ELSEVIER

Journal of Wind Engineering
and Industrial Aerodynamics 96 (2008) 1197–1206

Journal of wind engineering and industrial aerodynamics

www.elsevier.com/locate/jweia

Simulation of simplified-frigate airwakes using a lattice-Boltzmann method

G.F. Syms

Aerodynamics Laboratory, Institute for Aerospace Research, National Research Council Canada, Ottawa, Canada K1A 0R6

Available online 23 July 2007

Abstract

The airwakes of two simplified-frigate shapes were analysed as part of a study of the complex flow field generated as a helicopter interacts with the airwake of a ship. The current computational studies simulate ship airwake without an immersed helicopter rotor. This flow simplification provides insight into the environment a maritime helicopter pilot will encounter when operating from a frigate or frigate-like ship. The flow around the simplified frigate shapes were simulated using the lattice-Boltzmann flow solver PowerFLOW and the results compared to experimental data. That data included the surface flow visualization on one shape and unsteady off-body velocity measurements on the other. From both the surface and off-body mean flow comparisons, it is shown that the lattice-Boltzmann method captures the correct flow topology. The root-mean-square (RMS) of the fluctuating velocities of the simulated airwake are also seen to match well with the measured RMS velocities. The presented results indicate that the lattice-Boltzmann algorithm will capture both the mean and unsteady portion of the airwake on and off the surface of the frigate for low to moderate wind angles.

Keywords: CFD; Ship airwake; Lattice Boltzmann; Frigate; Maritime aerodynamics

1. Introduction

The operation of helicopters from ships is a challenging endeavour. The maritime helicopter pilot must land on and take-off from a platform moving with six degrees of freedom which is most often immersed in the airwake of the flow over the ship's

E-mail address: Jerry.Syms@nrc-cnrc.gc.ca

doi:10.1016/j.jweia.2007.06.040

superstructure, a region of highly turbulent unsteady flow. The airwake contains a wide range of turbulent length scales, some small enough to be felt as vibration, others large enough to affect the performance of the rotor and the helicopter's handling qualities.

As a first step in studying this coupled problem, the ship airwake can be studied without the rotor or helicopter present. This will give insight into the type of environment in which the helicopter may be operating. To this end, two variants of the simple frigate shape, designated as SFS 1 and SFS 2, were studied (Fig. 1). These stylized shapes roughly match the dimensions of a frigate, with the SFS 2 including a realistic bow. These shapes were developed as part of an international collaboration in which Canada, Australia, the United Kingdom and the United States studied the ability of computational fluid dynamics (CFD) codes to simulate complex airwakes (Wilkinson et al., 1998).

The SFS 1 geometry has been used as a validation case by other authors using different approaches. Steady-state Reynolds-averaged Navier–Stokes (RANS) solvers have been applied (Reddy et al., 1999; Wakefield et al., 2002) to this configuration with good agreement being achieved for low to moderate yaw angles but only qualitative agreement on off-body flows being shown for the 90° yaw case. Other work (Syms, 2004; Tulin, 1998) suggests that the use of a steady-state code to simulate accurately the ship airwake might be inappropriate as the steady-state RANS solutions do not necessarily represent the time-average of the unsteady flow. Unsteadiness has been added through the use of perturbations on a mean flow field (Liu and Long, 1998) yet the underlying flow topology is still that of a RANS solution.

The Aerodynamics Laboratory (AL) of the National Research Council Canada (NRC) has studied the SFS configurations experimentally. The mean surface flow of the SFS 1 was examined through surface pressure measurements and oil flow visualization. These data were collected using a 1:60 scale model in the NRC/AL 2 m × 3 m closed circuit wind tunnel (Cheney and Zan, 1999; Zan, 2001). The off-body flow field of the SFS 2 was also measured at the NRC using hot-film anemometers and a 1:100 scale model of the geometry in the same wind tunnel (Lee, 2007).

The AL is studying the use of CFD to simulate frigate airwakes. Initial CFD simulations relied on the use of a commercial RANS solver (Syms, 2004) and pointed toward the need to perform unsteady calculations to map accurately a frigate's airwake. Subsequent studies of the SFS 1 using the commercial lattice Boltzmann method (LBM) solver, PowerFLOW (developed by Exa Corporation) demonstrated the ability of the numerical method to predict the mean surface flows generated by these complex flow fields (Syms, 2003). This paper extends the work on the SFS 1 and examines the steady and unsteady aspects of the off-body flow field of the SFS 2.

