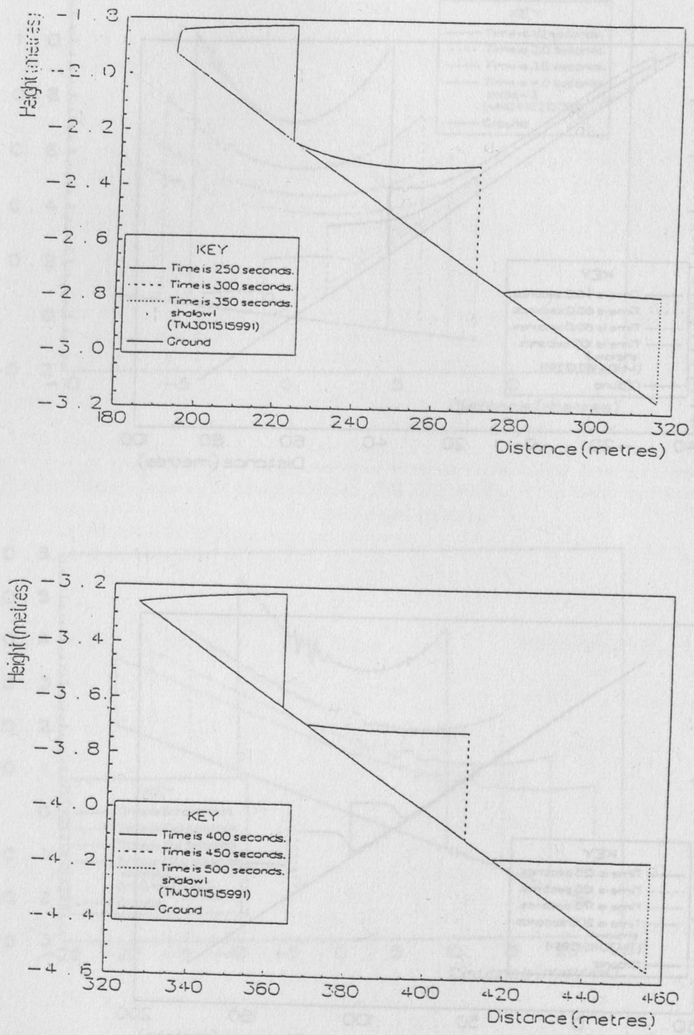


Figures 7e,f: Continuing the flow from Figures 7a–d: in the final regime the hydraulic jump collapses and the cloud reaches the wedge shape which is moving down the hill but no longer spreading.

COMPUTER SIMULATIONS OF WIND AND VENTILATION AID PLATFORM DESIGN AND SAFETY

A. Ronold
DNV Technica, P.O. Box 300, 1322 Høvik, Norway

Computer simulations of wind and ventilation provide a cost efficient tool in the fields of area classification, smoke and gas dispersion and wind-related platform design aspects such as positioning of turbine exhaust and air intakes.

Adequate natural ventilation in petroleum process plants is of crucial importance when classification of hazardous areas is considered. The term "adequate" is most often referring to either the number of air changes in a given area or to the frequency of occurrence of wind speeds less than a given value, typically 0.5 m/s or 2 m/s. In an attempt to specify these parameters accurately, and to aid in the assessment of area classification as well as platform design, computational fluid dynamics (CFD) can be used to simulate the wind flow field and gas or smoke dispersion around and within process plants.

For a given installation, wind simulations are typically carried out for three different wind speeds and eight wind directions. Combined with information from the wind rose for the actual site, these simulations yield the frequency of occurrence of number of air changes on all areas of the installation considered, as well as air velocity distributions. Examples are presented from offshore installations in the North Sea.

Given the wind flow field and air velocity distributions for a certain installation, gas or smoke dispersion simulations can be performed, for example to consider the likelihood of having ignitable gas-air concentrations into non-hazardous areas of the plant in the case of a major gas release. Examples on calculated gas concentration profiles for certain release scenarios are presented.

