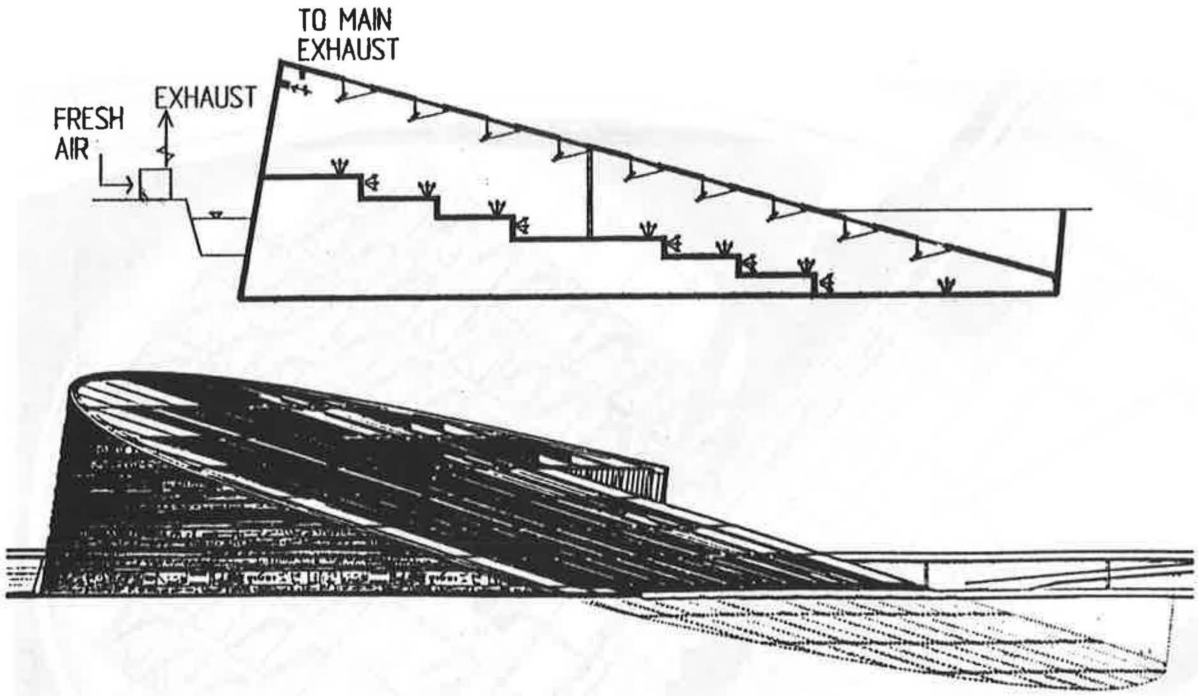

Figure 2. The main library space.
The staged floor and the ventilation principle for the main library space.

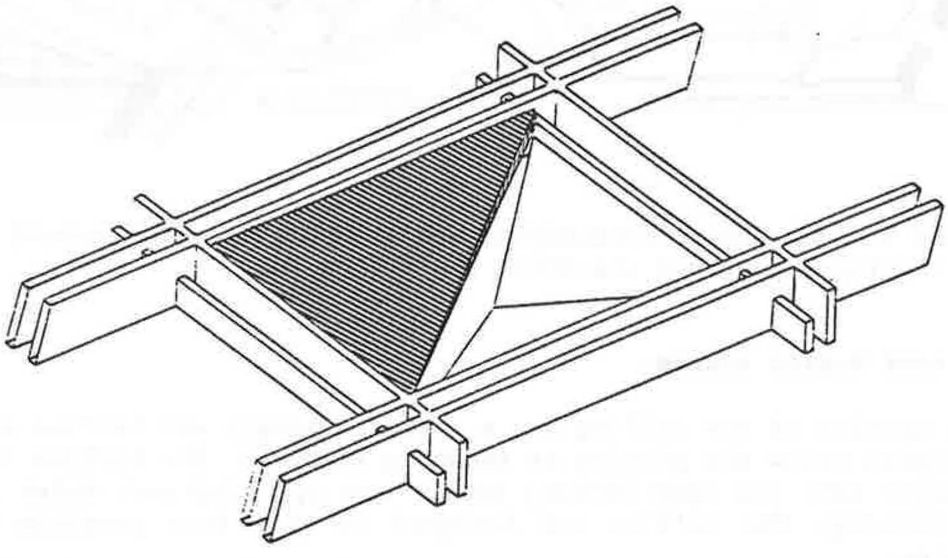

Figure 3. Air plenum seen from above.
The ceiling are based on the 9.6 m x 14.4 m2 library modules. The roof window form a triangle and has an enclosed ventilated battle system (air plenum) under neath.