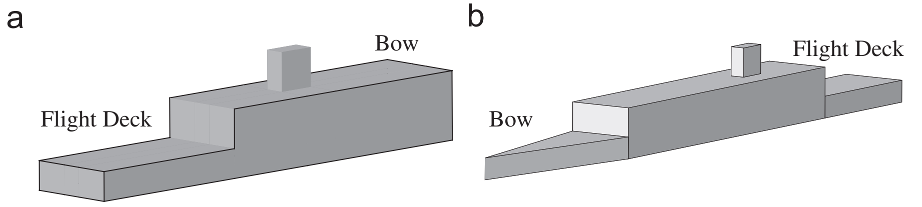

Fig. 1. Simplified frigate shape geometry: (a) SFS 1; (b) SFS 2.

2. Lattice–Boltzmann method

PowerFLOW uses an extended lattice gas/Boltzmann method which is a discrete form of kinetic gas theory operating at the microscopic or mesoscopic scale (Chen et al., 1997; Chen and Doolen, 1998). In the lattice–Boltzmann approach, particles exist at distinct locations in space, moving in discrete directions with discrete speeds at discrete intervals in time. Particles move through the lattice (advection phase or streaming process) then undergo a collision process, at discrete time steps. The goal of the collision process is to drive the flow toward local equilibrium while conserving mass, momentum and energy. The physical time step of this process is governed by the physical characteristic length and velocity of the lattice and is determined by requiring a particle to move from one lattice location to a neighbour in one discrete lattice time step. The LBM method is thus a sequence of particle advection and collisions.

PowerFLOW's fundamental algorithm has the ability to compute flow structures at a variety of scales. However, for high Reynolds number flows, it is not computationally feasible to simulate all length scales and thus one must fall back on modelling of some of these scales. PowerFLOW uses a very large eddy simulation (VLES) approach to turbulence that directly simulates resolvable flow scales and models unresolved scales. It uses a time accurate renormalization group (RNG) form of the standard $k-\varepsilon$ equations with proprietary extensions. The effect of the sub-grid flow is incorporated into the LBM through an eddy viscosity. At no-slip surface boundaries, PowerFLOW implements a proprietary non-equilibrium wall function.

3. Results and discussion

The flow field around the SFS 1 and SFS 2 were computed at several wind angles. Presented here are results for bow (0° yaw) and starboard-bow (45° yaw) winds. The surface flow topology on the SFS 1 flight deck is compared to available experimental surface oil flow data. For the SFS 2, both the mean of the airwake velocity field and its unsteady component are analysed for accuracy against measured quantities.

PowerFLOW is an explicit time-marching algorithm and as such provides unsteady surface and volume data for the flow around the object being simulated. To obtain the mean flow field, flow quantities are averaged over a sufficiently long time after the initial transients of the flow have disappeared. To obtain quantities relating to the unsteadiness of the flow field, time histories are saved during the simulation at experimental measurement points and these time records are used to derive suitable quantities for comparison. In this paper, the root-mean-square (RMS) of the fluctuating component of the velocity field is considered for analysis.

3.1. SFS 1 off-body flow

3.1.1. 0° Yaw

The geometry of the SFS 1 presents to the oncoming flow of a forward facing step at the bow followed by two backward facing steps. The flow topology over the flight deck is that commonly seen in behind a three-dimensional backstep (Fig. 2). A shear layer separates from the roof and sides of the hangar and reattaches on the deck. Inside the recirculation

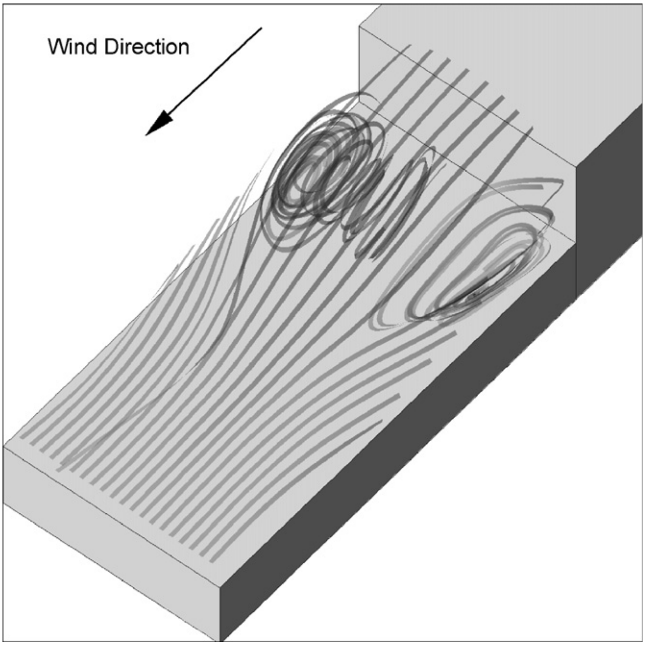

Fig. 2. Flow topology over flight deck for bow winds.

