WIND LOADS ON SHIPS AND OFFSHORE STRUCTURES ESTIMATED BY CFD

Christian Aage [1], Søren L. Hvid [2], Philip H. Hughes [3] and Michael Leer-Andersen [1]

[1] Department of Naval Architecture and Offshore Engineering, Technical University of Denmark,
Building 101 E, DK-2800 Lyngby, Denmark
[2] Danish Maritime Institute, Hjortekærsvej 99, DK-2800 Lyngby, Denmark
[3] MAN B&W Diesel A/S, Teglholmsgade 41, DK-2450 Copenhagen SV, Denmark

ABSTRACT

Wind loads on ships and offshore structures could until recently be determined only by model tests, or by statistical methods based on model tests. By the development of Computational Fluid Dynamics or CFD there is now a realistic computational alternative available. In this paper, wind loads on a seagoing ferry and on a semisubmersible offshore platform have been estimated by CFD. The results have been compared with wind tunnel model tests and, for the ferry, a few full-scale measurements, and good agreement is obtained. The CFD method offers the possibility of a computational estimate of scale effects related to wind tunnel model testing. An example of such an estimate on the ferry is discussed. Due to the time involved in generating the computational mesh and in computing the solution, the CFD method is not at the moment economically competitive to routine wind tunnel model testing.

KEYWORDS

Wind loads, ships, offshore, computational fluid dynamics, CFD, model tests, full-scale tests, scale effects.

INTRODUCTION

Wind loads on ships and offshore structures play an important part in almost any operation at sea. The function and safety of floating vessels is dependent upon an accurate prediction of the wind loads at the design stage. More specifically, the following technical areas, where wind loads are crucial, can be mentioned:

Wind loads on *ships* are of special importance in relation to the manoeuvrability at low speed, where the hydrodynamic forces on the underwater hull are small and difficult to control. Consequently, accurate wind load data are of great importance for the programming of ship manoeuvring simulators. The wind overturning roll moment is the determining factor for the dynamic stability of ships. The wind resistance of ships does not generally influence the design as much as in earlier times, when streamlined superstructures were more usual. However, the importance of obtaining accurate knowledge of the ship's wind resistance is unchanged for the purpose of optimum design of the propulsion plants and for the analysis of trial trips.

The use of ships and ship-like structures as moored offshore production, storage and offloading vessels has underlined the need for reliable prediction of the wind loads on ships, also in an offshore context.

Wind loads on *floating offshore platforms*, especially on semisubmersibles with their rather low metacentric height, have a determining influence on the stability and deck load capacity of these vessels. The wind overturning moment on a semisubmersible is one of the important design parameters, especially in the damage condition where the wing lift effect on a tilted open deck can significantly increase the overturning moment, as pointed out by Bjerregaard & Sørensen (1981). Design of dynamic positioning systems, as well as offshore anchor mooring systems, is relying on precise information and prediction of the wind forces and moments in the horizontal plane.

Until recently, the only methods available to estimate these loads would be experimental, either in the form of direct wind tunnel model testing, or in the form of statistical estimates, based on model tests of a large number of similar vessels, and combined with theoretical or empirical descriptions of the aerodynamic loads. Isherwood (1973) and Blendermann (1993), among others, have made valuable contributions to this method.

In recent years, however, the Computational Fluid Dynamics or CFD methods have been developed to a stage where they offer a realistic computational alternative to the experimental methods.

The CFD method is based on the finite volume principle, in which the flow domain is divided into a very large number of small fluid volumes or cells, each being governed by a set of dynamic and kinematic equations, interacting with its neighbour elements and with the flow domain boundaries. The investigated structure, e.g. a ship or an offshore platform, will act as one of these boundaries. When the large system of equations and boundary conditions is solved, one of the results is the total mapping of the pressures upon the surface of the investigated structure. By integrating the pressures over the vessel's surface, the wind loads, i.e. the forces and moments, can be found as the most important result in this context.

Even if the basic theory is quite simple, many practical problems have to be solved before useful results can be expected. Crucial for good results are the choice of a suitable CFD program or flow solver package, the mesh generation, the formulation of boundary conditions, the wind profile and turbulence modelling.

In this paper, wind loads on a seagoing ferry and on a semisubmersible platform have been estimated by CFD. The vessels were chosen as representative of two very important types of maritime structures. Both vessels have been model tested in the wind tunnels of the Danish Maritime Institute (DMI). Furthermore, some full-scale wind load measurements have been carried out on the ferry in the harbour of Rønne in Denmark.

When determining wind loads on ships and offshore structures by model testing, the possibility of scale effects must always be considered. Even in large wind tunnels, the model Reynolds number will at best be about a hundred times lower than the full-scale number.

The only way to experimentally determine a possible scale effect is by comparison with full-scale measurements, which are both difficult and costly to carry out, and very few full-scale wind load measurements have been published. Boonstra & Leynse (1982) have compared wind-tunnel and full-scale wind loads on a pentagon semisubmersible platform. Their measurements display a very large scatter on the full-scale results with mean values at about 60 per cent of the model test results and maximum values close to, but not exceeding the model test values.

The CFD method offers an interesting solution to the scale-effect problem. By executing the CFD computation with both model-scale and full-scale flow parameters a computational estimate of the scale effects can be obtained. Such an investigation has been carried out on the ferry for a single wind direction.

Except for this scale-effect investigation, all CFD computations on the ferry and the semisubmersible have been carried out with full-scale flow parameters.