Dynamic solar data, combined with special window glazing and daily outdoor air temperature variations are important parameters. In addition the thermal characteristics of the massive beams and the insulated baffles with low mass have to be taken into consideration.

The rather complicated and large simulation task was done very efficiently by use of the numerical program ROYAL ELISE (Borresen & Harsem, 1990). The program is a further development of ROYAL DEBAC built at the Norwegian Institute of Technology in the early 1980's.

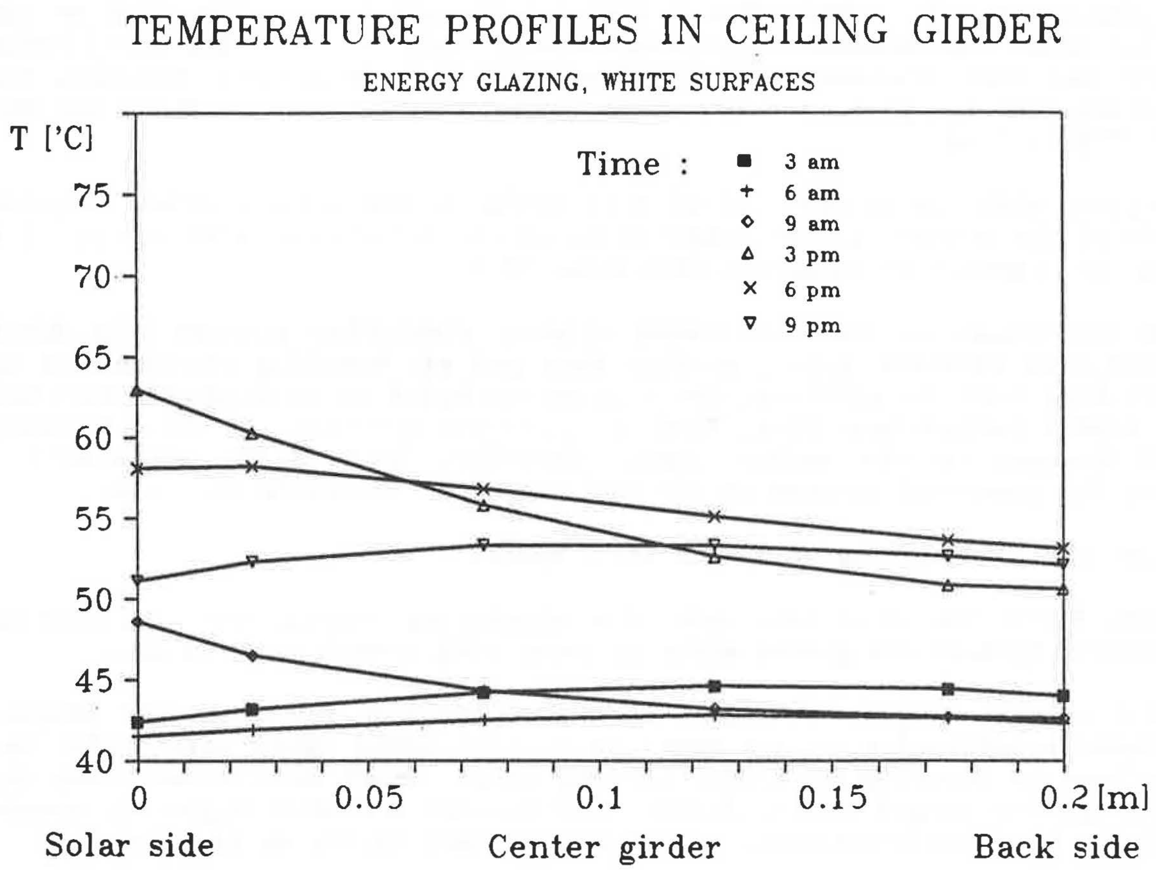

Figure 4 shows the daily temperature variations within one of the sun exposed ceiling girders.

