Experimental and Numerical Study of the Turbulence Characteristics of Airflow around a Research Vessel

STÉPHANE POPINET, MURRAY SMITH, AND CRAIG STEVENS

National Institute of Water and Atmospheric Research, Kilbirnie, Wellington, New Zealand

(Manuscript received 10 October 2003, in final form 8 April 2004)

ABSTRACT

Airflow distortion by research vessels has been shown to significantly affect micrometeorological measurements. This study uses an efficient time-dependent large-eddy simulation numerical technique to investigate the effect of the R/V *Tangaroa* on the characteristics of the mean airflow and the turbulent wake. Detailed comparison is given between the numerical results and an extensive experimental dataset. The study is performed for the whole range of relative wind directions and for instruments located in regions of high and low flow distortion. The experimental data show that both the normalized wind speed and normalized standard deviation are only weakly dependent on wind speed, ship speed, ship motion, and sea state, but strongly dependent on relative wind direction. Very good agreement is obtained between the experimental and numerical data for the mean flow, standard deviation, and turbulence spectra in the wake, even in areas of strong turbulence.

1. Introduction

Interactions between the atmosphere and ocean form an important part of the climate system. Measurement of fluxes of momentum, heat, gases, and the accurate parameterization of those fluxes, are required for global climate and weather models. Research vessels are the most common measuring platform for these flux measurements, but they are also susceptible to airflow distortion effects around the hull and superstructure. It is therefore important to understand and accurately model the influence of the sampling platform on airflow and turbulence.

Yelland et al. (1998) examined two airflow effects: (i) the acceleration/deceleration of flow and (ii) the tilt of the flow that results in air with turbulent characteristics from a different height being measured. These effects influence the drag coefficient for neutral atmospheric stability ($C_{D_{10N}}$) calculated using the inertial dissipation method. This method is preferred at sea since it uses frequencies in the inertial subrange of turbulence that are well above the frequencies associated with the wave-induced platform motion. The drag coefficient is given by

$$C_{D_{10N}} = \frac{u_*^2}{U_{10N}^2} = \frac{(kz\epsilon)^{2/3}}{U_{10N}^2},$$

where u_* is the wind stress, k is the von Kármán con-

stant, z is the height at which the air originated, ϵ is the turbulent dissipation rate (determined from spectra of wind fluctuations), and U_{10N} is the wind speed at 10-m height under neutral stability conditions (Geernaert and Plant 1990, chapter 5). The bias in C_D arises from U_{10N}^2 but also more weakly from $z^{2/3}$ through uplifting of airflow. Yelland et al. (1998) showed that the resulting bias on the drag coefficient could be as large as 60%. They also concluded that the azimuthal dependence of flow distortion could explain much of the open ocean variation of wind stress between experiments that had formerly been attributed to a wave-age effect.

Various attempts have been made to model the airflow distortion effects using both physical and numerical models. Thiebaux (1990) used wind tunnel tests to show that flow over Canadian research ships was accelerated by 7% at the position of the ship's anemometer above the bridge. Brut et al. (2002) performed scale model simulations in a water flume at various static angles of pitch and azimuthal wind direction. In particular they showed that a heavily instrumented mast can have strong effects on the measured values.

However, physical models have their limitations. While they can be used effectively to describe mean flow distortion, in order to model the distortion effects on turbulent fluxes, the turbulent length scale must also be scaled. This has proved difficult to achieve (Oost et al. 1994).

Potential flow estimates of flow distortion have been used by Kahma and Lepparanta (1981) and Oost et al. (1994). The first three-dimensional computational fluid dynamics (CFD) modeling was carried out by Yelland

Corresponding author address: Dr. Stéphane Popinet, National Institute of Water and Atmospheric Research, P.O. Box 14-901, Kilbirnie, Wellington, New Zealand. E-mail: s.popinet@niwa.co.nz

^{© 2004} American Meteorological Society



FIG. 1. A CAD representation of the R/V *Tangaroa* used in the present simulations. Locations of the validation instruments are also marked.

et al. (1998) for RSS *Discovery* and RSS *Charles Darwin*, using the commercially available package Vectis. A single bow-on flow direction was examined. Yelland et al. (2002) extended this work with simulated flows at five different angles to the bow, in the range -30° to $+30^{\circ}$. Bow-on flow simulations were also carried out for several other research ships. The main conclusion of their work was that model-derived corrections for mean flow distortion and vertical displacement of flow are essential for the calculation of $C_{D_{10N}}$ to avoid biases greater than 20%. The study considered mean flow properties only, not the turbulence properties.

Recently, Dupuis et al. (2003) presented CFD model results for *L'Atalante* using the Fluent 5 numerical model at a range of six azimuth angles from 0° to 180° of the bow. As with Yelland et al. (2002), they found that the wind speed errors were independent of wind speed. Dupuis et al. also found no significant difference between using a $k-\varepsilon$ turbulent boundary layer and laminar flow, which allowed a considerable saving in computation time.

While these studies have examined the distortion of the time-averaged flow, the effects on the turbulent structures are largely unknown (Oost et al. 1994). Questions remain about the distortion of turbulence by the mean flow disturbance, phase angles between u and w(Oost et al. 1994; Pedreros et al. 2003), and generation of turbulent vortices by superstructures and anemometer support structures.

Our aims here are the following:

- to develop a robust, efficient CFD model that is able to give access to the time-dependent turbulent characteristics of the flow;
- to evaluate the performance of this model for mean airflow distortion effects against shipborne measure-



FIG. 2. (top) Side and (bottom) top views of the (right) Campbell 1 and (left) Campbell 2 instruments.

ments with an emphasis on areas of high flow distortion (turbulent wakes and recirculation zones); and to examine initial CFD results of time-dependent turbulence generated by flow interaction with the ship superstructure.