DEFINITIONS OF LOADS AND COEFFICIENTS

The wind forces and moments on the above-water parts of a ship or an offshore platform, and the relative wind direction, are defined as shown in Figure 1. The origin of the co-ordinate system is placed at the intersection of the centreline-plane, the waterline-plane and the midship-plane. The non-dimensional force and moment coefficients are defined as follows:

$C_X = X / (½ \rho U_{10}^2 A_f)$ longitudinal force
$C_Y = Y / (½ \rho U_{10}^2 A_s)$ transverse force
$C_Z = Z / (½ \rho U_{10}^2 A_f)$ vertical force
$C_K = K / (½ \rho U_{10}^2 A_s H_s)$ roll moment
$C_M = M / (½ \rho U_{10}^2 A_f H_f)$ pitch moment
$C_N = N / (½ \rho U_{10}^2 A_s L_{oa})$ yaw moment

where

ρ	density of air
U_{10}	wind velocity at a height of 10 m above sea level
A_f	projected front area
A_s	projected side area
H_f	height of geometric centre of front area
H_s	height of geometric centre of side area
L_{oa}	length over all

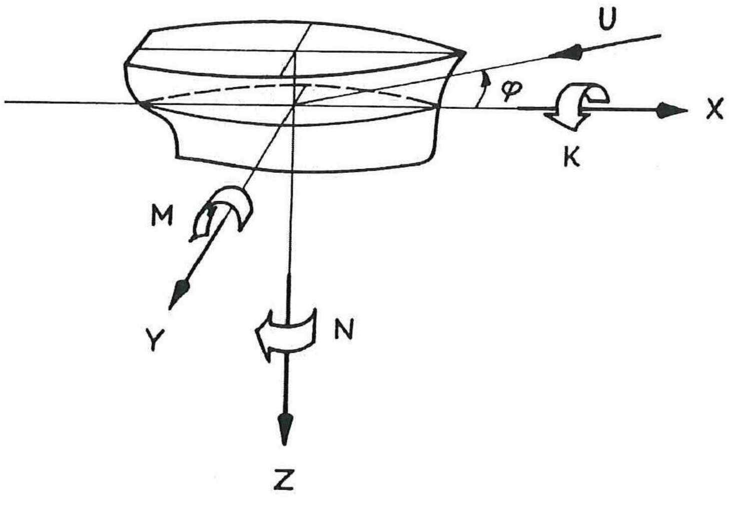


Figure 1. Co-ordinate system of wind forces and moments

Table 1. Main dimensions of the ferry M/F "Povl Anker"

Length over all	L_{oa}	121.00 m
Length between perpendiculars	L_{pp}	109.00 m
Breadth	B	21.50 m
Draught fore	T_f	4.45 m
Draught aft	T_a	5.25 m
Projected front area	A_f	448.00 m^2
Projected side area	A_s	2079.00 m^2
Height of geometric centre of front area	H_f	10.40 m
Height of geometric centre of side area	H_s	10.10 m

THE INVESTIGATED VESSELS

The Ferry

The ship chosen for this investigation was the seagoing ferry M/F "Povl Anker" and its sister ship M/F "Jens Kofoed", owned by BornholmsTrafikken and sailing between Copenhagen and Rønne on the island of Bornholm in the Baltic Sea. The ships were built by Aalborg Shipyard A/S in 1978 and 1979. The main dimensions of the ferry are found in Table 1. The CFD model of the ferry is seen in Figure 2.

From an aerodynamic point of view this type of vessel is interesting because it features some sharp edges and corners, e.g. at the bulwarks, gunwales and the bridge, together with a rather streamlined above-water hull. The shape is typical of many ferries and passenger ships. It should be noted that in this CFD model the two masts and the lifeboats have been omitted.

A specific reason for choosing the two sister ships M/F "Povl Anker" and M/F "Jens Kofoed" for this investigation was the possibility, by the kind co-operation of BornholmsTrafikken and the local harbour authorities, of carrying out full-scale wind load measurements in the harbour of Rønne.

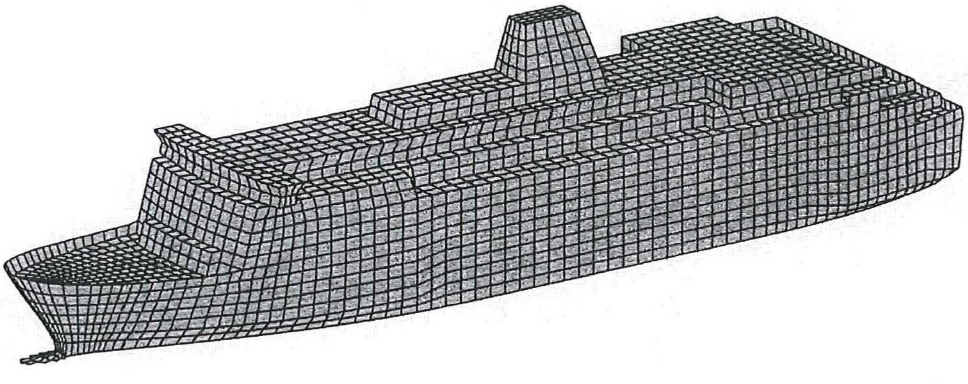


Figure 2. CFD model of the seagoing ferry M/F "Povl Anker"

Table 2. Main dimensions of the semisubmersible "Deepcore"

Length over all	L_{oa}	76.20 m
Length between perpendiculars	L_{pp}	68.30 m
Breadth	B	64.50 m
Height to bottom of deck	H_{bd}	18.30 m
Height to top of deck	H_{td}	21.00 m
Height over all	H_{oa}	38.00 m
Projected front area	A_f	1019.00 m^2
Projected side area	A_s	1025.00 m^2
Height of geometric centre of front area	H_f	21.00 m
Height of geometric centre of side area	H_s	16.70 m

The Semisubmersible

The offshore structure chosen for this investigation was a generic semisubmersible platform "Deepcore" with eight columns and a single deck with superstructures. The "Deepcore" may be considered as a typical accommodation platform. The main dimensions are found in Table 2. The CFD model is seen in Figure 3.

The aerodynamic loads on the semisubmersible will be composed of Reynolds number dependent components from the columns as well as components from the sharp-edged superstructure that are more or less Reynolds number independent. Of special interest is the lift force induced on the deck, especially in a tilted position. Lift forces on "Deepcore" in tilted positions can be found in Hvid et al. (1997). But even at level trim, as in the present case, substantial lift forces can be seen on a semisubmersible.