ROYAL ELISE is a dynamic building air conditioning simulation program. Procedures for hour-by-hour solar loads, internal loads and outdoor temperature are integrated with an air handler model. The room model is generally built up in matrices that gives it a many-sided usefulness, which goes far beyond traditional buildings. Special attention has in these simulations been paid to include the different angle factors for radiant heat exchange. The simulation procedure is based on matrix operations and non-linear systems may be simulated by using a set of up to 4 linear system matrices.

The results of the 24 hour steady state cycle simulations with one of the girder sides sun-exposed in the roof section is shown in figure 4. The windows consist of a low energy glazing which gives little reduction in daylighting. The results obtained when all baffle and girder surfaces are kept white, show a temperature variation within 20 K on the solar side. The maximum temperature reaches almost 65°C. The ventilation rate was kept constant at 0.4 m3/s. The air temperature reaching the ceiling zone was given a daily variation from 24°C and reached its maximum of 33°C at 3 pm.

Compared with the use of typical grey surfaces, the white surface solution reduced the maximum girder temperature at the solar side with nearly 15 K and the temperature variation with about 10 K.

The conclusion is that an indoor climate simulation program like ROYAL ELISE with climatic files, weather data and air handling procedure is an efficient tool. In addition, the program has build in an advanced procedure to handle radiant heat flux. This is not included in most of the more heavy CFD-programs on the market today. Therefor, ROYAL ELISE represents a powerful numerical program in micro-climatic and structure analysis.

Local air conditioning of staged floor spaces.

ROYAL ELISE has also been used for simulating displacement ventilation effects. Typical are glazed atria or rooms with considerable height.

Local conditioning at the floor level is important for books and people. Reduced conditioning of the upper space will often imply significant reductions in installation costs, running costs, total ventilation rates and ducting. The staged floor, however, introduces a higher degree of uncertainty. Air flow simulations are often necessary in the design phase.

CFD-programs are turning out to be an alternative to laboratory models. However, they require considerable computer facilities and running time. To do such analysis we are running the "classic" CFD-program PHOENICS from CHAM/UK.

A 9 m section of the staged floor in the library space was simulated with PHOENICS. The height of the 2-dimensional domain is 6 m. From earlier simulation with ROYAL ELISE the temperatures of the floor was found to be about 25°C. Placed at the left and right edge are book-shelves of 2 m height and the same surface temperature as the floor. Air inlets are placed at 3 inner corners of the floor. Vertical airflow rate is 150 m3/h through each inlet. The inlet air temperature is 21°C.

The K-E model was applied for calculation of the turbulence. The coefficients in the K-E model can be adjusted to optimise simulations of temperature layers in rooms with "calm" conditions for the air-flow.

The upper part of the domain has "free" surfaces. I.e. the boundary conditions were set so that all values outside the domain had the same quantities as in the first cell into the domain. In PHOENICS this means use of the command "SAME".

The boundary conditions for all the solid surfaces were set as follows :

- Non-slip conditions at the walls (floor) was applied by use of a logarithmic law.
- Turbulence kinetic energy and its dissipation rate are fixed at the walls.
- The surface temperature was fixed as well, and an overall heat transfer coefficient of 10 W/K per m2 surface area was chosen.

The gas density was calculated using the ideal gas law. The domain pressure differences were neglected during these calculations.

The variables solved had to be strongly under-relaxed to get stability. Convergence was reached after 490 iterations and the residuals were 1.6E-4 for the pressure and 5.1E-5 for the temperature.

The results shown in figure 5 for the air velocities indicate convective vertical flow along all the vertical warm surfaces. The cooled air slides horizontally out to the edges of the terraces. The air-flow velocities are low. With a careful design the preliminary results indicate that there is no need to build up walls at the edges to prevent cold air falling from one level down to the next. However, this has to be analysed more carefully in the predesign and the design phase.

Temperature contures have been plotted for the domain in figure 6. The result indicates a well conditioned temperature environment at all the floor levels. The air temperature 1 meter above the floor is about 23°C, and the gradient between floor level and 1 m above the floor is typical 1-2 K.

Fire and safety aspects have also been looked into. The smoke control design follows the guidelines given in Borresen & Madsen 1990.