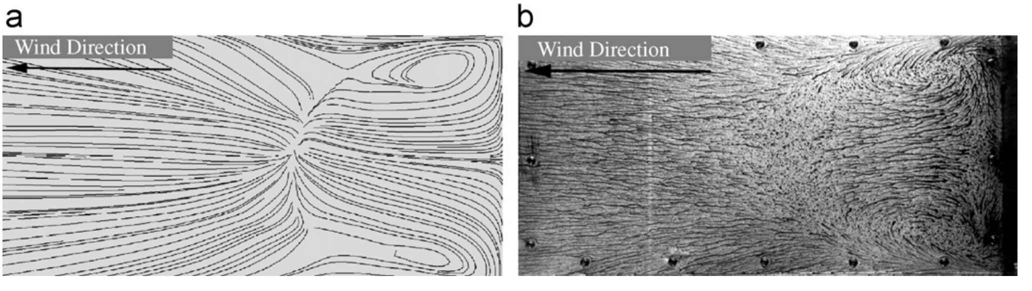

Fig. 3. Surface streaklines on flight deck of SFS 1 (0° yaw): (a) CFD; (b) wind tunnel.

zone formed by that shear layer is a horseshoe vortex parallel to the hangar face with the ends of the horseshoe planted on the flight deck.

The surface streaklines for the computed flow field are presented alongside the experimental surface visualization in Fig. 3. In both flow patterns, the reattachment line of the shear layer is apparent with its maximum downstream extent being just under half the length of the flight deck. Also visible are the two footprints of the horseshoe vortex seen in the two swirl patterns next to the hangar. This comparison between computed and experimental flow fields shows that the LBM method can reproduce the surface flow topology for bow winds over the SFS 1.

3.1.2. 45° Yaw

The flow topology over the flight deck for starboard-bow winds is more complicated than that seen for bow winds. With the wind approaching the starboard side of the

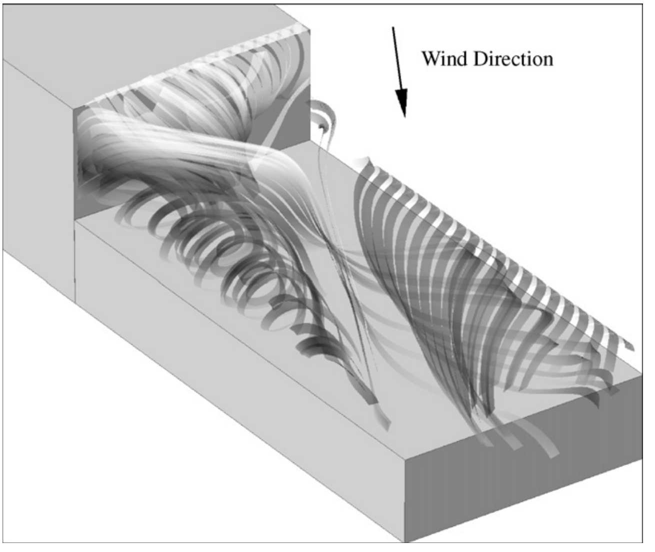

Fig. 4. Flow topology over flight deck for 45° starboard winds.

geometry at a 45° angle, two dominant vortices are formed (Fig. 4). One vortex originates at the windward corner of the intersection of the hangar and flight deck. Its core travels down the flight deck combining with the roll-up of the shear layer separating from the flight deck edge with downwash of the second vortex. This second vortical structure is rooted to the windward corner of the hangar roof. It proceeds diagonally across the hangar before turning and continuing down the leeward edge of the flight deck.

The surface flow pattern resulting from this intricate flow topology can be seen in Fig. 5. The reattachment line separating the two vortices, seen as the streakline running diagonally down the flight deck from which other streamlines emanate, is captured in the computed results. The simulations do show a flow which is generally more inclined to the wind direction; however, the surface flow topology generated by the simulated flow field is in good agreement with that seen experimentally.

The surface flow field is one indicator by which the accuracy of the simulated airwake can be assessed and PowerFLOW computes a reasonable representation thereof. A more informative analysis for the helicopter/ship dynamic interface study examines the off-body flow because it is the flow in which a helicopter is immersed and that aspect of the flow is investigated in the next section.