Keywords: Ventilation, Area Classification, Gas Dispersion, Wind

INTRODUCTION

Wind and ventilation play an important role in many aspects of offshore technology and a thorough understanding of the air flow behaviour is essential in decision making in platform design. This knowledge can be obtained through experiments, but prototype measurements are often impossible for cost and safety reasons, and model experiments such as wind tunnel tests are also expensive and time consuming - and may suffer from scaling problems. Alternatively, numerical simulations may be used to determine the ventilation characteristics of a plant, and at DNV the in-house computer programme COFAN has become increasingly important within the areas of process technology and consequence analyses, and has proven to be an efficient tool for solving problems comprising complex flow phenomena.

COFAN - complex flow analysis - simulates fluid flow, heat and mass transfer in two- and three-dimensional geometries by solving the 'finite volume' versions of the differential equations governing the flow (Navier-Stokes) and using the k-ϵ model of turbulence, as described by Rastogi (1). The 'finite volumes' in question are arrays of contiguos 'cells', each of which possesses a typical value of the pressure, temperature, velocity components, etc. of the fluid. One of the advantages of COFAN is the adaptability to complex geometries. COFAN utilises a technique developed by Karki (2) that makes it possible to generate a body-fitted curvilinear grid, in which the grid lines may be chosen as coincident with arbitrarily shaped boundaries within the flow domain. The DNV pre- and post-processors PREFEM and POSTFEM are used to generate the numerical grid and to display the computed flow phenomena visually. POSTFEM can produce perspective views, contour diagrams, patterns of streamlines, vector diagrams and other quantitative representations of the simulated flow.

VENTILATION AND AREA CLASSIFICATION

Natural ventilation on offshore platforms is of crucial importance for area classification purposes. Obviously, ventilation will dilute and remove releases of explosive gases, and failure to do so could result in accumulation of explosive atmospheres and subsequent potential for ignition and explosions/fires. For these reasons, ventilation is made a subject of principal interest in the determination of zones, and extensions of such, in all codes and standards relating to area classification.

Adequate natural ventilation with respect to area classification considerations is defined through DNV RP C102 (3): "Areas with natural ventilation are adequately ventilated provided wind speed normally is greater than 2 m/s and only exceptionally less than 0.5 m/s". The IP-Code Part 15 (4), "Area Classification Code for Petroleum Installations", distinguishes between adequate natural ventilation in an open area and in an obstructed or sheltered open area. For an open area, adequate natural ventilation requires that air velocities should rarely be less than 0.5 m/s and should frequently be above 2 m/s. For obstructed or sheltered open areas the IP-Code defines adequate natural ventilation as "a uniform ventilation rate of at least twelve air changes per hour with no stagnant areas."

The number of air changes in a given module, the identification of stagnant areas, and the frequency of occurrence of air velocities less than 0.5 m/s and 2 m/s, respectively, are thus of specific interest to determine whether or not the requirements for adequate natural ventilation are met.

To establish the ventilation characteristics of an offshore installation by numerical simulations, a platform model of typically 50.000 grid elements is constructed. An example from the North Sea is shown in Fig. 1. Platform structures and process equipment are modelled as solid elements. An example is shown in Fig. 2a. A wind profile is further chosen for the inlet boundary conditions, and for this purpose a logarithmic profile as suggested by Lettau (5) is most often used. COFAN then solves for the local air velocities and pressure coefficients around the installation and within the platform modules (Fig. 2b). Repeated simulations with varying wind scenarios, typically 24 (3 wind speeds and 8 wind directions), in combination with the wind rose for the actual site will yield the frequency of occurrence of number of air changes in a given module (Fig. 3) as well as the frequency of occurrence of low velocities (Fig. 4). An example of typical wind rose data to be utilised for such calculations are shown in Table 1.

This method was applied on the EKOFISK Field in 1989. When the protective barrier was installed on the EKOFISK 2/4 Tank, extensive wind and ventilation simulations with COFAN were carried out for area classification purposes to establish the necessary data for reclassification after the installation of the barrier. Extensive pre- and post-barrier wind and ventilation measurements on the tank were also performed, and the agreement between measured and simulated values were, as a whole, found to be excellent. Today, the COFAN simulations from this study are used as a basis for the area classification on the 2/4 Tank.