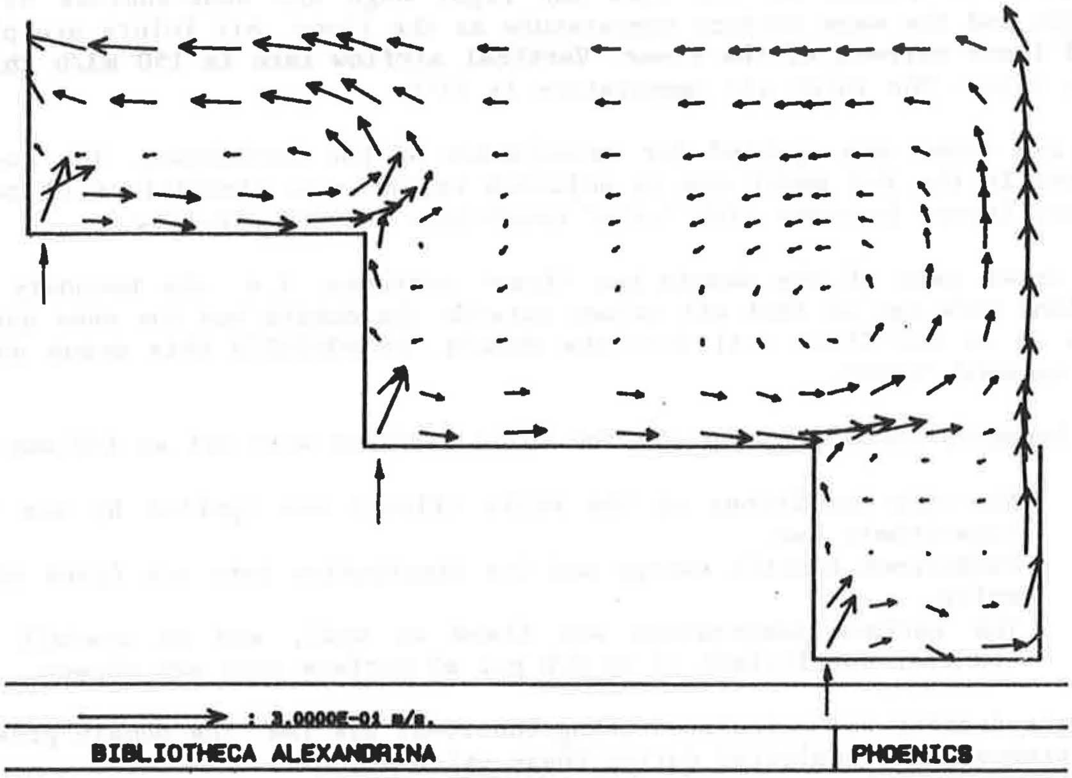

Figure 5. Velocity vectors in a local ventilated section of the bibliotheca.

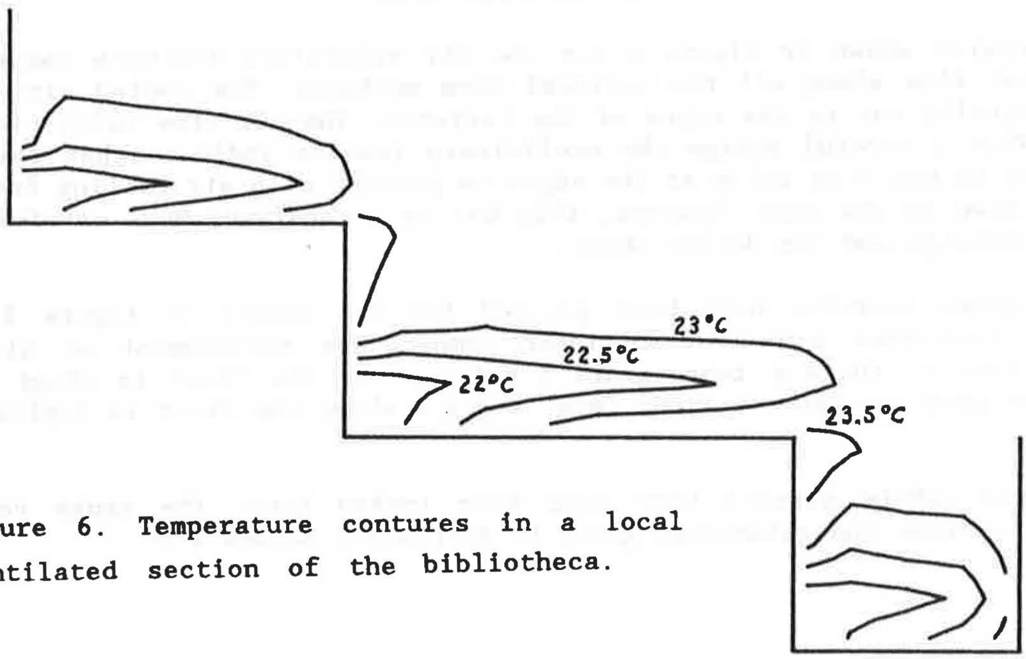

Figure 6. Temperature contures in a local ventilated section of the bibliotheca.