3.2. SFS 2 off-body flow

To further ascertain the accuracy of lattice Boltzmann airwake simulations, the SFS 2 was simulated at a free stream velocity of 60 m/s at model scale. Hot-film anemometer measurements gathered at the NRC provided the necessary validation data. For the SFS 2, the flow over the flight deck retains the same topology though of slightly different proportions due to the slightly different geometry. This is true for both wind directions simulated.

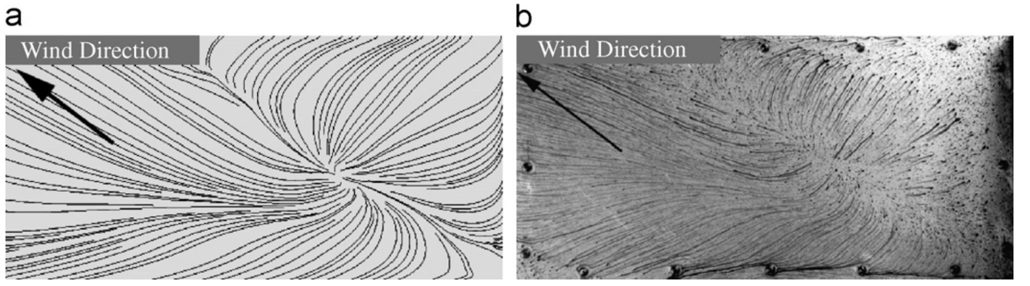

Fig. 5. Surface streaklines on flight deck of SFS 1 (45° yaw): (a) CFD; (b) wind tunnel.