MEAN WIND SPEED (m/s) 10 m above sea level	WIND DIRECTION (from)								TOTAL	EXCEEDENCE
	N	NE	E	SE	S	SW	W	NW		
0 - 3	0.8	0.6	0.6	0.5	0.6	0.6	0.6	0.6	4.9	95.1
3 - 6	2.8	2.3	1.9	1.9	2.6	2.7	2.7	2.7	19.5	75.6
6 - 9	3.7	2.6	1.5	2.2	4.2	3.6	3.5	3.0	24.2	51.4
9 - 12	3.2	2.3	0.6	2.0	4.7	3.9	2.9	2.3	21.9	29.4
12 - 15	2.1	1.2	0.3	1.5	4.0	2.5	1.8	1.5	14.8	14.6
15 - 18	1.2	0.6	0.1	1.0	2.9	1.4	1.1	1.0	9.2	5.4
18 - 21	0.4	0.2	0.02	0.4	1.4	0.5	0.4	0.4	3.6	1.8
21 - 24	0.1	0.04	-	0.2	0.5	0.2	0.1	0.1	1.2	0.6
24 - 27	0.02	0.01	-	0.1	0.2	0.05	0.03	0.05	0.4	0.16
27 - 30	-	-	-	0.02	0.1	-	0.01	0.1	0.1	0.05
30 - 33	-	-	-	-	0.02	-	-	0.01	0.04	0.02
33 - 36	-	-	-	-	0.01	-	-	-	0.02	-
TOTAL	14.2	9.8	5.0	9.6	21.0	15.2	13.3	11.7	100.0	

TABLE 1 All-year percentage frequency distribution of hourly mean winds. An example from the North Sea.

If both ventilation and working environment requirements are considered, the CFD simulations can further be used to define or optimise the use of louvres or cladding around the platform and evaluate the need for mechanical ventilation and space heaters.

GAS AND SMOKE DISPERSION

Having established, for different wind scenarios, the air velocity flow fields around and within the platform under consideration, gas or smoke release and consequent dispersion scenarios can be simulated by "adding" appropriate gas or smoke source terms at selected release locations in the model. The governing equation for the dispersion of turbulent buoyant jets is of the same nature as the heat transfer equation, and COFAN can thus be used to solve for the local gas or smoke concentration values in each of the cells comprising the grid. The application of CFD to predict the behaviour of such turbulent buoyant jets is described by e.g. Chen and Rodi (6).

Examples on problems to be studied this way include the following:

1) Determination of the likelihood of having ignitable gas-air concentrations into non-hazardous areas on the platform;

2) Determination of gas detector locations;

3) Investigations of smoke dispersion scenarios related to visibility and asphyxiation limits;

4) Influence of combustion gas from the flare tip and compression gas from burner booms on critical areas on the platform;

5) Influence of generator or turbine exhaust plumes on air intakes for the heating, ventilation and air conditioning (HVAC) system.

An example of a major gas release (100 kg/s) in the wellhead area of an offshore installation and the consequent gas dispersion scenario as calculated by COFAN is shown in Fig. 5. Another scenario caused by a 25 kg/s release below main deck level is presented in Fig. 6. The gas concentration distribution profiles are shown as contour diagrams in selected cross sections of the platform.

ACCURACY

At this point it would be appropriate to stress that the accuracy of the CFD calculations depend on the grid refinement. The number of elements chosen for a certain model depend on the actual geometry of the structure to be modelled and represent an assessment of the desired accuracy while minimising computing costs. A platform computer model of 50.000 elements implies local grid volumes of typically 1-10 m^3, which means that the calculated velocity components, temperatures, pressures and concentrations represent average values for each such volume. This is normally considered sufficient to obtain a good description of the ventilation characteristics for the platform and selected accident scenarios. If a higher accuracy is required, a finer grid resolution would be needed. For more accurate nearfield calculations of jet releases, it might also be worth while considering alternative turbulence models, e.g. the Reynolds stress transport closure.

WIDER APPLICATIONS

COFAN solves the governing equations of the fluid flow in a general manner, and the code thus comprises an "all-purpose" complex geometry flow solver that can be used for a wide variety of problems.

If the wind scenarios around an offshore installation once again are considered, the information gathered this way may also be used to evaluate the air flow structure above the platform helideck. Air velocities and turbulence parameters can be extracted from selected wind scenarios to evaluate air flow patterns, turbulence, possible vortices and wind gusts around the helideck and surrounding areas, thereby identifying critical areas or wind scenarios. The information gathered this way can further be used to define an optimal helicopter approach path.