The conclusion is that PHOENICS is a powerful CFD-program that can be used in the design phase to improve the ventilation system. High costs and poorly designed ventilation solutions can be avoided. CFD-calculations are also beginning to reach an economic level where they can be taken into more common use as a HVAC design tool.

Offshore modules

The process areas of offshore oil platforms are sectioned in modules. These may be half open to provide for the often used natural ventilation.

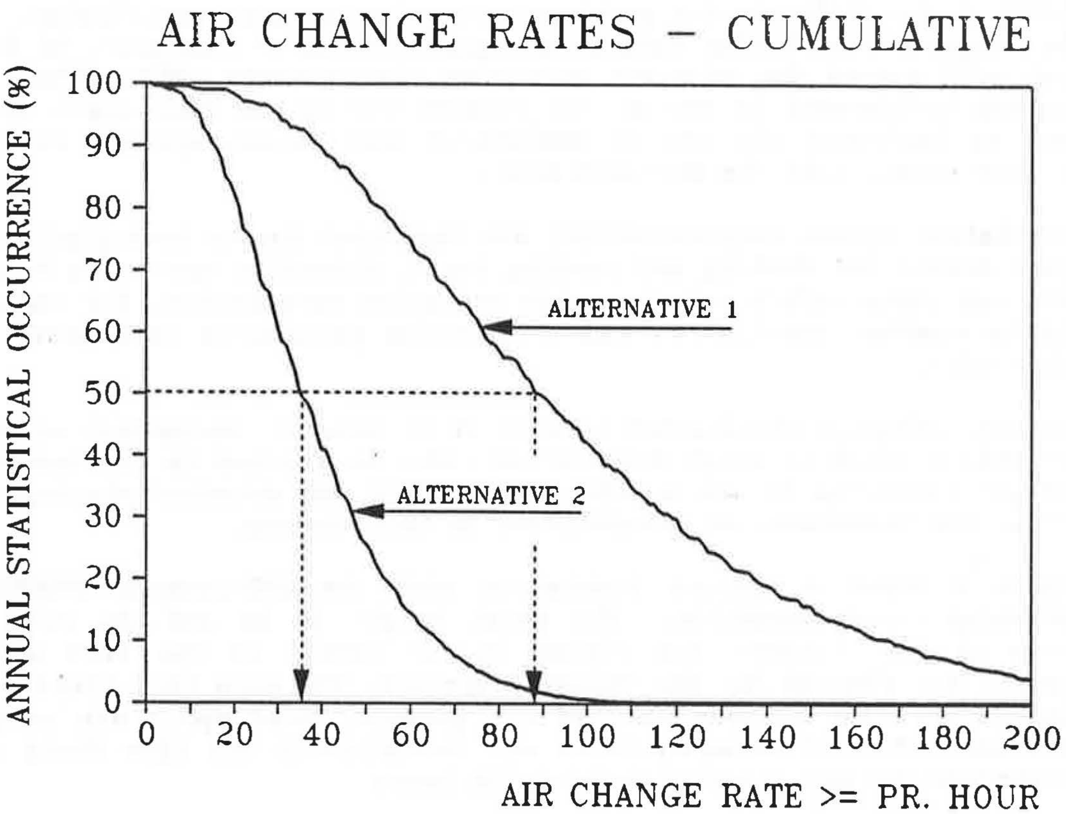

Figure 7. The occurrence of different air change rates over a statistical Statfjord area year.

The opening areas have to meet the requirements of an "adequate" ventilation rate, say 95% of the year. To ensure that these are met, simulations taking the variating wind speed and wind directions have to be taken into account. Of course, there is also a large variety of options regarding placing of weather openings, i.e. towards which directions, how to split them into floor level and ceiling level openings etc. Probabilities for gas leakages, gas leakage rates, internal heat loads and working environment are parameters.

A typical process area design will require about 5,000 to 10,000 single case simulations. For Techno Consult this has lead to development of a simplified, but highly efficient natural ventilation simulation program, TC-ROI SOLEIL. TC-ROI SOLEIL integrates necessary weather data, and works out, for instance, statistical air flow rate duration curves. Figure 7 presents typical results.