The model does not comprise any cranes, booms, or drilling derrick. As mentioned below, modelling of lattice structures for a CFD calculation is very time consuming with respect to both man-hours and CPU-time, and was not found feasible in this investigation. The equivalent modelling of such details for a physical model test is also a matter of discussion and uncertainty, and further study of the optimum modelling of lattice structures seems to be needed.

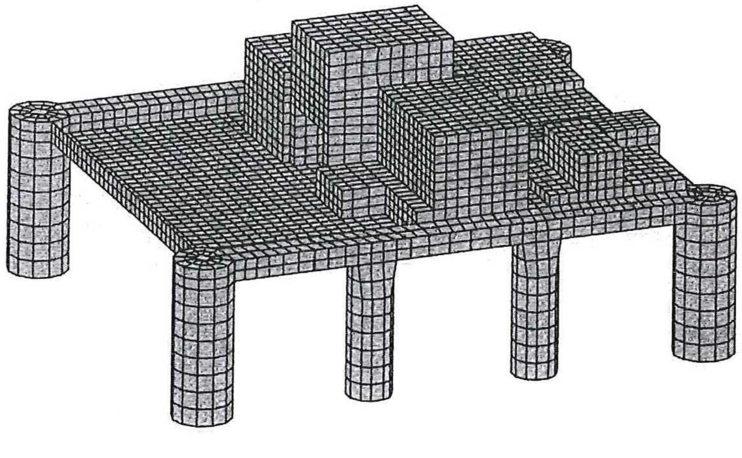


Figure 3. CFD model of the semisubmersible "Deepcore" (front end to the right)

THE CFD METHOD

The basic principle underlying the Computational Fluid Dynamics or CFD method is similar to the principle of the well-known finite element method used in solid mechanics: By dividing the domain of interest into a very large number of volumes or elements, each governed by the differential equations that describe the dynamics and kinematics of the problem, for the flow problem the Navier-Stokes equations, and each interacting with its neighbour elements and with the flow domain boundaries, the flow problem is reduced to the solution of a large number of linear equations. The general CFD theory has been described by e.g. Patankar (1980).

In this investigation the applied program was the STAR-CD® general purpose CFD-code, Version 2.3, developed by Computational Dynamics Ltd. The program is based on the Finite Volume or FV method. The program has been designed primarily for internal flow in components related to the automobile industry. When applying the code to a ship or a platform, the vessel is considered as an obstacle in a semi-infinite space. The domain boundaries must be placed so far away from the vessel that the exact position of the boundaries does not influence the flow in the near-field significantly. The water surface and the surface of the vessel are the only impermeable boundaries. Over the water surface an atmospheric boundary layer is established by a set of upstream conditions. At the structural surfaces local boundary layers are developed by the CFD program in as much detail as the fineness of the mesh allows.

In the following, some of the more important CFD modelling problems will be discussed: geometric modelling and mesh generation, boundary conditions, the atmospheric boundary layer and turbulence modelling.

Geometric Modelling and Mesh Generation

The geometric modelling of the vessel and the corresponding mesh generation is done by hand and is a very time consuming process, taking several hundred hours for a typical ship or platform. Automatic mesh generation procedures, that would greatly improve the practical possibilities of the CFD method, are not yet developed sufficiently to be useful.

Even with the application of very fast computers the number of finite volumes can easily become prohibitive for a practical CFD calculation, and therefore the number of elements must be limited to the smallest number that can still provide a reasonably correct description of the vessel geometry and model the flow to an adequate detail. A fine mesh is necessary near the structure, whereas a gradually coarser mesh will suffice at a distance away from the structure. The mesh resolution was chosen to be about 1 m close to the structures and about 15 m in the outer region. The computational flow domain was 7 times the overall length of the ferry and 15 times the overall length of the semisubmersible. In both cases, the domain was 4 times the overall height and 16 times the overall breadth of the vessels, as seen in Figure 4. To minimize the number of cells, the vessels were placed asymmetrically to the upstream side of the flow domain. For the two vessels investigated here, the number of computational cells was about 100 000.

Ships and offshore platforms usually exhibit very complex geometries, including masts, lattice towers, bulwarks, funnels, etc., that will require an extremely large number of mesh points to be accurately modelled. Unfortunately, some of these elements can have a significant effect on the vortex formation, the local turbulence and the flow separation points, and thereby on the aerodynamic loads. So even if simplifications are made whenever considered allowable, complex details as the ones mentioned above should be included and will require a substantial part of the mesh generation efforts, and indeed also of the computation time.

In the present study some simplifications had to be made to limit the number of mesh points. On the ferry, the two masts and all lifeboats were excluded from the CFD model, but not from the wind-tunnel model. On the semisubmersible platform, all booms, cranes and other lattice structures, as well as the helideck were excluded both from the CFD model and from the wind-tunnel model. These simplifications should be borne in mind when comparing CFD results with the wind-tunnel and full-scale results.

Boundary Conditions

Apart from the influence of spatial resolution the computational solution should also be proved to be insensitive to the applied boundary conditions. It is often possible to choose boundary conditions that minimize unintended effects on the solution. In the case of external flow around an obstacle, constant pressure boundaries that allow mass to enter and leave the domain are well suited to avoid a possible blockage effect. However, the boundaries should be placed so far away from the structure that the uniform pressure condition can be assumed correct. The following boundary conditions have been applied to the flow domain:

1. Specified velocity, turbulent kinetic energy and dissipation profiles at the inflow boundaries.
2. Constant pressure boundaries at the top and side boundaries, except at the inflow boundaries, allowing fluid to pass out of as well as into the domain.
3. No-slip, adiabatic walls with specified surface roughness for all other boundaries, i.e. water surface and structural surfaces.

The mean roughness height of the sea surface was set to a value of 1.0 mm, because this indeed unrealistic roughness resulted in the best equilibrium atmospheric boundary-layer profile, i.e. the profile closest to the power law profile, as discussed below. The mean roughness height of the structural surfaces was set to a value of 0.8 mm.