Landbased petroleum plants or refineries are faced with almost identical problems to those of offshore installations, and hence the CFD solution procedure remains favourably applicable for onshore installations. For such plants, topographical influences on wind, ventilation and dispersion scenarios can be included in the models.

Finally, the use of CFD simulations on ventilation of large enclosures should be mentioned. Keywords here are air quality, contaminant dispersion and energy-efficient ventilation.

CONCLUSIONS

The concept of using numerical flow simulations has been presented as an efficient and economical tool compared to wind tunnel tests, to solve wind, ventilation and dispersion related problems on offshore installations. Particular emphasis has been put on area classification considerations and gas dispersion scenarios, and examples have been presented from North Sea installations. The numerical methods described provide for a wide applicability within safety engineering, and numerous supplementary application areas have been suggested.

ACKNOWLEDGEMENTS

My colleague, Jon Erik Lindeman of DNV Technica London, has contributed many fruitful discussions and clarifications for the numerical analysis of natural ventilation on offshore installations. The oil companies Norske Shell A/S and Phillips Petroleum Company of Norway A/S are gratefully acknowledged for permission to publish model examples from the North Sea.

REFERENCES

1 Rastogi, A. K., 1986, "Numerical calculation of fluid flow, heat and mass transfer in three-dimensional irregular domains", Veritas Research Report, 86-2028.

2 Karki, C. K., 1986, "A Calculation Procedure for Viscous Flows at all Speeds in Complex Geometries", PhD Thesis, University of Minnesota.

3 Det Norske Veritas, Recommended Practice C102, 1987, "Area Classification and Ventilation".

4 Institute of Petroleum, Model Code of Safe Practice in the Petroleum Industry, 1990, "Area Classification Code for Petroleum Installations", Part 15.

5 Lettau, H. H., 1959, "Wind Profile, Surface Stress and Geostrophic Drag Coefficients", Advances in Geophysics, Vol. 6.

6 Chen, C. J. and Rodi, W., 1980, "Vertical turbulent buoyant jets", Pergamon Press.

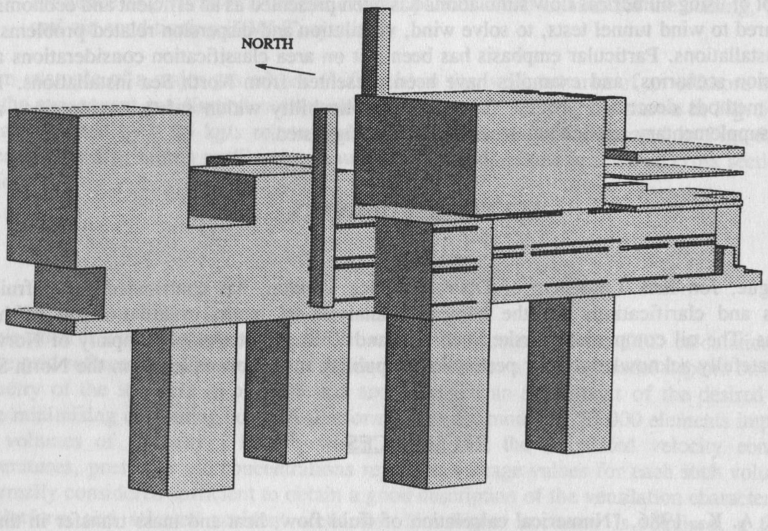
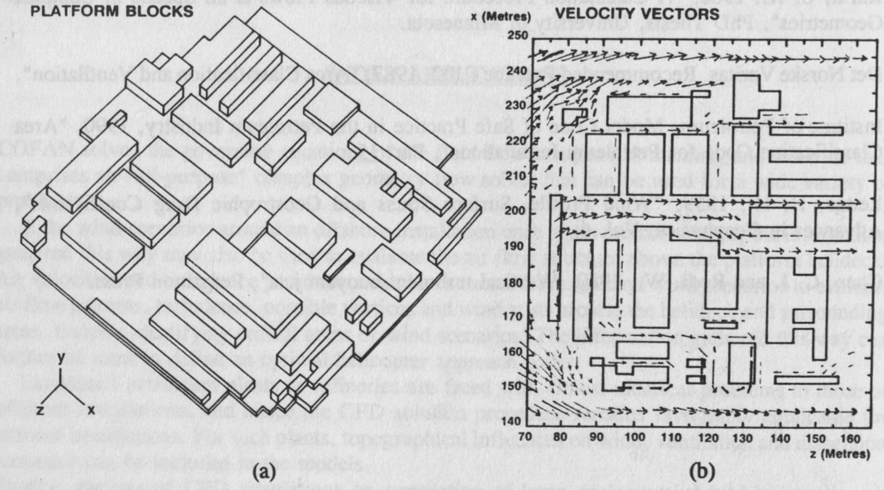


