Possible biases in wind speed measurements from merchant ships

B. I. Moat and M. J. Yelland
James Rennell Division for Ocean Circulation and Climate, Southampton Oceanography Centre, UK.

A. F. Molland
School of Engineering Sciences, Ship Science, University of Southampton, UK.

ABSTRACT: Wind speed measurements obtained from ship-mounted anemometers are biased by the presence of the ship which distorts the airflow to the anemometer. Until recently this bias had only been quantified for a few well-exposed anemometer sites on individual research ships, whereas the magnitude and even the sign of the bias was unknown for anemometers on merchant ships. Three-dimensional numerical simulations of the airflow over a typical tanker/bulk carrier have been performed to quantify the pattern of the airflow above the ship's bridge. The accuracy of the numerical simulations has been verified by comparison to wind tunnel studies. Typically, the flow is accelerated by up to 18±6 % or decelerated by 100% depending on position. In practice, an anemometer located above the bridge should be mounted as high and as far forwards as possible.

1.0 INTRODUCTION

A proportion of merchant ships routinely report meteorological parameters at the ocean surface as part of the Voluntary Observing Ship (VOS) programme. These observations include wind speed and direction, air and sea surface temperature, cloud cover and sea state. Over many years these measurements have been collected together to form a large database known as the International Comprehensive Ocean Atmosphere Data Set (ICOADS, [1]). VOS observations are used both in numerical weather prediction and in climate studies.

Anemometer wind speed measurements are made from fixed anemometers located at the bow of the ship or more generally from a mast on top of the bridge. It has long been suspected that wind speed measurements from these anemometers may be affected by the presence of the ship distorting the flow of air [2]. Until now, the sign and magnitude of the possible bias in these measurements has not been quantified.

Previous work to determine the error due to the airflow distortion caused by a ship's structure has mainly been concerned with research ships [3 to 7]. Research ships generally have streamlined shapes and the anemometers are located in well-exposed locations. Nevertheless, wind speed biases in the order of 10 % are possible.

Computational Fluid Dynamics (CFD) has been used to study the smoke behaviour from funnels on merchant ships [8, 9] and to reproduce the flow conditions around the deck of a frigate for landing helicopters [10]. However, little work has been undertaken to examine the possible biases in the wind speed measurements from anemometers located on merchant ships. This is because it is not possible to simulate the airflow past every individual ship and correct the anemometer measurement for the effects of flow distortion.

This paper defines a generic merchant ship shape (Section 2.0) and presents the results of a CFD study of the airflow over the resulting bluff body geometry (Section 3.0). The accuracy of the simulations is verified by comparing a CFD simulation of the flow over a bluff body to the wind tunnel study of Moat *et al.* [11] (Section 4.0). Recommendations for locating anemometers above the bridge are made in Section 5.0.

2.0 GENERIC SHIP MODELS

Principal dimensions were taken from a total of forty-four tanker and bulk carriers [12]. The general shape of the two types of ship are similar so linear regressions were fitted to the combined dimensions of both ships against the ship length overall. The resulting equations were used to create a generic bluff body representation of a tanker with an overall length of 170 m (Figure 1). The geometry was scaled by 1:46 to agree with the model size in the wind tunnel investigation of [11]. The scaled dimensions are shown in Table 1.

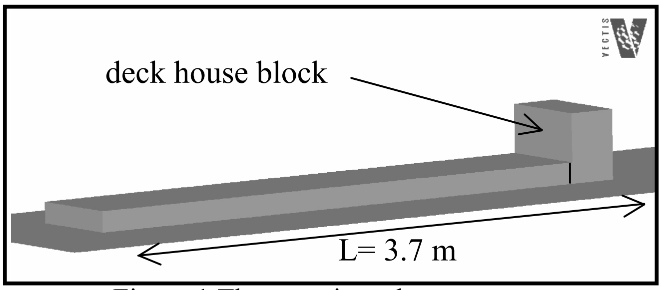

Figure 1 The generic tanker geometry.

Bridge to deck (m)	Bridge to sea (m)	Bridge length, (m)	Freeboard (m)	Breadth (m)
0.294	0.422	0.294	0.128	0.595

Table 1 : Dimension of a scaled tanker/bulk carrier of length 3.7 m.