Figure 7 also reflects the problems related to natural ventilation. I.e. if the adequate ventilation rate is designed for 95% of the year, 50 % of the year will exceed the adequate rate more then 5 times. This effect on the working environment is one of the reasons why it now once again is important to implement the use of mechanical ventilation systems in critical weather areas, like the Northern seas.

Ventilation system simplifications are important due to installation costs, space demand for ducting and running costs. Attention must also be paid on fire and smoke safety together with explosion suppression. For instance we try to combine ventilation and preinerting procedures with inert gas or water mist.

However, adequate ventilation whether it is natural, mechanical or mixed is not only related to total ventilation rate. It relates to the internal air and gas movements in the modules. Dead spots and detector placing are key words. The importance of CFD-programs is then obvious.

Figure 8 shows a typical simulation with the CFD-program PHOENICS for analysing these conditions. The total height is 6m and the half length shown is 20m. Louvers are placed in two levels at the left wall. The results are plotted for the velocity vectors. The main flow field causes a circulating flow between the ceiling girders. Dead spots are avoided in this case. Production equip-ments are obstacles in the flow field and can accelerate the air velocity between the beams.

Gas leakage simulations has also been done by use of CFD programs. Based on field tests and 3D simulations we have been able to make significant reductions in platform installations.

However, more research is needed to improve the simulation procedures. Our company is supporting a PhD study on measuring turbulence in offshore modules, at Hatfield Polytechnic UK. One of our own employees is also working on his Dr.ing.-degree at the Norwegian Institute of Technology.

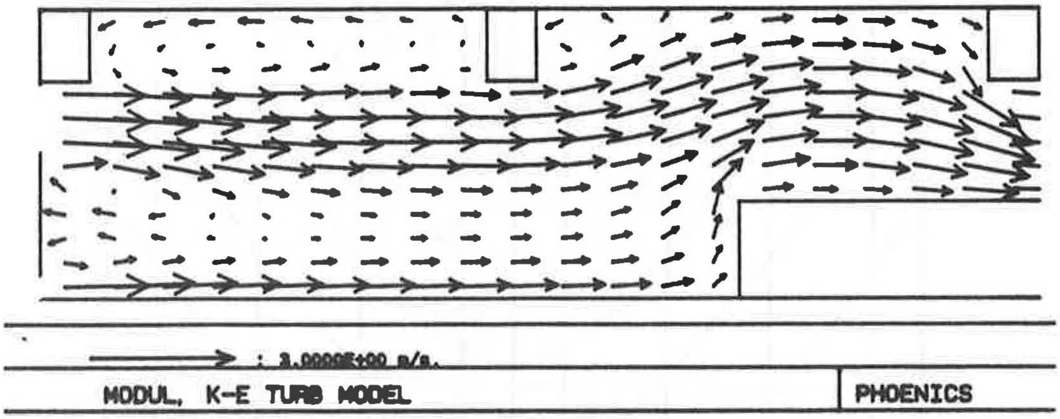

Figure 8. Typical internal flow in a platform module, simulated with Phoenics. The wind forced ventilation enters through openings lowers at the left site.

Tunnel ventilation

Adequate ventilation of enclosed spaces will cover long and narrow rooms, like corridors, mines and vehicle tunnels. These are areas where both simplified and in advanced approaches are necessary. As an example we will focus on vehicle tunnels.

Running costs in the enormous road tunnel systems, built in Norway, are mainly connected to the electrical consumption of tunnel fans. Typical lengths for fjord crossing, mountain traverses or city communications are 2-4 km. Longitudinal ventilation driven by impuls fans placed directly in the tunnel ceiling is dominating. However, significant electric energy and load reductions often can be obtained by introducing shafts, due to the improvement of fan efficiency.

As a flow pattern problem a closer analysis of the impuls fan effect is of interest. To do such analysis we have simulated the flow in tunnels with PHOENICS.

As a test case the simulations were done 3D within a squared tunnel. The Z-direction is the longitudinal direction with a total length of 180 m. The height in the Y-direction is 6 m, and the total width is 8 m. Two impuls fans with an outlet area of 1 m2 was mounted 1 m from the ceiling to the centre. The impuls fans where placed at Z = 20 m and Z = 100 m.