Figure 1 Computer model of an offshore installation in the North Sea.



(a) (b)

Figure 2 Blocked elements (a) simulating process equipment on the main deck of an offshore module, and corresponding calculated air velocity vectors (b) within the module.

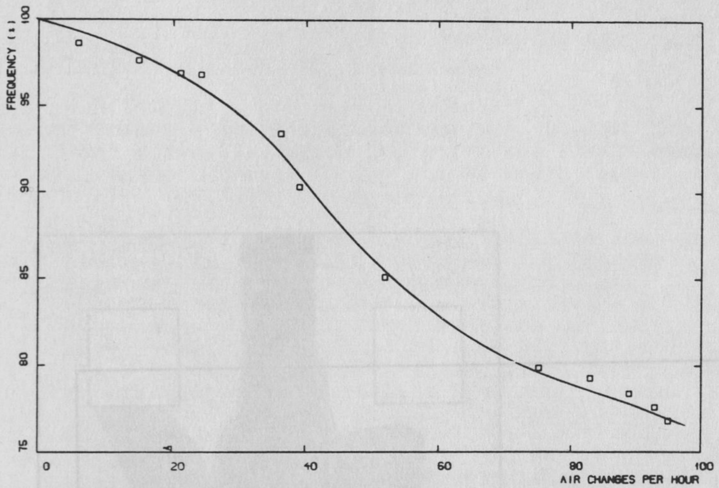


Figure 3 Calculated frequency of occurence of air changes per hour within an offshore module, based on information from the wind rose from the actual site and numerical simulations of the air flow fields for different wind speeds and directions.

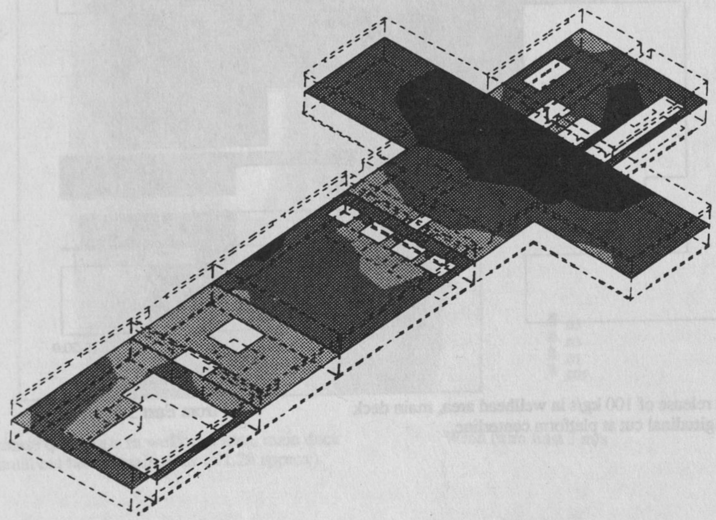


Figure 4 Calculated frequency of occurrence of air velocities less than 0.5 m/s on the main deck of an offshore installation, based on information from the wind rose for the actual site and numerical simulations of the air flow fields within the platform area for different wind speeds and directions. Shade darkness increases with decreasing frequency of occurrence of low velocities.