In section 2 we describe the monitoring during the experimental voyage. Section 3 gives an overview of the main features of the large-eddy simulation (LES) numerical technique we developed. Section 4 presents a summary and analysis of the experimental data, and section 5 includes a detailed comparison of the experimental data with numerical simulations spanning the whole range of relative wind directions (from bow-on to stern-on) for both mean and turbulent flow properties.

2. Experimental setup

Eight Vector cup/vane anemometers and two robust Weathertronics 3D propeller anemometers were installed on R/V *Tangaroa*. Figures 1, 2, and 3 and Table 1 illustrate the location of the instruments. Several instruments (Campbell 1 3D prop, Campbell 2 3D prop, Starlogger 3) are located in sites that are typically used as permanent sampling sites. The other instruments are deliberately located in areas of flow that are likely to be strongly affected by the ship (in front of and behind the central superstructure).

For practical reasons, several instruments are also mounted on short booms and thus lie relatively close to the ship. In particular, we expect Starlogger 2, 3, and 4 to sample the strong velocity gradients caused by flow



Starlogger 1







Starlogger 6





r 6 Campbell 1 3D prop, cup Campbell 2 3D prop FIG. 3. Individual mountings of instruments marked in Fig. 1.

TABLE 1	Elevation	above	sea	level	of	the	instruments
IADLL I.	Licvation	abbve	sou	10,001	O1	unc	moutumento

Instrument	Elev (m)				
Starlogger 1					
cup/vane	13.8				
Starlogger 2					
cup/vane	18.4				
Starlogger 3					
cup/vane	19.4				
Starlogger 4					
cup/vane	17.4				
Starlogger 5					
cup/vane	15.6				
Starlogger 6					
cup/vane	8.7				
Campbell 1					
cup/vane	11.5				
3D prop	14.4				
Campbell 2					
cup/vane	19.6				
3D prop	19.9				

separation above the central superstructure. Together with instruments located in the turbulent wake of the central superstructure (Starlogger 5 and 6), this will provide a stringent test of the numerical method. All the cup/vane anemometers are sampled every 3 s, while the 3D propellers are sampled at 4 Hz. The propeller anemometers were calibrated after the experiment and agreed to within 1% of their factory calibration, which is less than the error arising from the departure from an ideal cosine response. Within 20° of the bow this error is less than 3% but increases to a maximum of 12% at 40°. The Vector anemometers have a manufacturer accuracy of 1% at the wind speeds here. Both anemometers are subject to unquantified errors due to ship pitching and rolling.

3. Numerical method

Most CFD programs used for engineering applications provide a solution of the Reynolds-averaged Navier–Stokes equations (RANS) whereby the averaging is carried out in space and time. The solution obtained is thus a stationary, time-averaged representation of the flow and provides only limited information on the turbulence characteristics. Another possibility is to carry out the averaging only spatially. The resulting timedependent solution is then obtained using methods usu-



FIG. 4. Adaptive mesh used for a side-on wind flow. The horizontal and vertical cross sections illustrate the three-dimensional adaptive octree. The computational volumes are colored according to the norm of the local velocity.

ally referred to as large-eddy simulations (Herring 1979; Rogallo and Moin 1984; Lesieur 1990). While these methods can be more computationally expensive, they require fewer assumptions for modeling turbulent stresses and have the potential to provide better solutions, particularly in wakes or recirculating regions (Shah and Ferziger 1997; Rodi et al. 1997) or around the bluff bodies we are interested in (Baetke et al. 1990; Murakami 1993).

The numerical method we used is described in detail in Popinet (2003). Its implementation, called Gerris, is freely available (Popinet 2002). In the following we summarize the main characteristics of the technique.

a. Spatial discretization

The computational domain is discretized using collocated cubic finite volumes organized as a spatial octree (Samet 1989; Khokhlov 1998). This type of discretization is very flexible and allows the spatial resolution to dynamically adapt to follow the evolving flow structures (Popinet 2003). An example of such a discretization is given in Fig. 4. The wake created by the ship for a side-on wind flow is resolved using the finest mesh. Far from the ship, only large structures are present and the spatial resolution decreases accordingly. The mesh is adapted at each time step to follow the evolving turbulent boundary of the wake.

Various choices are possible for the refinement criterion. We use a simple criterion based on the norm of the local vorticity vector. Specifically, a cell is refined whenever

$$rac{h \| oldsymbol{
abla} imes oldsymbol{U} \|}{\max \| oldsymbol{U} \|} > au$$

where $\max \|\mathbf{U}\|$ is evaluated over the entire domain and h is the size of the cell. The threshold value τ can be interpreted as the maximum acceptable angular deviation (caused by the local vorticity) of a particle traveling at speed max $\|\mathbf{U}\|$ across the cell.

This adaptation mechanism allows substantial savings in computation time. Fine meshes can thus be used to resolve the ship geometry and the small turbulent structures it creates. A drawback is that, contrary to traditional unstructured finite-element/finite-volume techniques, the boundary of the discretized volumes cannot be made to correspond with complex boundaries. This problem can be solved by using "cut cell" techniques (Quirk 1994; Almgren et al. 1997; Ye et al. 1999), which take into account the exact shape of finite volumes cut by the solid boundary. When implemented properly, these techniques have the added advantage of allowing simple automatic mesh generation independently of the complexity of the solid boundaries considered.

b. Temporal discretization

 ∇

We consider a constant-density, incompressible, and inviscid fluid. Given a velocity field

$$\mathbf{U}(x, y, z, t) = [u(x, y, z, t), v(x, y, z, t), w(x, y, z, t)],$$

and a pressure field p = p(x, y, z, t) defined at location (x, y, z) and time *t*, on some domain Ω with a solid wall boundary $\partial \Omega$, the incompressible Euler evolution equations for **U** are

$$\mathbf{U}_t = -u\mathbf{U}_x - v\mathbf{U}_y - w\mathbf{U}_z - \nabla_p,$$

$$\cdot \mathbf{U} = 0.$$

The boundary condition for the velocity at solid wall boundaries is the no-flow condition:

$$\mathbf{U}(x, y, z, t) \cdot \mathbf{n} = 0 \qquad \text{for } (x, y, z) \in \partial\Omega,$$

where **n** is the outward unit vector on $\partial \Omega$.