In addition, the flow above the deck house block was simulated for comparison with [11]. The deck house block is indicated in Figure 1 and is 0.422 m high, 0.595 m wide and 0.294 m in length.

3.0 CFD INVESTIGATIONS

The CFD investigation was carried out using the VECTIS software package [13], which solves the Reynolds Averaged Navier-Stokes equations. The code was run with the RNG $k \sim \varepsilon$ turbulence model.

Simulations of the 3-dimensional airflow over the tanker and deck house block were performed for flows normal to the geometries, i.e. directly over the bow of the ship. All results presented here are from the centreline of the ship and have been normalised by the wind speed profile at the measurement location with no model present, i.e. the free stream wind profile.

The scaled tanker geometry was 3.7 m in length, L. It was placed in the centre of a flow domain of overall length 2.68L, width of 5L and height of 2.7L. The flow over the models was investigated using a uniform inlet wind speed profile of 7 ms^{-1}, leading to a nominal Reynolds number based on the bridge length of 1.4×10^5.

The simulations were performed on a non-uniform Cartesian grid. The number of cells can be increased in specific areas of interest, such as the bridge. At large distances from the ship, where the flow does not vary a great deal, the number of cells are minimised. The number of cells in the computational domain varied between 485,000 and 700,000.

For bluff body flows Moat [14] performed a sensitivity study of VECTIS to determine the dependence of the solution on the mesh density, the turbulence closure scheme and the shape of the upstream wind speed profile. The findings of [14] were applied to this investigation and resulted in an effectively mesh independent solution with variations in wind speed of 4 % or less.

The general flow pattern above the bridge of the tanker is shown in Figure 2. The mean flow direction is from left to right.

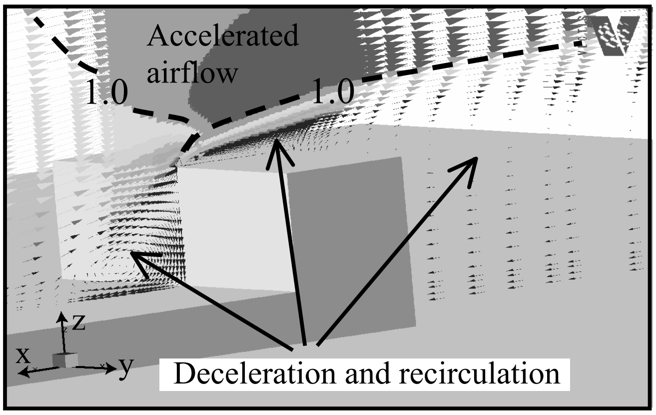

Figure 2 Normalised flow field above the bridge of the tanker.

A standing vortex is produced in front of the bridge and there is flow separation at the upwind leading edge. Close to the top of the bridge the airflow is decelerated and a flow counter to the mean flow direction is present. The depth of the decelerated region increases with distance back from the front edge of the bridge. Above the decelerated region is a line of equality where the wind speed is equal to the free stream wind speed (normalised wind speed of 1.0). Above the line of equality the wind speed is accelerated and then decreases with increase in height.

Normalised wind speed profiles at a scaled distance of x/H=0.5 back from the leading edge for both geometries are shown in Figure 3. In the case of the tanker, H is the height of the bridge top to the deck (0.294 m). For the deck house block alone, H is the height of the block from the surface (0.422 m).

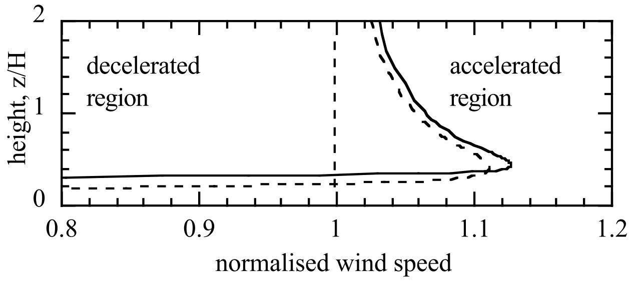

Figure 3 Normalised wind speed profiles above the tanker (dashed line) and the deck house block (solid line).

The thickness of the decelerated region is approximately the same for the tanker and deck house block. In addition, the scaled vertical positions of the wind speed maxima for the tanker and the deck house block agree well. This suggests that the flow pattern above the geometries scales with H.