Atmospheric Boundary Layer Profile and Turbulence Modelling

The airflow past a streamlined body, and hence the aerodynamic loads, depend strongly on the properties of the upstream flow, its velocity and turbulence profiles. For bluff bodies, however, the flow separation points are determined by the geometry of the body rather than by the flow conditions. Fortunately, the above-water parts of most ships and offshore structures can be characterized as bluff bodies with sharp edges, and therefore the modelling of the flow is not very critical. This is a definite advantage when carrying out wind tunnel tests on these vessels, where correct Reynolds number cannot be obtained. But still, parts of the airflow, e.g. along the hull sides of a ship and around the circular columns of a semisubmersible will depend strongly on the Reynolds number and the upstream flow conditions.

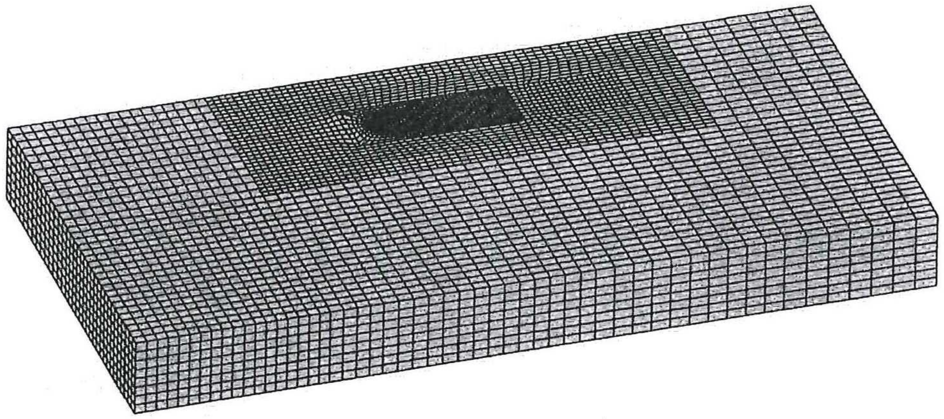


Figure 4. Computational flow domain around the ferry

The atmospheric boundary layer velocity profile is an important flow parameter, because different profile shapes will create different pressure distributions on the body, and hence different loads. For the same reason, the boundary layer should be in equilibrium. If the flow field changes significantly as it approaches the tested model, the results will be affected and cannot be associated with any well-defined condition. The correct method by which to check that the applied inlet velocity and turbulence profiles are in equilibrium is to replace the model geometry by computational fluid cells. In the resulting obstacle-free domain, the specified inlet and computed outlet profiles should be identical, which is easily checked.

It is generally necessary to adjust either the coefficients of the turbulence model or other model parameters to obtain an equilibrium profile. It is difficult to match both the velocity and the turbulence profiles with the assumed target boundary-layer profiles, and the computational result does depend on the available turbulence model. Whereas the velocity profile has been well matched, the turbulence intensity is far too high in the CFD model. The expected consequence is an increase in loads, not as a result of the increased turbulent energy in itself, but rather as a result of smaller detachment zones behind the structural elements.

The decrease in size of detached flow regimes is associated with the increase in apparent viscosity. This is a result of the fundamentally diffusive nature of the applied turbulence model, which is an RNG variant of the k-ε model. This model is known to underpredict the size of recirculation zones in the wake of obstacles.

Computer facilities

The CFD computations were performed on a Hewlett-Packard computer workstation type HP J210, operating a PA7200 CPU at 120 Mhz. The solution of each flow condition required between 1½ and 2 hours of CPU time before the normalized residuals had fallen to below the accuracy limit of 0.0001.

WIND TUNNEL TESTS

The model tests were carried out in two different wind tunnels at the Danish Maritime Institute. In both tunnels, atmospheric boundary layers could be modelled with a correct velocity profile, whereas the turbulence characteristics could only partially be matched with the full-scale conditions.

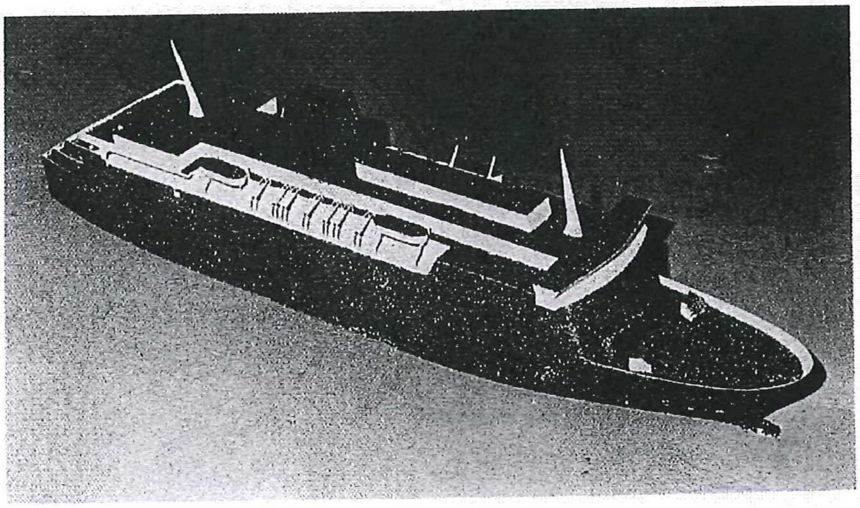


Figure 5. Wind tunnel model of the ferry M/F "Povl Anker" at scale 1: 100

The Ferry

Model testing of the ferry M/F "Povl Anker" was performed in an open-circuit boundary-layer wind tunnel with a measurement section of 2.6 m × 1.8 m, fitted with a six-component balance. Upstream of the model there is a 20 m inlet section, where different roughness elements can be installed to obtain the best possible fit to the full-scale atmospheric boundary layer. The maximum wind speed during the tests was 20 m/s.

The M/F "Povl Anker" model was built to the scale of 1:100. The model is seen in Figure 5. Contrary to the CFD model the wind-tunnel model was fitted with masts and lifeboats.

An initial Reynolds number sensitivity test showed that, the maximum wind speed of 20 m/s was not high enough to obtain constant wind load coefficients. The problem seemed to be almost solved by the application of sand roughness on the model.