We use a classical fractional-step projection method (Chorin 1968; Peyret and Taylor 1983; Brown et al. 2001). At any given time step n, we assume that the velocity at time n, \mathbf{U}^n , and the fractional step pressure $p^{n-1/2}$ are known at cell centers. In a first step, a provisional value \mathbf{U}^{**} is computed using

$$\frac{\mathbf{U}^{**} - \mathbf{U}^n}{\Delta t} = -\mathbf{A}^{n+1/2},\tag{1}$$

where $\mathbf{A}^{n+1/2}$ is an approximation to the advection term $[(\mathbf{U} \cdot \nabla)\mathbf{U}]^{n+1/2}$. The new velocity \mathbf{U}^{n+1} is then computed by applying an approximate projection operator to \mathbf{U}^{**} , which also yields the fractional step pressure $p^{n+1/2}$ (Almgren et al. 2000).

The advection term $\mathbf{A}^{n+1/2}$ is computed using a second-order, unconditionally stable, Godunov-type scheme (Bell et al. 1989), with a cell-merging technique for small cut cells (Quirk 1994). The overall scheme is thus second-order in space and time.

c. Poisson equation

The projection method relies on the Hodge decomposition of the velocity field as

$$\mathbf{U}^{**} = \mathbf{U} + \nabla \phi, \qquad (2)$$

where

$$\nabla \cdot \mathbf{U} = 0$$
 on Ω and $\mathbf{U} \cdot \mathbf{n} = 0$ on $\partial \Omega$. (3)

Taking the divergence of (2) yields the Poisson equation

$$\nabla^2 \phi = \nabla \cdot \mathbf{U}^{**},\tag{4}$$

while the normal component of (3) yields the boundary condition

$$\frac{\partial \phi}{\partial \mathbf{n}} = \mathbf{U}^{**} \cdot \mathbf{n} \quad \text{on } \partial \Omega.$$

The divergence-free velocity field is then defined as

$$\mathbf{U}^{n+1} = \mathbf{U}^{**} - \boldsymbol{\nabla}\boldsymbol{\phi}.$$

where ϕ is obtained as the solution of the Poisson problem (4). This defines the *projection* of the velocity U^{**} onto the space of divergence-free velocity fields.

This projection step is the most expensive part of the solution algorithm because Eq. (4) results in a spatially implicit problem (i.e., a linear system of equations for each discrete volume). We use an efficient multigrid-accelerated relaxation technique that combines naturally with the octree spatial discretisation (Popinet 2003).



FIG. 5. Track of the ship position.

d. Turbulence modeling

Given the very high Reynolds number of a typical airflow around a ship ($\Re \approx 10^8$), direct numerical simulations are not feasible: the scale of the smallest possible structures (the Kolmogorov scale) being of the order of $1/\Re$. Some turbulence modeling is thus necessary to approximate the energy transfer at scales smaller than the mesh size. In LESs this subgrid energy transfer is usually assumed to take the form of a subgrid turbulent viscous stress where the viscosity coefficient is variable both in space and time and described using semiempirical relationships (Lesieur and Métais 1996).

As described above, the numerical model we use does not contain any explicit viscous terms. In practice, numerical schemes always have some numerical viscosity due to higher-order errors associated with the discrete representation of the solution. Remarkably, several authors (Boris et al. 1992; Porter et al. 1994) have shown that this numerical dissipation can describe turbulent subgrid energy transfer as well or sometimes better than more complex LES semiempirical models. The advection scheme we use (Bell et al. 1989) has been shown by Rider (1995) to have similar dissipation properties. Consequently, this first study will not use any explicit turbulent dissipation, while we certainly do intend to investigate more complex LES models in the future.

4. Experimental results

Data were collected continuously during a weeklong cruise in the Pacific Ocean, southeast of the New Zealand mainland, during March 2002 (see Fig. 5). In the rest of the text we refer to the wind speed as seen by an observer moving with the ship as the *relative wind speed*. The *relative wind direction* is defined as the angle between the wind vector as seen by an observer moving with the ship and the longitudinal axis of the ship. A relative wind direction of 0° corresponds to a bow-on wind, while a 90° relative wind direction corresponds to a wind coming directly from starboard. A range of



FIG. 6. (a) Measured relative wind speed at the Campbell 1 3D prop location. (b) Ship speed. (c) Distribution of relative wind directions. (d) Distribution of wind speeds.

relative wind speeds, up to 20 m s⁻¹, was sampled with sea conditions and ship motion varying accordingly (from calm seas to up to 6–7-m swells). Figure 6a illustrates the variability of the measured relative wind speed at location "Campbell 1 3D prop" during the whole window of observation we consider in this study. The corresponding ship speed is given in Fig. 6b. The data sampled are roughly evenly distributed between periods where the ship was stationary (while carrying out maintenance work on moorings at locations NBM and SBM) and periods with a cruising speed of around 6 m s⁻¹.