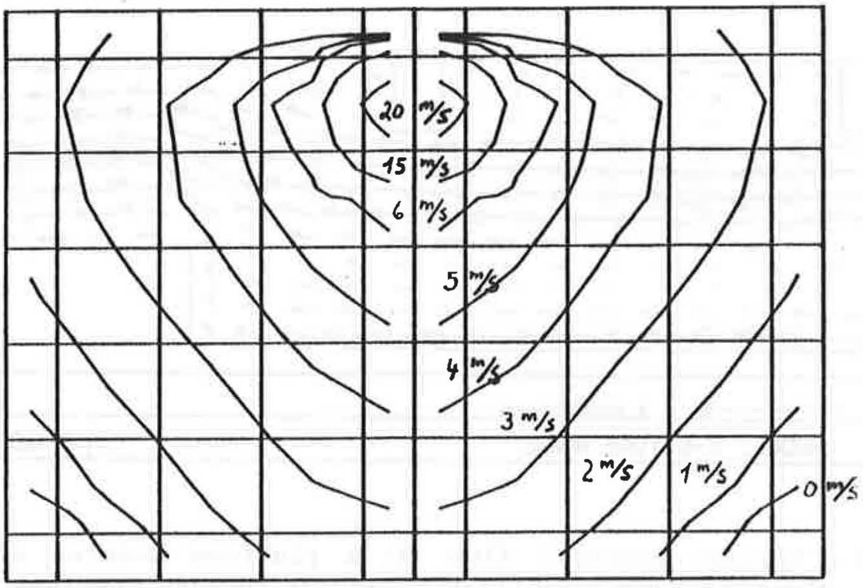

Figure 9. Contures of velocities in the z-direction at a cross-section 2 m downstreams the jet fan.

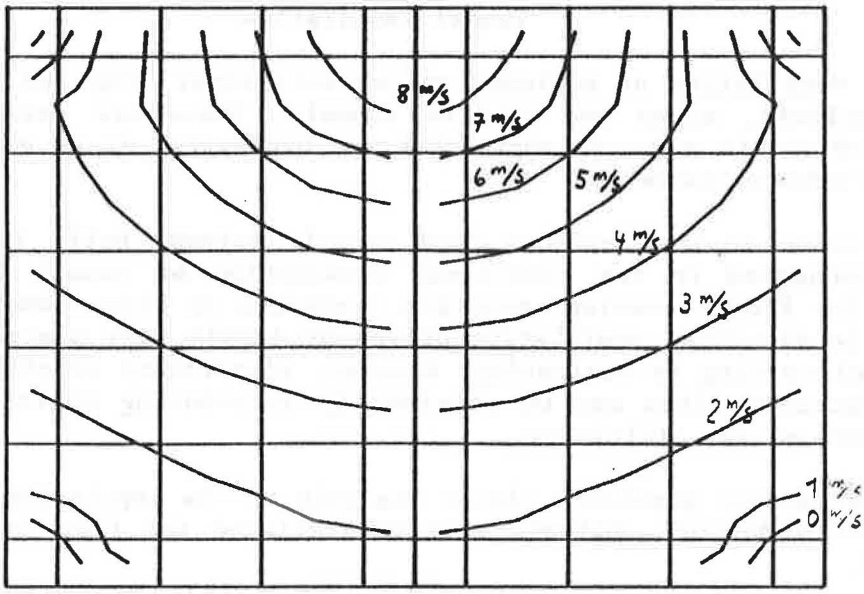

Figure 10. Contures of velocities in the z-direction at a cross-section 45 m downstreams the jet fan.

To maintain pressure loss in the tunnel, usually caused by vehicles, the outlet was partly blocked. Non-slip conditions along the walls were used. Turbulent quantities were taken care off by the KE-model. Outlet velocity from the impuls fan was set to 25 m/s.

The contures for velocity in the Z-direction are shown for the XY sections in figure 9 and 10, at a distance downstreams after the first impuls fan of 2 m and 45 m respectively. The contures are marked for the velocities in m/s. These plots give a lot of information. The coanda-effect for the jet is obtained for a long distance downstreams as shown in figure 10. This is one of the reasons why impuls fans are more expensive in use when the ceiling are rough.

Although the average velocity in the tunnel is about 4 m/s, figure 11 shows that the velocities at the center-axis are considerable higher.