The ferry model was tested at wind directions 0° to 180° with 5° intervals. All tests were carried out with the model on even keel. The model was tested both in a free stream, simulating the open sea conditions as in the CFD computation, and in a landscape model of Rønne Harbour, as seen in Figure 7, simulating the full-scale test conditions. The results are presented in Figures 9 and 11.

The Semisubmersible

Model testing of the semisubmersible "Deepcore" was performed in a closed-circuit wind tunnel with a measurement section of 1.0 m × 0.7 m, fitted with a six-component balance. The maximum wind speed during the tests was 32 m/s.

The "Deepcore" model was built to the scale of 1: 250. The model is seen in Figure 6. As mentioned above, the model was built without any lattice structures and without helideck. It should not be considered as a realistic scale model of an existing semisubmersible, but rather as a simplified generic accommodation platform. The purpose was to test a model that could be realistically modelled in the CFD computations.

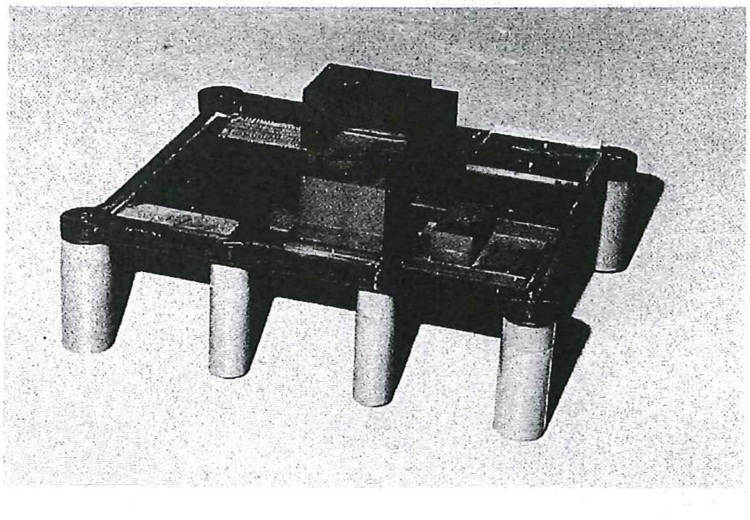


Figure 6. Wind tunnel model of the semisubmersible "Deepcore" at scale 1: 250

A Reynolds number sensitivity test was carried out, showing a levelling out of the aerodynamic loads at a wind speed below the test wind speed. Sand roughness was not applied on this model.

The semisubmersible was tested at wind directions 0° to 360° with 10° intervals. All tests were carried out with the model on even keel. The results are presented in Figure 10. CFD and wind tunnel test results of the same model in different tilted conditions can be found in Hvid et al. (1997).

FULL-SCALE TESTS

A series of full-scale wind load measurements have been carried out on the two sister ferries M/F "Povl Anker" and M/F "Jens Kofoed" in Rønne Harbour. One reason for choosing the harbour for the test, instead of the open sea, was that waves and current, which will normally appear together with strong winds at sea, were absent. Another reason was that the ships could be kept in a steady position by use of the necessary number of moorings, whereas a similar number of anchors at sea could be prohibitive.

The disadvantage of measuring in a harbour is of course that the airflow will be disturbed by the piers, buildings and other ships in the harbour. To overcome this problem and make the comparison between model-scale and full-scale results meaningful, the harbour environment was modelled in the wind tunnel together with the ship, as seen in Figure 7. The intention is that the model landscape shall create a natural boundary layer, as well as create the correct local wind field near the ship, which is influenced by nearby buildings and other obstacles.

The wind speed and direction were measured at the top of the aft mast, both in the full-scale and in the model-scale tests. This local set of wind speeds and directions was used as a link to the free stream airflow, which was unknown in the full-scale tests.

The wind loads were measured by means of a number of force gauges shackled into the mooring ropes, as seen in Figure 8. The measured forces combined with the positions and angles of the mooring ropes yielded the longitudinal force (X) and the transverse force (Y) as presented in Figure 11.

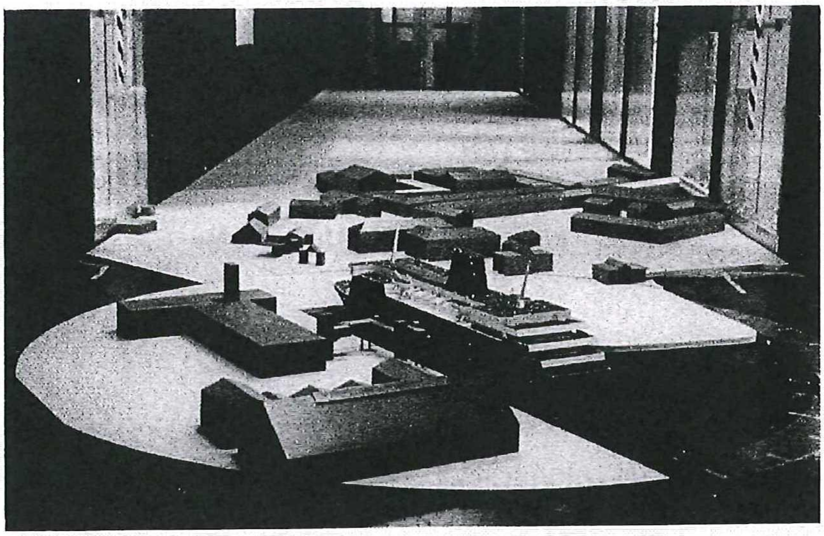


Figure 7. Wind tunnel model of the ferry M/F "Povl Anker" in Rønne Harbour at scale 1: 100