In order to obtain a synthetic representation of all the data collected, we made two assumptions.

- The wind speeds measured at the different locations should scale linearly with some reference velocity; that is, the fluid flow is essentially independent from the Reynolds number.
- 2) This normalized velocity depends only on the relative wind direction.

The first assumption is justified as the Reynolds number is very high ($\approx 10^8$ based on ship length and a 10 m s⁻¹ wind speed), well within the asymptotic regime for flow around a solid obstacle. The second assumption is much stronger in that we have chosen to neglect the influence of sea conditions as well as ship motion. More specifically, we chose not to take into account the difference in vertical (apparent) wind profiles as seen from a moving or stationary ship.

To apply these assumptions to the data, we need to choose a reference velocity. Ideally one would have access to some reference measurement away from the ship. As we do not have such reference, we chose to use both the Campbell 1 3D propeller data and the Campbell 2 cup anemometer data. Being farthest away from the ship superstructure, these instruments are the best possible approximation to a reference measurement. This is of course not the case when either of them lie directly in the wake of the central superstructure. We avoid this problem by taking the Campbell 1 site as reference for all relative wind directions in $[-100^\circ, +100^\circ]$ and Campbell 2 otherwise. The relative wind direction is defined in the same way.

Figure 7 illustrates the dependence of the relative wind speed measured at each location on the relative wind direction. Each dot is a 1-min average of the experimental time series. All the data for the time window considered (17–22 March 2002) are represented (around 6000 samples for each graph). A very clear relationship is obtained for all the measurement locations, the standard deviations being of the order of 5% of the reference



FIG. 7. Normalized relative wind speeds measured at each location as functions of relative wind direction (reference Campbell 1 3D prop/ Campbell 2 cup). Each dot is a 1-min average. The vertical dotted lines indicate whether Campbell 1 3D prop or Campbell 2 cup is used as reference.

velocity. This confirms that the assumptions we made are justified despite the wide range of sea conditions, and ship and wind speeds. Regarding the structure of the variations observed, a first general observation is that we expect the relationships to be symmetrical around 180° for instruments close to the centerline of the ship (the ship being roughly right/left symmetrical). This is indeed the case for the Starlogger 2 and Campbell 1 cup instruments, while others show varying degrees of asymmetry.

A number of distinctive features can be seen that are easy to relate to the geometry of the ship. The strong variations in velocities near 90° and 270° for Starlogger 1, 2, 3, 4, and 5 can be linked directly to the flow upstream of the instruments moving behind local obstructions (mainly from the center deck). Similarly, the strong decrease in velocity near 180° for Campbell 1 cup can be associated with the instrument being in the wake of the central superstructure. The structure of the dependence for most of the instruments can be characterized by two regimes:

1) a more or less "laminar" regime, wherein the in-

strument sits in a relatively undisturbed flow (possibly with some potential flow acceleration), for example, Starlogger 1 and 2 between 0° and 75° , Starlogger 3 and 4 between 0° and 180° ; and

 a strongly turbulent regime, wherein the instrument sits downstream of an obstruction, for example, Starlogger 1 and 2 above 90°, Starlogger 3 and 4 between 180° and 270°.

Starlogger 6 is almost always downstream of some obstruction so that only the turbulent regime is present.

Some features of Campbell 1 cup deserve particular attention. This instrument is located immediately below Campbell 1 3D prop, used as reference for all "bow on" winds. One would thus expect a nearly constant relationship for all bow-on angles, but a sharp decrease is observed around 15° as well as other well-defined structures near 270°. Apart from systematic instrumental errors (improbable given the well-defined features and wide range of wind speeds), a likely explanation is the influence on the measured wind speed of small-scale details like mast mounting and fittings (Gill et al. 1967; Barthlott and Fiedler 2003). Figure 2 shows that for a relative wind direction of 30° , Campbell 1 cup lies directly in the wake of the vane. Although this does not correspond exactly to the 15° suggested by Fig. 7, the difference may be due to misalignment of the reference anemometer.

5. Comparison with numerical simulations

a. Numerical setup

Using the numerical method previously described we performed a number of simulations of the flow around a CAD model of R/V Tangaroa (Fig. 1) for different relative wind directions. We aimed for a spatial resolution near the ship of around 50 cm. The smallest details represented in the CAD model are consistently of this order. In order to minimize the influence of the boundary conditions, the CAD model is positioned in a cubic domain 276 m wide (4 times the ship length). The corresponding maximum blockage ratio obtained for beam-on flows is of the order of 1%. A constant, unity inflow velocity is imposed to the left-hand side of the domain, simple outflow conditions to the righthand side, and slip conditions on all the other boundaries (including the sea surface). As in Dupuis et al. (2003), we chose not to impose a more complex velocity profile (logarithmic boundary layer or other models) at the inflow for two reasons.

- We solve the Euler equations and thus cannot impose the explicit dissipative terms consistent with a nonzero stress at the sea surface, and
- the experimental results have shown that the flow distortion is largely independent of the ship motion and thus of the detail of the vertical wind velocity profile.

The simulations are all started with the potential flow solution as initial conditions. As time passes, the vorticity generated at the solid boundaries (essentially near sharp features of the CAD model) is advected away from the ship and evolves into a fully developed turbulent wake. The computational mesh is adapted dynamically to follow this evolution using the vorticity criterion (Fig. 4). We chose to refine the mesh in areas of high vorticity down to a spatial scale of 1 m (50 cm if close to the ship). Depending on the relative wind direction, between 200 000 and 350 000 grid points were necessary to resolve the fully developed turbulent wakes.