The variation in the magnitude of the wind speed maximum with scaled distance from the upwind leading edge of the bridge is shown in Figure 4.

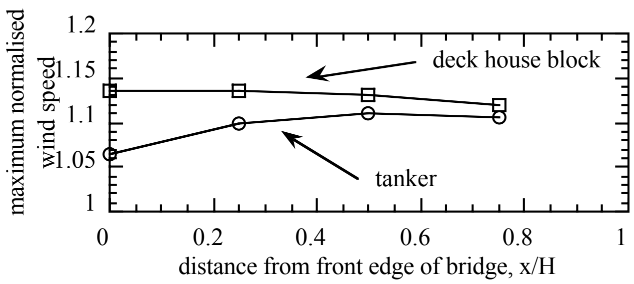

Figure 4 The CFD predicted wind speed maximum.

The wind speed maximum for the tanker increases with distance back from the upwind leading edge and reach a maximum of 1.11 at a distance of x/H=0.5. It then reduced to 1.05 at a distance of x/H=0.75. The wind speed magnitude for the deck house block was 1.14 at the upwind leading edge and decreased slowly to 1.12 at a distance of x/H=0.75.

There is good agreement in the flow between the tanker and deck house geometries.

4.0 VERIFICTION OF CFD CODE

A comparison of the CFD simulation of the flow over the deckhouse block with wind tunnel results of [11] is presented.

The normalised wind speed profile at a scaled distance of x/H=0.5 is compared to a normalised wind speed profile measured above the same geometry in a wind tunnel (Figure 5). The error bars indicate the standard error.

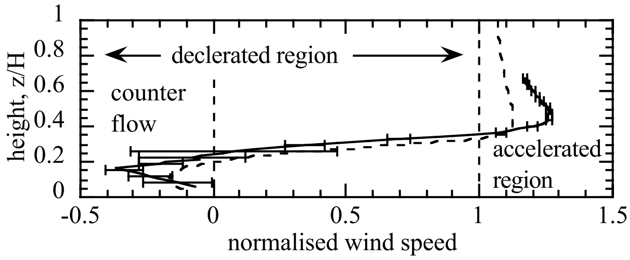

Figure 5 Comparison of normalised wind speed profiles simulated by CFD (dashed line) and measured in the wind tunnel (solid line).

There is very good agreement in the thickness of the decelerated region and the CFD code has correctly predicted a flow counter to the mean flow. The scaled vertical positions of the wind speed maxima for the wind tunnel and CFD agree well. The magnitude of the wind speed maxima for the wind tunnel and CFD were 1.26 and 1.13 respectively. Some of this difference is explained by a residual bias in the wind tunnel wind speed data of 4 % [11] and the CFD simulations may have possible variations of up to 4 % in wind speed [14].

This comparison suggests that the CFD is also underestimating the acceleration of the flow above the tanker and the best estimate of the wind speed maximum is therefore 18±6 % compared to the free stream wind speed.

5.0 LOCATING ANEMOMETERS ON SHIPS

Anemometers located above the bridge of merchant ships should be placed as far forwards as possible and as high as possible; ideally, above the front edge of the bridge (Figure 6).

If an anemometer cannot be located at the front edge of the bridge it should at least be located above a height of z/H=0.3 to measure wind speeds outside the decelerated region. An anemometer should not be placed close to the line of equality (normalised wind speed of 1.0), as high velocity gradients are present in this region (Figure 6).

Canadian coast guard vessels have already made use of these findings [15].

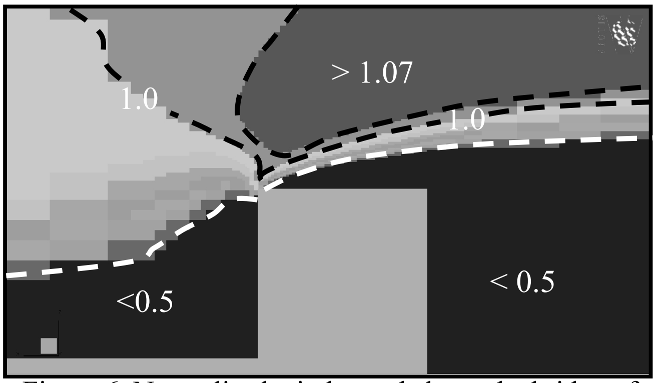

Figure 6 Normalised wind speed above the bridge of the tanker.