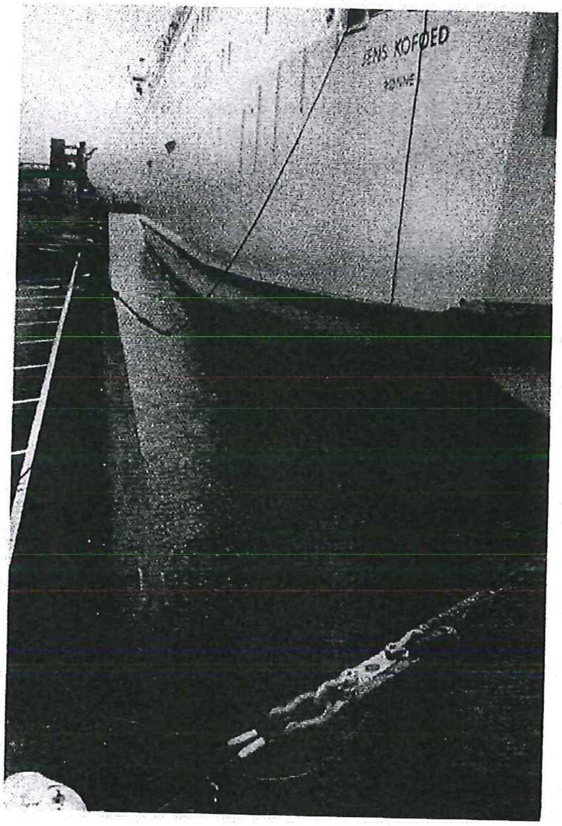


Figure 8.
Full-scale wind load measurements on the ferry M/F "Jens Kofoed", sister ship of M/F "Povl Anker", in Rønne Harbour

SCALE EFFECTS ESTIMATED BY CFD

The CFD method offers an interesting solution to the scale-effect problem. By executing the CFD computation with both model-scale and full-scale flow parameters, a computational estimate of the scale effects can be obtained. Such a computation has been performed on the seagoing ferry M/F "Povl Anker" at a wind direction of $\varphi = 40°$, i.e. bow wind. The wind speed was $U_{10} = 8{,}5$ m/s in the model-scale as well as the full-scale case, both modelled with an open water boundary layer. The results are presented in Table 3.

Table 3. Wind load coefficients of the ferry M/F "Povl Anker" computed by CFD with model-scale and full-scale flow conditions for wind direction $\varphi = 40°$

Coefficient	Model scale	Full scale	Difference %
C_X	- 0.13	- 0.28	54
C_Y	0.89	0.97	8
C_Z	- 2.81	- 2.93	4
C_K	0.84	0.95	12
C_M	0.29	0.31	6
C_N	0.05	0.07	29

In this test, which can only be considered as one example, not necessarily typical, the model-scale wind load coefficients differ from the full-scale coefficients by up to 54 per cent, with the longitudinal force being absolutely the problematic one. A difference in the longitudinal force at a longitudinal wind direction is expected, because the flow along the streamlined hull is very Reynolds-number dependent.

RESULTS AND CONCLUSIONS

The comparison of CFD computations, wind tunnel model tests, and full-scale measurements show an overall good agreement, even if large discrepancies are indeed seen at some wind directions. The differences between CFD and model-test results are not generally larger than between full-scale and model-scale results. Actually, the differences are not much larger than often found when the same vessel is tested in different wind tunnels. Therefore, it is concluded that determination of wind loads on ships and offshore structures by CFD is a realistic computational alternative to the experimental methods. However, due to the time involved in generating the computational mesh and in computing the solution, the CFD method is not at the moment economically competitive to routine wind-tunnel model testing.

The comparison of CFD computations carried out with model-scale and full-scale flow parameters, respectively, on the ferry for one wind direction has given an estimate of the scale effects. The model-scale coefficients seem to be somewhat smaller than the full-scale values. The same trend is actually found in the comparison of the wind-tunnel results with full-scale measurements. A more systematic investigation of the scale-effect problem along these lines will be necessary before firmer conclusions can be drawn.

ACKNOWLEDGMENTS

The co-operation of the management of BornholmsTrafikken, the Rønne Harbour personnel, the crews of M/F "Povl Anker" and M/F "Jens Kofoed" and several students of the Technical University of Denmark during the full-scale measurements, is gratefully acknowledged. Thanks are also due to Mr. Uffe Lind, Mr. Peter Tyroll and Mr. Arne C. Mikkelsen of the Danish Maritime Institute for their assistance with instrumentation and measurements.

REFERENCES

Bjerregaard, E.T.D. & Sørensen, E.G. (1981): "Wind Overturning Effects Obtained from Wind Tunnel Tests with Various Semisubmersible Models", Proceedings of Thirteenth Annual Offshore Technology Conference, Houston, Texas, OTC 4124, Vol. 4, pp. 89-97.

Blendermann, W. (1993): "Wind Loads on Moored and Manoeuvring Vessels", Proceedings of the 12th International Conference on Offshore Mechanics and Arctic Engineering, Glasgow, OMAE / ASME, Vol. I, pp. 183-189.

Boonstra, H. & Leynse, C. (1982): "Wind-Tunnel Tests on a Model of a Semisubmersible Platform and Comparison of the Results With Full-Scale Data", Proceedings of Fourteenth Annual Offshore Technology Conference, Houston, Texas, OTC 4245, Vol. 2, pp. 177-192.

Computational Dynamics Ltd. (1995): "STAR-CD® Manual", Version 2.3, Volumes 1 & 2.

Hvid, S.L., Leer-Andersen, M. & Hughes, P.H. (1997): "Wind Load Prediction on Offshore Structures Using CFD", Proceedings of the Conference on Application of Fluid Dynamics in the Safe Design of Topsides and Superstructures, The Institute of Marine Engineers, London, England, 15 pp.

Isherwood, R.M. (1973): "Wind Resistance of Merchant Ships", Transactions of The Royal Institution of Naval Architects, London, Volume 115, pp. 327-338.

Leer-Andersen, M. & Hughes, P.H. (1996): "Computation of Wind Loads on Ships and Offshore Structures", M.Sc. Thesis, Department of Naval Architecture and Offshore Engineering, Technical University of Denmark, 327 pp.

Patankar, Suhas V. (1980): "Numerical Heat Transfer and Fluid Flow", Hemisphere Publishing Company, Cambridge, England, 212 pp.