Typical results are illustrated in Figs. 8 and 9 for bowon and stern-on flows, respectively. The pictures are a snapshot in time of the fully developed wake. The stream ribbons (streamlines twisted according to the local vorticity vector) pictured go through individual instrument locations. In Fig. 8 two regimes can be clearly distinguished: a laminar flow upstream of the central superstructure and a strongly turbulent flow downstream. The signature of this turbulent wake is clearly seen in the large fluctuation in wind velocity near sea level (colored plane), while the near-potential flow solution upstream creates the characteristic low wind zone just upstream of the bow.

Since our model is time dependent, it was necessary to select a window over which to time average the numerical fields in order to compare the numerical results to the time-averaged experimental data of Fig. 7. We chose to stop the simulations at $t^* = tU/L = 3$, where U is the inflow velocity and L = 276 m is the domain size, and to time average the fields for $t^* \in [1, 3]$. One t^* unit was enough in all cases to obtain a fully developed turbulent regime from the initial potential solution.

b. Mean flow distortion

A series of simulations was performed with a relative wind direction varying from 0° to 360° by increments of 15°. Each simulation took approximately 20 h of CPU time on a 2-GHz compatible PC. The results for the time-averaged relative wind speeds calculated at each instrument location are pictured in Fig. 10, together with the experimental data. For clarity, the experimental data of Fig. 7 is summarized here by the two curves: mean plus or minus standard deviation.

When looking more closely at the results, it is useful to distinguish the laminar and turbulent regimes. Very good agreement between the simulations and the experimental data is obtained in the laminar regime: 0°-90° for Starlogger 1 and 2 and 0°-180° for Starlogger 3 and 4. In the turbulent regime, good agreement is still obtained: correct low values for Starlogger 1 between 90° and 270°, "M shaped" structure between 100° and 260° for Starlogger 2, and sharp gradients for Starlogger 3, 4, and 5. As noted earlier, Starlogger 6 is a difficult case, being located on the lower deck and always in a turbulent regime. While the general trend is reproduced by the model, a number of small structures do not seem to match very well. A possible explanation is that several small-scale structures (winches, railings, deck crane, etc.) close to the instrument location are not represented in the CAD model. Similarly, the small structures in Campbell 1 cup and Campbell 2 cup that we attributed to local perturbations are not reproduced by the numerical model. Overall, and taking into account the variability of the experimental data, the agreement is very satisfying.

The two 3D propellers at locations Campbell 1 and Campbell 2 give an experimental measurement of the deviation of the flow from the horizontal. Figures 11a and 11b summarize the experimental and numerical results obtained at these two locations. Again, a welldefined experimental relationship is obtained. The numerical results match well with the experimental data. Strong gradients in angular deviation are well captured near 180° for Campbell 1 and near 0° for Campbell 2, while the total range of variation ([-10:10] for Campbell 1 and [-5:15] for Campbell 2) is well reproduced. The systematic shifts of $\approx +5^{\circ}$ for both locations can



FIG. 8. Streamlines passing through instruments' locations for a bow-on wind. The sea surface and the streamlines are colored according to the norm of the local wind velocity.



FIG. 9. Streamlines passing through instruments' locations for a stern-on wind. The sea surface and the streamlines are colored according to the norm of the local wind velocity. Note the much wider wake compared to Fig. 8.

be attributed to an approximate alignment of the instruments (as can the slight asymmetry in relative wind direction for the measurements at Campbell 2). One feature that does not seem to be captured by the model is the decrease in angular deviation near 180° at the Campbell 2 site (Fig. 11b). For this relative wind direction, the 3D propeller is immediately downstream of the ladder (on top of the fantail) used as support, which may explain the discrepancy.

c. Turbulence intensity

The numerical simulations also provide detailed information about the structure of the turbulent wake. The turbulence intensity can be characterized by the standard deviation of the velocity measured at each location:

$$\sigma_{v} = \sqrt{\frac{\int_{0}^{T} [v(t) - \overline{v}]^{2}}{T}},$$

where v is the norm of the velocity and \overline{v} is the temporal mean of the norm of the velocity. Figure 12 gives a summary of the normalized standard deviation (using the same reference velocities as in Fig. 10) measured and simulated at each site. The laminar and turbulent regimes described for Fig. 10 are clearly apparent in



FIG. 10. Relative wind speeds at each location as functions of relative wind direction. The experimental data are represented by the bounding curves defined by mean plus or minus standard deviation. The symbols are the results of numerical simulations. The gap in data for Campbell 2 cup corresponds to this instrument being used as reference.



FIG. 11. Angle from horizontal of the time-averaged velocity vector. The experimental data are represented by the bounding curves defined by mean plus or minus standard deviation. The symbols are the results of numerical simulations. (a) Campbell 1 3D prop site. (b) Campbell 2 3D prop site.



FIG. 12. Normalized standard deviations of the wind speeds at each location as functions of relative wind direction. The experimental data are represented by the bounding curves defined by mean plus or minus standard deviation. The symbols are the results of numerical simulations.

this graph, with low average velocities corresponding with large standard deviations.

Good agreement is obtained between the experimental data and the numerical simulations. In the laminar regimes, the numerical solution shows very little variation in the velocity field. This is expected since the numerical inflow profile is strictly constant. In contrast, the small but nearly constant experimental standard deviation obtained in these regimes is best explained as the signature of the background (undisturbed) atmospheric turbulence. It is also remarkably well defined at around 5% of the reference velocity.