6.0 CONCLUSIONS

1) The flow field above the tanker scales with the height of the bridge top to deck, H. Therefore, the results of this paper could be applied to any size of tanker as long as the height, H, is known.

2) Close to the top of the bridge of the tanker the wind speed was severely decelerated and reversed in direction. Above the decelerated region the wind speed was accelerated by up to 11 % compared to the free stream wind speed.

3) The CFD model results compared very well with the wind tunnel measurements in the general flow pattern. The CFD results underestimated the magnitude of the wind speed maximum and suggest that the maximum wind speed bias for the tanker may be 18±6 %.

4) Anemometers located above the bridge of merchant ships should be placed as far forwards as possible and as high as possible; ideally, above the front edge of the bridge.

7.0 ACKNOWLEDGEMENTS

The authors would like to thank Val Swail (Meteorological Service of Canada) for partial funding throughout this project.

8.0 REFERENCES

[1] H. Diaz, C. Folland, T. Manabe, D. Parker, R. Reynolds, and S. Woodruff, Workshop on Advances in the Use of Historical Marine Climate Data. WMO Bulletin, 51 (2002) 377-380.

[2] F. W. Dobson, Review of reference height for and averaging time of surface wind measurements at sea. World Meteorological Organisation, Marine Meteorology and Related Oceanographic Activities Report No. 3, 1981.

[3] K. K. Kahma, and M. Leppäranta, On errors in wind speed observations on R/V Aranda, Geophysica, 17 (1-2) (1981) 155-165.

[4] T. V. Blanc, Superstructure flow distortion corrections for wind speed and direction measurements made from Tarwa class (LHA1-LHA5) ships. NL Report 9005, Naval Research Laboratory, Washington D. C. 1986.

[5] D. Surry, R. T. Edey and I. S. Murley, Speed and direction corrections factors for ship borne anemometers. Engineering Science Research Report BLWT-SS9-89, University of Western Ontario, London, ON, Canada, 1989.

[6] M. J. Yelland, B. I. Moat, P. K. Taylor, R. W. Pascal, J. Hutchings, and V. C. Cornell, Wind stress measurements from the open ocean corrected for airflow distortion by the ship. Journal of Physical Oceanography, 28 (7) (1998) 1511-1526.

[7] M. J. Yelland, B. I. Moat, R. W. Pascal and D. I. Berry, CFD model estimates of the airflow over research ships and the impact on momentum flux measurements. Journal of Atmospheric and Oceanic Technology, 19 (10) (2002) 1477-1499.

[8] O. E. Moctar, and V. Bertram, Computation of viscous flow around fast ship superstructures. 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan (2002) 68-77.

[9] E. Jin, J. Yoon, and Y. Kim, CFD based parametric study of the smoke behaviour of a typical merchant ship. Practical design of ships and other floating structures. Y-S Wu, W-C Cui, and G-J Zhou (Ed.), Elsevier Science Ltd. (2001) 459-465.

[10] T. C. Tai and D. Carico, Simulation of DD-963 ship airwale by Navier-stokes method. Journal of Aircraft. 32 (6) (1995) 1399-1401.

[11] B. I. Moat, A. F. Molland and M. J. Yelland, A Wind tunnel study of the mean airflow around a simple representation of a merchant ship. SOC Research and Consultancy Report No. 87, Southampton Oceanography Centre, Southampton, UK, 21 pp.

[12] RINA Ltd., Significant ships. The Royal Institution of Naval Architects, London, United Kingdom. 1990-1993.

[13] Ricardo, VECTIS Computational Fluid Dynamics (Release 3.5) User manual, Ricardo Consulting Engineers Ltd., Shoreham-by-Sea, United Kingdom, 2001.

[14] B. I. Moat, Quantifying the effect of airflow distortion on anemometer wind speed measurements from research ships. PhD. Thesis. School of Engineering Sciences, University of Southampton. UK, 2003.

[15] R. Fordyce, Personal communication. Marine Data Unit, Meteorological Service of Canada. 2001.