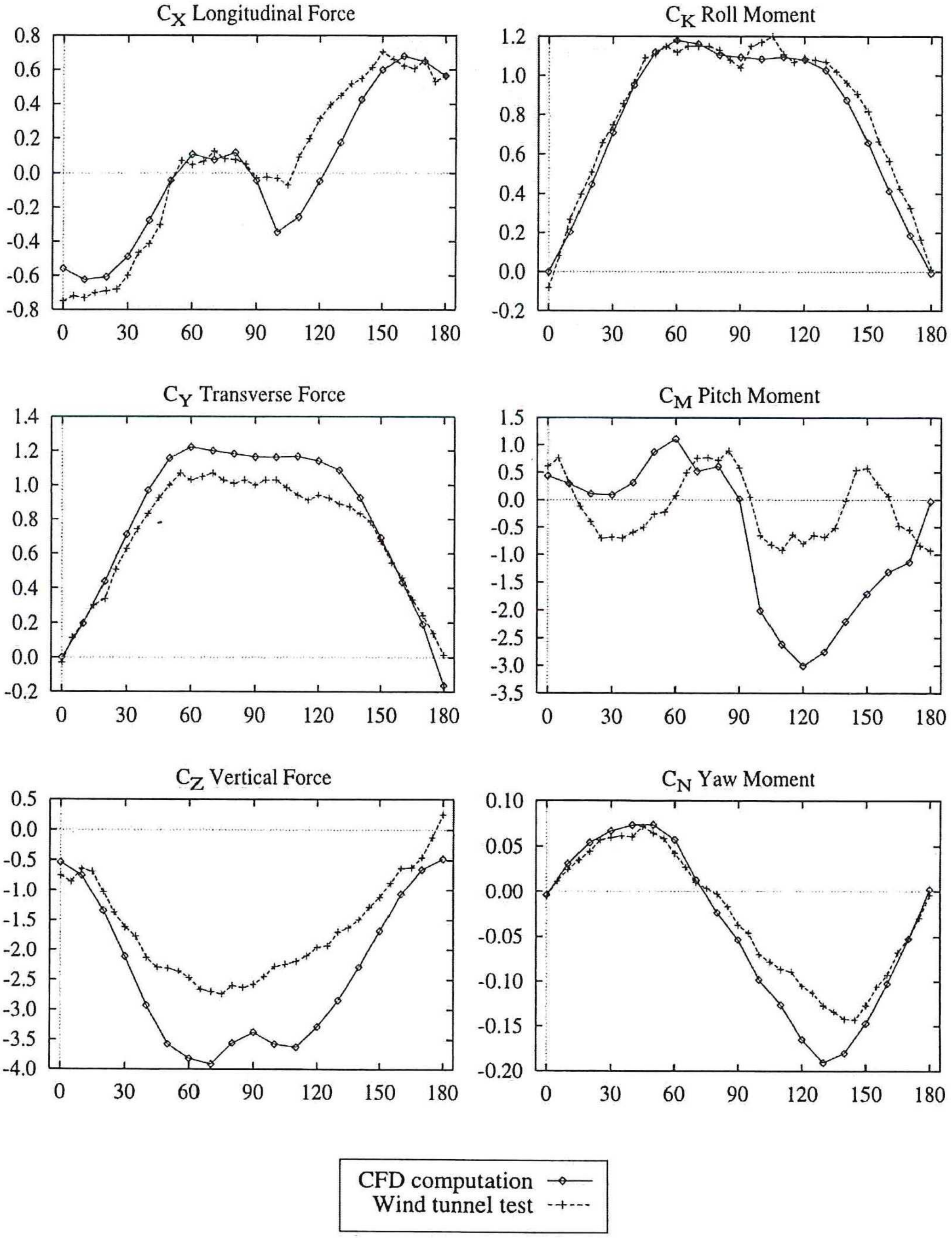


Figure 9. Wind load coefficients as functions of the free stream wind direction on the ferry M/F "Povl Anker" determined by CFD computations and wind tunnel model tests