We also note that the numerical solution tends to somewhat overpredict the intensity of the turbulent fluctuations. This is also consistent with the tendency to underpredict the average velocities in turbulent regime, apparent in Fig. 10. This trend could be explained by the lack of any subgrid turbulent viscosity in our model. If a turbulence model based on a subgrid turbulent viscosity was introduced, increased momentum diffusion in the turbulent regime would lead to smaller velocity fluctuations and larger average velocities. Another possible contribution could be the filtering of the highfrequency part of the energy spectrum by the instruments, although we would expect a similar filtering to occur in the numerical simulations.

Another interesting feature of Fig. 12, when examined together with Fig. 10, is the correlation of increased turbulence with the small features seen for the Campbell 1 cup and Campbell 2 cup sites (near 15° and 75° , and 180° , respectively). This tends to confirm our hypothesis that these local average velocity variations are caused by small-scale upstream obstructions. The signature of a similar upstream obstruction, though of larger spatial extend, is clearly seen near 180° for the Starlogger 6 location and is reproduced by the numerical simulation. It corresponds to one of the legs of the fantail moving upstream of the instrument location. This is clearly illustrated by the complex shape of the streamline going through this location for a stern-on wind (Fig. 9).

d. Turbulence spectra

The standard deviation of the velocity is only an integrated measure of the turbulent energy content of the signal measured. Being time dependent, LESs give information about the detailed spatiotemporal distribution of turbulent structures down to scales comparable with



FIG. 13. Comparison of modeled downwind power spectral density of velocity (solid line) with measured spectra upwind (dotted) and downwind (dot–dash). The relative wind speed was 10 m s⁻¹ from astern.

the grid size. Figure 13 gives spectra obtained from experimental and numerical time series. The experimental spectra were calculated using 30 min of data, separated into 1024-point segments and averaged. Two experimental locations are used: the foremast 3D propeller (Campbell 1 3D prop) and the fantail propeller (Campbell 2 3D prop). These instruments are known to have high-frequency limitations that are usually modeled as a first-order system with a power spectral response attenuated by the factor $(1 + 4\pi^2 f^2 L^2/U^2)^{-1}$, where L is the distance constant of the anemometer (1 m) and U is wind speed (Horst 1973). The power spectrum is reduced to half at $f = U/(2\pi L)$, which occurs at 0.8 and 1.1 Hz for the foremast and fantail, respectively. A correction for this attenuation has been included in Fig. 13. For this comparison we deliberately chose the difficult situation of turbulent airflow in the wake of the vessel with the wind almost directly astern (165°).

It is clear that the total energy content of the measured downwind turbulent spectrum (FM) exceeds the upwind spectrum (FT) by at least a factor of 6, indicating that the turbulent wake intensity significantly exceeds the background atmospheric turbulence intensity (which is consistent with the standard deviation data of Fig. 12/ Campbell 1 cup). Above 0.25 Hz the measured spectra approximately follow a slope of -5/3, characteristic of the inertial subrange. The modeled frequency spectrum of the wake agrees very well with the measured spectrum both qualitatively and quantitatively. The modeled spectrum does show a faster falloff above 1 Hz, which we attribute to the finite spatial resolution of the model. Further model runs at higher resolution (not shown) extended the cutoff region to higher frequencies, as ex-

pected. The modeled spectrum peaks at 0.15 Hz, corresponding to a dominant eddy scale size of ≈ 66 m, which is close to the scale size of the ship length. The modeled falloff at lower frequencies indicates that no spatial structures larger than the ship length are created. This is consistent with vortex shedding occurring at scales comparable to the ship length and then decaying into smaller structures. At low frequencies (below 0.05 Hz) the upwind and downwind experimental spectra converge, indicating that at the corresponding length scales the influence of the ship on the background atmospheric turbulence is negligible. In the 0.04–0.2-Hz frequency range, ship motion (both pitching and changes in relative wind direction due to yaw) typically has a strong influence on the measured spectrum.

e. General characteristics

A strength of the numerical simulations is that they give a global picture of the flow structure, which is difficult to infer from point measurements. Three-dimensional maps characterizing various measures of flow distortion are easily obtained. As an example, Fig. 14 uses an isosurface of the time-averaged velocity at 90% of the inflow velocity to illustrate the 3D structure of the velocity field. The large pressure building up at the bow and in front of the central superstructure creates the two rounded low-velocity zones in these areas. These two features would be described by a laminar potential flow approximation. Most of the other features are linked to vorticity generation at the ship boundary and subsequent advection by the flow. Particularly noticeable features are the wake of the whole ship extending far into the domain, the wake created by the crow's nest, and a tubular structure starting near the bow and extending the whole length of the ship. Closer examination reveals that this low-velocity zone corresponds to the core of a longitudinal vortex fed by the strong vorticity generation near the bow.

Using only Fig. 14, it is difficult to gauge of the velocity fluctuations, although one might guess that the downstream wake is turbulent while the upstream part of the flow is more or less stationary. Figure 15 uses the same type of representation but for the standard deviation. A clear qualitative and quantitative picture of the strongly turbulent wake just downstream of the ship is obtained. It is interesting to note that, while the wake extends very far from the ship as seen in Fig. 14, the fluctuations tend to decrease rapidly when the distance to the ship increases. It is also seen that the bow vortices described in Fig. 14 are not associated with any significant fluctuation in velocity (i.e., they are stationary).

Figure 16 is a similar representation but for a 45° relative wind direction. A much wider turbulent wake is generated with several clearly defined subwakes linked to specific parts of the ship. Particularly interesting is the strongly turbulent bow wake.



FIG. 14. Isosurface of the time-averaged wind velocity at 90% of the inflow velocity for a bow-on flow. The velocity inside the volume pictured is lower than 90% of the inflow velocity.



FIG. 15. Isosurface of the standard deviation at 25% of the inflow velocity for a bow-on flow. The standard deviation of the velocity inside the volume pictured is larger than 25% of the inflow velocity.