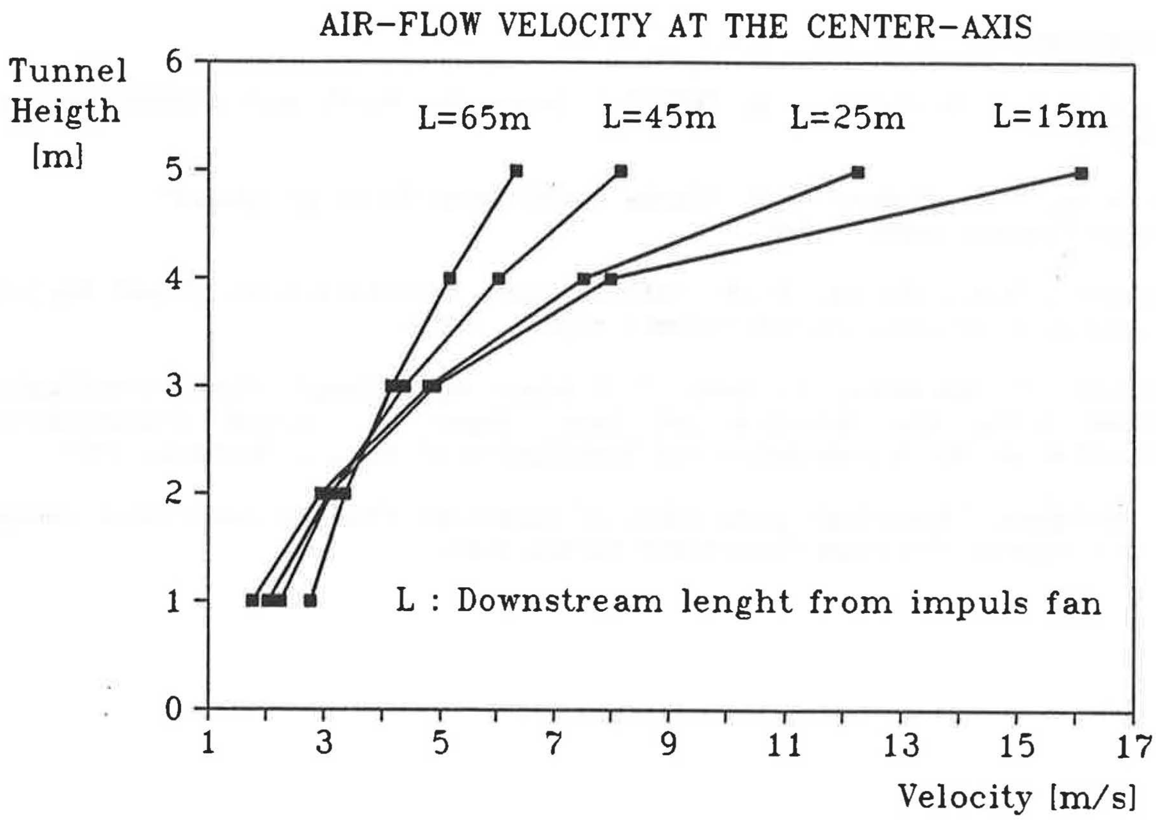

Figure 11. Airflow velocity in the z-direction at the center-axis for different downstream length from a jet fan.

In the corners at the floor level, vortexes are detected. In these zones dust and particles will accumulate. Known from the laboratories, the corners in wind-tunnels are often blocked to avoid unwanted production of vortexes.

The importance of CFD-simulations of ventilation in tunnels are related to improving fan installations. I.e. how fan efficiency is affected by :

- tilting of fans in one way traffic tunnels
- parallel fan installations
- distance between fans
- distance between fans and ceiling
- roughness of tunnel surfaces

These are some of the challenges we will investigate in the future.

References:

Holgate, J.A, Børresen, B.A. "HVAC" to brave the North sea" ASHRAE Journal, April 87.

Børresen, B.A., Madsen, C.N. "Smoke ventilation in large spaces" Techno Consult report 1990

Børresen, B.A., Harsem, T.TH. "Displacement Ventilation in Glazed Atria". Harsem Prosjektering/Techno Consult report, 1990.

H.Ohasi, T. Kuwamura, T. Baba, " A study on a longitudinal ventilation system using the enlarged jet fans. Paper D2, Second International Symposium on the Aerodynamics and Ventilation of Vehicle Tunnels, 1976.

L. Davidson, "Numerical simulation of turbulent flow in ventilated rooms. Ph.D.- thesis, Chalmers University Sweden 1989.