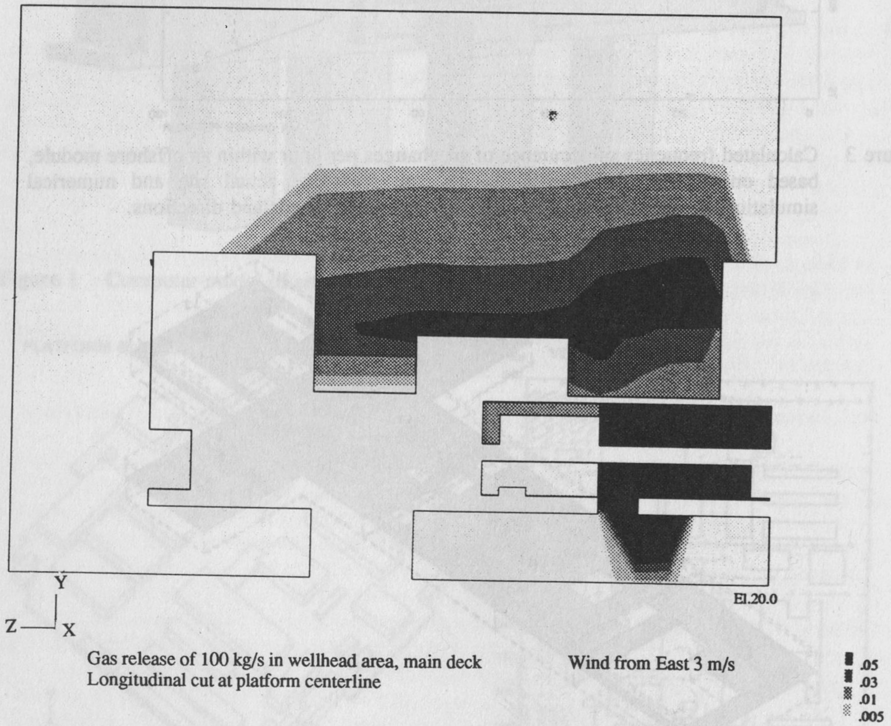


Figure 5 Calculated gas concentration contours for a gas dispersion scenario comprising a major release of 100 kg/s in the wellhead area of an offshore installation and wind from East at 3 m/s. Contours are given in a longitudinal cut along the platform centerline.

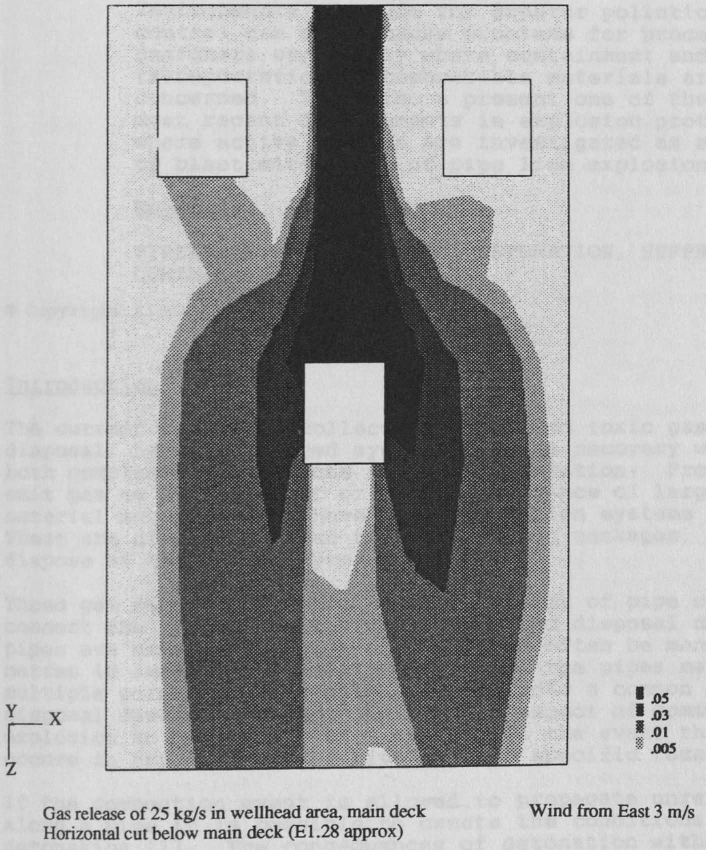


Figure 6 Calculated gas concentration contours for a gas dispersion scenario comprising a major release of 25 kg/s in the wellhead area of an offshore installation, directed downwards below the main deck, and wind from East at 3 m/s. Contours are given in a horizontal cross section below the main deck.