FIG. 16. Isosurface of the standard deviation at 25% of the inflow velocity for a 45° wind flow. The standard deviation of the velocity inside the volume pictured is larger than 25% of the inflow velocity.



FIG. 17. Characterization of flow distortion at location Campbell 1 3D prop. (a) Vertical displacement. (b) Relative wind speed.

f. Application to micrometeorological measurements

Figures 17a and 17b illustrate a characterization of flow distortion at location Campbell 1 3D prop, which is the main location used for micrometeorological measurements on the Tangaroa. Both the full LES solution and the initial potential flow solution are given. Figure 17a gives the relative vertical displacement of a parcel of air reaching the measurement location as a function of relative wind direction. This value is computed from the numerical results by following the time-averaged streamline passing through the instrument location. The vertical displacement of 1.5 m for a bow-on flow and 6 m for a 90° relative wind direction are comparable to results obtained by Yelland et al. (1998, 2002). Figure 17b illustrates the dependence in relative wind direction of the wind speed measured relative to the inflow (exact) wind speed. The obstruction by the ship for a bow-on flow causes an underestimation of 7% of the wind speed, while for a 90° relative wind direction the wind speed is overestimated by 10%. These corrections will be applied to the determination of C_D in future work. It is interesting to note that, while the potential flow solution gives a reasonable prediction of the relative wind speed for bow-on flows, it severely underpredicts the elevation for all relative wind directions.

The experiments carried out on the *Tangaroa* have validated the ability of the Gerris CFD model to simulate both time-averaged flow and time-varying turbulent structure. We are now able to consider specific problems relating to both the generation and distortion of turbulence by flow disturbance. In particular, the region in front of the bow has been used as a gas flux profiling site in several recent experiments (e.g., McGillis et al. 2001), yet it is subject to the effects of pressure buildup, as shown in Fig. 14. We are now confident that the CFD model can be used to examine the effect of the ship on turbulent transfer at this location. This will be the subject of further study.

6. Conclusions

The experimental dataset collected as part of this study confirms that the mean flow characteristics are only weakly dependent on ship motion, ship speed, wind speed, or sea state, but strongly dependent on the relative wind direction. A new finding is that the normalized wind speed standard deviation (square root of the turbulent kinetic energy) is also well characterized as a function of relative wind direction only. The standard deviation of the background atmospheric flow measured by well-exposed instruments is consistently close to 5% of the incoming wind speed. For badly exposed instruments, located in the wake of the ship superstructure, normalized standard deviations as high as 40% can be observed. The experimental data also confirm that even quite small structural elements (such as instrument mountings) can cause significant flow distortion.

Numerical studies performed using our time-dependent LES code show a very good agreement with both experimental mean velocities and standard deviations. These results have been obtained for the whole range of relative wind directions (from bow-on to stern-on) and remain valid in zones of high turbulence and high flow distortion. We also made use of the time-dependent nature of LES to obtain turbulence spectra. They are in excellent agreement with experimental data. The adaptive mesh technique we use has proved to give fast and accurate solutions for turbulent flows. These solutions are particularly useful when a global understanding of the flow pattern is sought, for example, in order to optimize sampling location. The results also provide correction factors that can be applied to calculations of drag coefficients.

This work provides a validated basis for future studies. Although promising, the spectral analysis presented here is only preliminary and relies on a limited experimental dataset because of technical constraints (the low sampling rates and high response time of most of the anemometers used). In the future we intend to carry out a more extensive measurement campaign using sonic anemometers. From a numerical modeling perspective, the simulation of turbulent flows around bluff bodies is still very much a work in progress in need of improvements (Shah and Ferziger 1997; Rodi et al. 1997; Iacacarino et al. 2003). Finally, by providing an open source version of the code that can be freely redistributed and modified (Popinet 2002), we hope to encourage research and collaboration in this field.

REFERENCES

- Almgren, A. S., J. B. Bell, P. Colella, and T. Marthaler, 1997: A Cartesian grid projection method for the incompressible Euler equations in complex geometries. *SIAM J. Sci. Comput.*, 18, 1289–1309.
 - —, —, and W. Y. Crutchfield, 2000: Approximate projection methods: Part I. Inviscid analysis. SIAM J. Sci. Comput., 22, 1139–1159.
- Baetke, F., H. Werner, and H. Wengle, 1990: Numerical simulation of turbulent flow over surface-mounted obstacles with sharp edges and corners. J. Wind Eng. Ind. Aero., 35, 129–147.
- Barthlott, C., and F. Fiedler, 2003: Turbulence structure in the wake of a meteorological tower. *Bound.-Layer Meteor.*, 108, 175–190.
- Bell, J. B., P. Colella, and H. M. Glaz, 1989: A second-order projection method for the incompressible Navier–Stokes equations. *J. Comput. Phys.*, 85, 257–283.
- Boris, J. P., F. F. Grinstein, E. S. Oran, and R. L. Kolbe, 1992: New insights into large-eddy simulation. *Fluid Dyn. Res.*, **10**, 199– 228.
- Brown, D. L., R. Cortez, and M. L. Minion, 2001: Accurate projection methods for the incompressible Navier–Stokes equations. J. Comput. Phys., 168, 464–499.
- Brut, A., A. Butet, S. Planton, P. Durand, and G. Caniaux, 2002: Influence of the airflow distortion on air-sea flux measurements aboard research vessel: Results of physical simulations applied to the Equalant99 experiment. Preprints, 15th Conf. on Boundary Layers and Turbulence, Wageningen, Netherlands, Amer. Meteor. Soc., CD-ROM, P2.8.
- Chorin, A. J., 1968: Numerical solution of the Navier–Stokes equations. *Math. Comput.*, 22, 745–762.
- Dupuis, H., C. Guerin, D. Hauser, A. Weill, P. Nacass, W. Drennan, S. Cloché, and H. Graber, 2003: Impact of flow distortion corrections on turbulent fluxes estimated by the inertial dissipation method during the FETCH experiment on R/V L'Atalante. J. Geophys. Res., 108, 8064, doi:10.1029/2001JC001075.
- Geernaert, G. L., and W. J. Plant, Eds., 1990: Current Theory. Vol. I, Surface Waves and Fluxes, Kluwer Academic, 352 pp.
- Gill, G. C., L. E. Olson, J. S. Sela, and M. Suda, 1967: Accuracy of wind measurements on towers and stacks. *Bull. Amer. Meteor.* Soc., 48, 665–674.
- Herring, J. R., 1979: Subgrid scale modelling—An introduction and overview. *Turbulent Shear Flows I*, F. Durst et al., Eds., Springer-Verlag, 347–352.
- Horst, T. W., 1973: Corrections for response errors in a three-component propeller anemometer. J. Appl. Meteor., 12, 716–725.
- Iacacarino, G., A. Ooi, P. A. Durbin, and M. Behnia, 2003: Reynolds averaged simulation of unsteady separated flow. *Int. J. Heat Fluid Flow*, 24, 147–156.