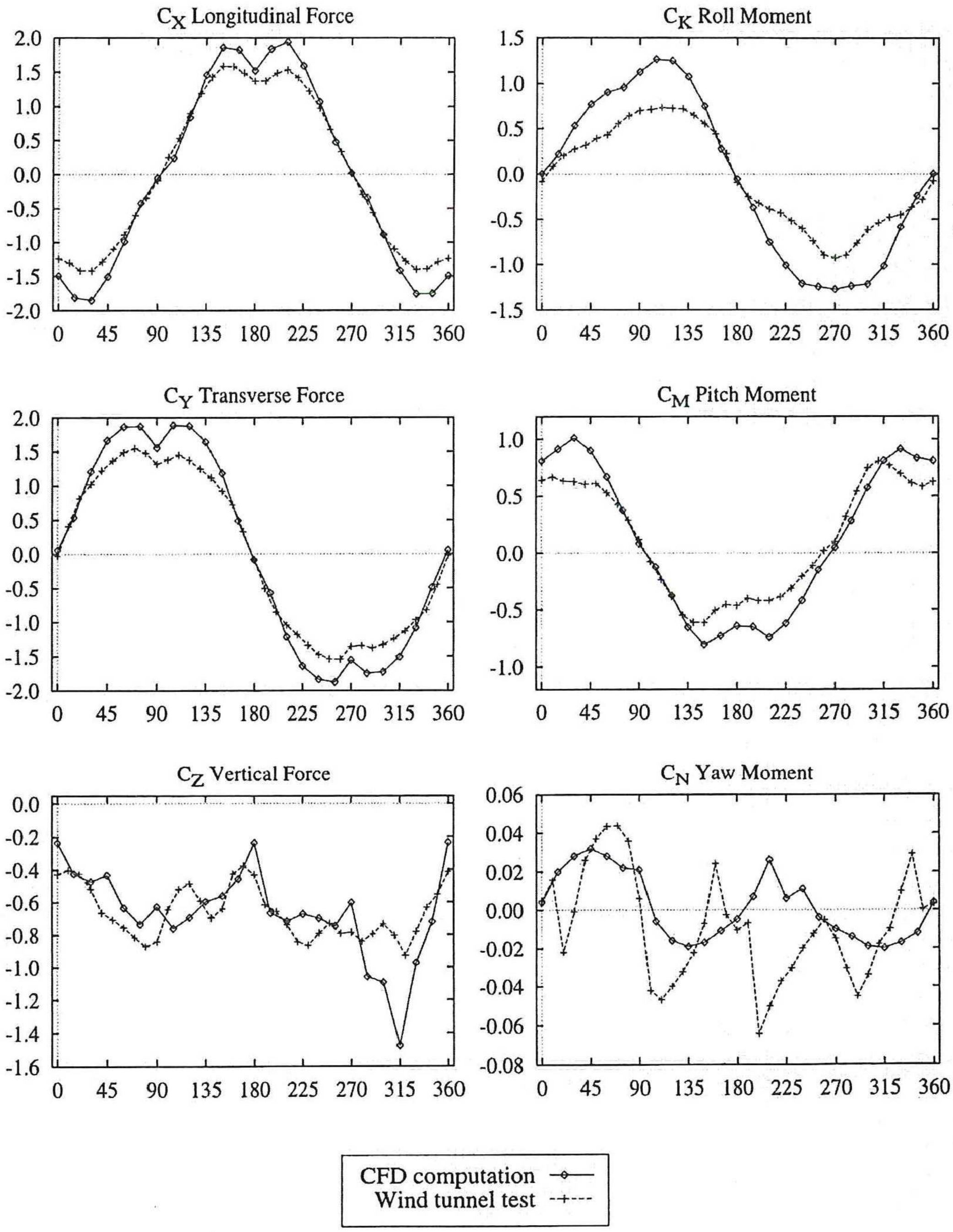
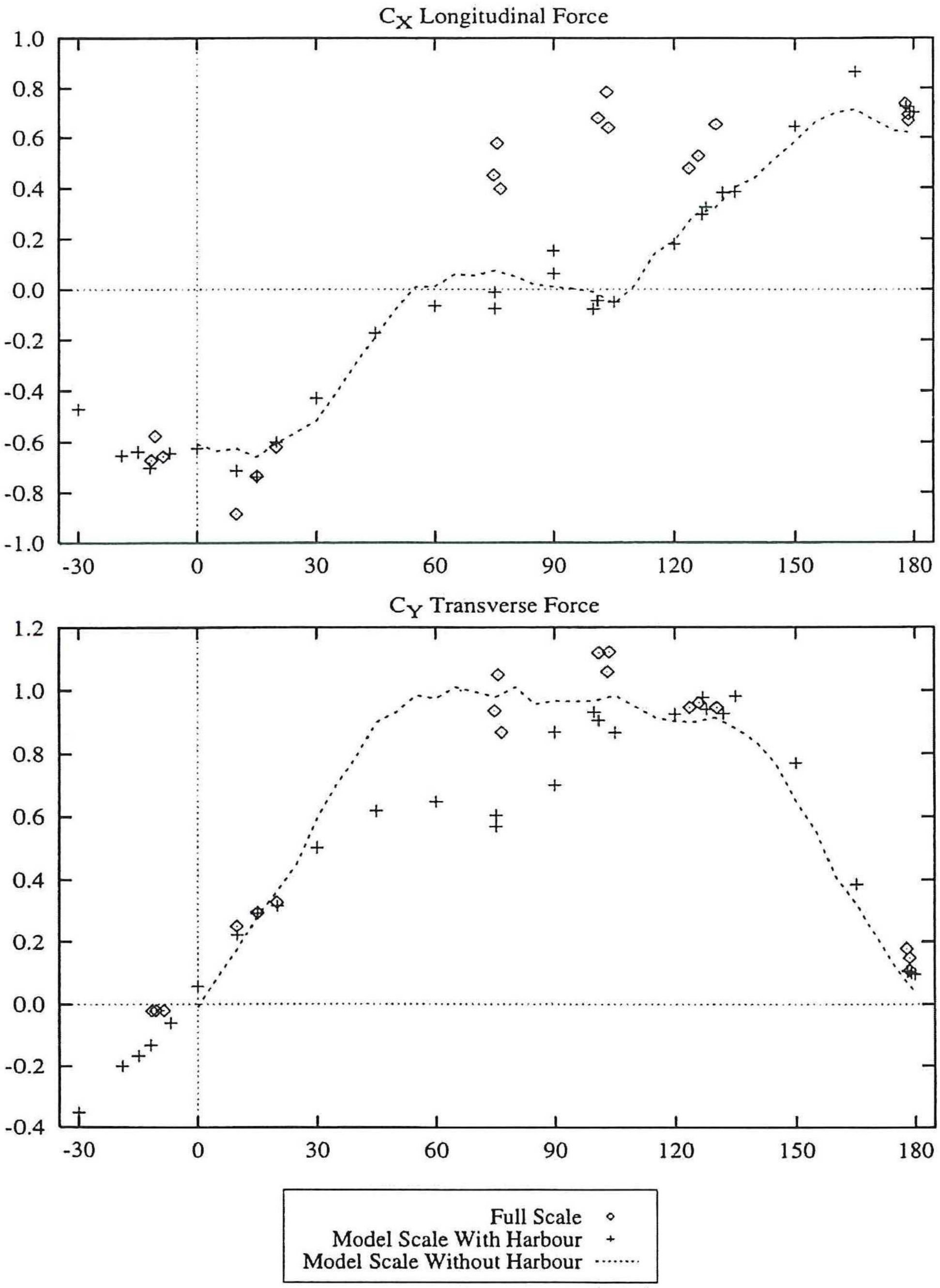


Figure 10. Wind load coefficients as functions of the free stream wind direction on the semisubmersible "Deepcore" determined by CFD computations and wind tunnel model tests



C_X Longitudinal Force

C_Y Transverse Force

Figure 11. Wind load coefficients as functions of the free stream wind direction on the ferries M/F "Povl Anker" and M/F "Jens Kofoed" determined by full-scale measurements and wind tunnel model tests