- Kahma, K. K., and M. Lepparanta, 1981: On errors in wind speed observations on R/V Aranda. Geophysica, 17, 155–165.
- Khokhlov, A. M., 1998: Fully threaded tree algorithms for adaptive refinement fluid dynamics simulations. J. Comput. Phys., 143, 519–543.
- Lesieur, M., 1990: Turbulence in Fluids. 2d ed. Kluwer, 298 pp. —, and O. Métais, 1996: New trends in large-eddy simulations of turbulence. Annu. Rev. Fluid Mech., 28, 45–82.
- McGillis, W. R., J. B. Edson, J. D. Ware, J. W. H. Dacey, J. E. Hare, C. W. Fairall, and R. Wanninkhof, 2001: Carbon dioxide flux techniques performed during GasEx-98. *Mar. Chem.*, **75**, 267– 280.
- Murakami, S., 1993: Comparison of various turbulence models applied to a bluff body. J. Wind Eng. Ind. Aero., 46, 21–36.
- Oost, W. A., C. Fairall, J. Edson, S. Smith, R. Anderson, J. Wills, K. Katsaros, and J. DeCosmo, 1994: Flow distortion calculations and their application in HEXMAX. J. Atmos. Oceanic Technol., 11, 366–386.
- Pedreros, R., G. Dardier, H. Dupuis, H. Graber, W. Drennan, A. Weill, C. Guérin, and P. Nacass, 2003: Momentum and heat fluxes via the eddy correlation method on the R/V L'Atalante and an Asis buoy. J. Geophys. Res., 108, 3339, doi:10.1029/2002JC001449.
- Peyret, R., and T. D. Taylor, 1983: Computational Methods for Fluid Flow. Springer Verlag, 359 pp.
- Popinet, S., 2002: The Gerris Flow Solver. Release 0.3.0, Gerris. [Available online at http://gfs.sourceforge.net.]
- —, 2003: Gerris: A tree-based adaptive solver for the incompressible Euler equations in complex geometries. J. Comput. Phys., 190, 572–600.
- Porter, D. H., A. Pouquet, and P. R. Woodward, 1994: Kolmogorovlike spectra in decaying three-dimensional supersonic flows. *Phys. Fluids*, 6, 2133–2142.
- Quirk, J. J., 1994: An alternative to unstructured grids for computing gas dynamics flows around arbitrarily complex two-dimensional bodies. *Comput. Fluids*, 23, 125–142.
- Rider, W. J., 1995: Approximate projection methods for incompressible flows: Implementation, variants and robustness. Tech. Rep. LA-UR-2000, Los Alamos National Laboratory, 176 pp.
- Rodi, W., J. H. Ferziger, M. Breuer, and M. Pourquie, 1997: Current status of large-eddy simulations: Results of a workshop. ASME J. Fluids Eng., 119, 248–262.
- Rogallo, R. S., and P. Moin, 1984: Numerical simulation of turbulent flows. Annu. Rev. Fluid Mech., 16, 99–137.
- Samet, H., 1989: Applications of Spatial Data Structures. Addison-Wesley, 507 pp.
- Shah, K. B., and J. H. Ferziger, 1997: A fluid mechanician's view of wind engineering: Large-eddy simulation of flow past a cubical obstacle. J. Wind Eng. Ind. Aero., 67/68, 211–224.
- Thiebaux, M. L., 1990: Wind tunnel experiments to determine correction functions for shipborne anemometers. Canadian Contractor Report of Hydrography and Ocean Sciences 36, Tech. Rep., Beford Institute of Oceanography, Dartmouth, NS, Canada, 57 pp.
- Ye, T., R. Mittal, H. S. Udaykumar, and W. Shyy, 1999: An accurate Cartesian grid method for viscous incompressible flows with complex immersed boundaries. J. Comput. Phys., 156, 209–240.
- Yelland, M., B. Moat, P. Taylor, R. Pascal, J. Hutchings, and V. Cornell, 1998: Wind stress measurements from the open ocean corrected for airflow distortion by the ship. J. Phys. Oceanogr., 28, 1511–1526.
- —, —, R. Pascal, and D. Berry, 2002: CFD model estimates of the airflow distortion over research ships and the impact on momentum flux measurements. J. Atmos. Oceanic Technol., 19, 1477–1499.