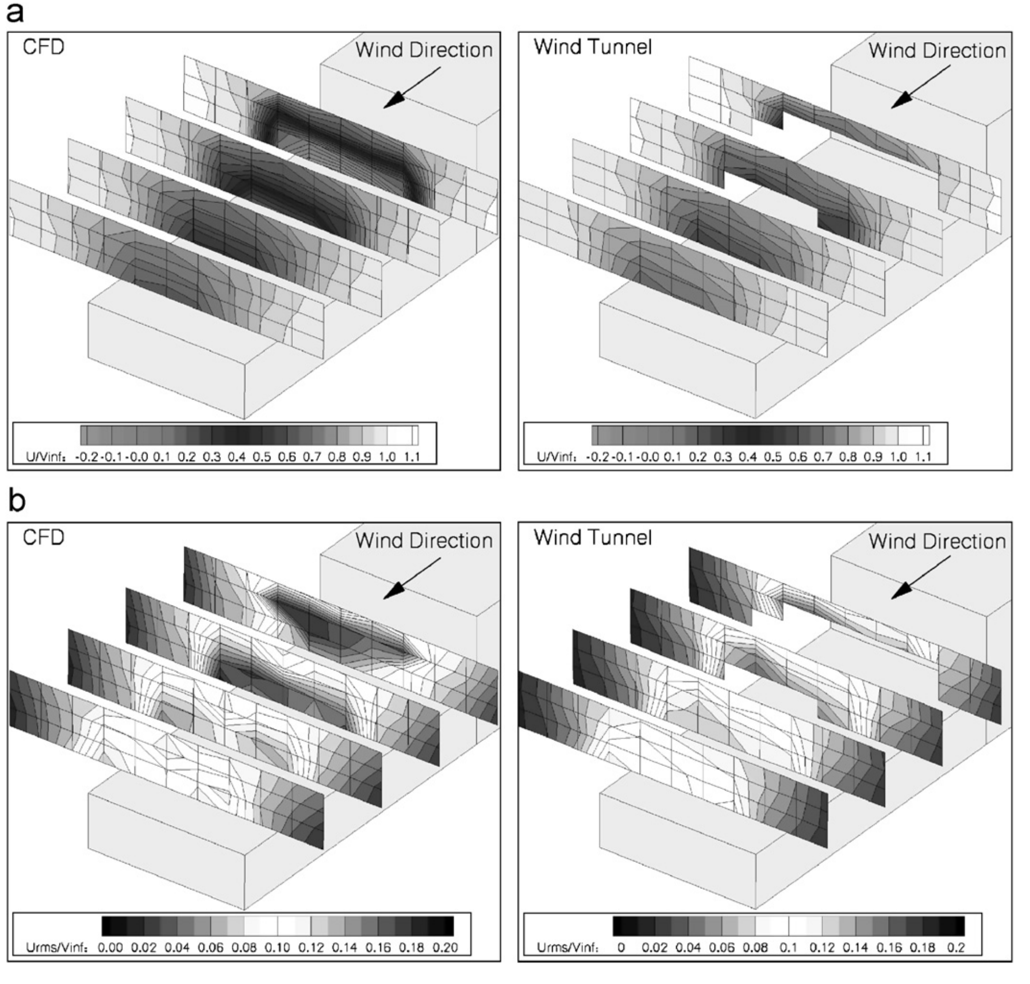

Fig. 6. Streamwise velocity over flight deck of SFS 2 (0° yaw): (a) mean velocity; (b) RMS velocity.

3.2.1. 0° *Yaw*

To analyse the unsteady airwake of the SFS 2, the local velocity is decomposed into its mean and fluctuating parts. Comparisons of the measured and computed mean streamwise velocity (non-dimensionalized by the free stream velocity) is shown in Fig. 6. The planes shown are those given by the experimental data. In the presentation of wind tunnel data, several data points have been excluded from the plot as the data at those points is considered to be unreliable due to measurement limitations of the stationary hot-film anemometers. The figure shows that the computed shear layer bounding the recirculation zone approximately matches the boundary of the excluded points in the measured data (which reflects the extent of the measured recirculation zone). Fig. 7 plots the streamwise velocity component across the flight deck at hangar height on the first plane downstream from the hangar. This plane is $\frac{1}{4}$ length of the flight deck downstream of the hangar and would roughly pass through the front portion of a helicopter rotor landing on the flight deck. The computed velocities show the correct trends with a slightly larger momentum deficit over the flight deck. Due to the high velocity gradients in the shear layer being shed from the hangar, small errors in the placement of that shear layer in the simulation (in this case slightly high) will be reflected as a lower velocity in the airwake. Two additional (dotted) lines in Fig. 7 show the sensitivity of the flow measurements to the position of the shear layer. These lines plot the streamwise velocity 0.5 m above and below hangar height. It can be seen that this small change in height (8% of hangar height) can result in up to 20% change in velocity. It is interesting to note the asymmetry present in both the computed and measured results. It is hypothesized that this is the result of the flow 'locking' into one side of the bow. The bow itself is fairly long and narrow and so it is possible that a fully symmetric flow would be, if not an unstable one, a highly fickle one. More analysis will be required to determine the exact cause of this flow characteristic.

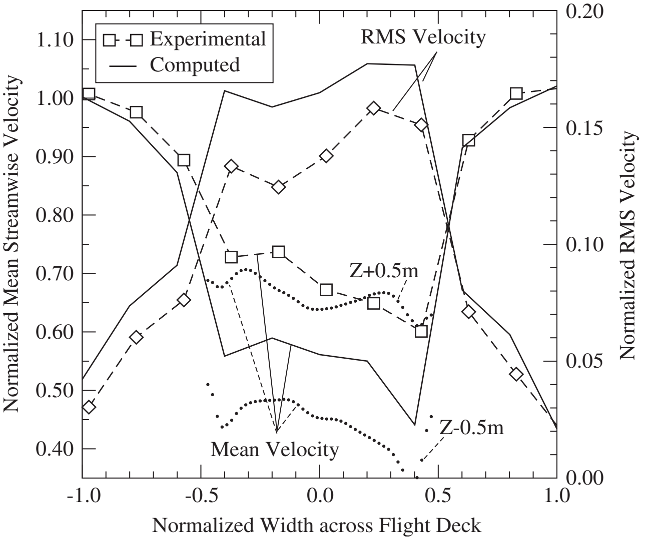

Fig. 7. Mean and RMS streamwise velocities across flight deck of SFS 2 (0° yaw).

A comparison of the computed and measured RMS velocities on the four measurement planes is also shown in Fig. 6 and a quantitative comparison along a line across the flight deck at hangar height (again at $\frac{1}{4}$ of the flight deck downstream) is also found in Fig. 7. These velocities have been non-dimensionalized by the free stream velocity in order to isolate the errors in the RMS quantities from the errors in the mean flow field. The high fluctuations due to the hangar shear layer are apparent in Fig. 6. This figure shows that the computed fluctuating velocities are slightly higher than those measured. This is believed to be a combination of the measurement line being more fully immersed in the unsteady shear layer/recirculating flow (consistent with the slightly high shear layer placement seen in the computed mean flow) and the LBM method incorporating slightly less dissipation than that seen in the experimental flow. The cross-stream plot of Fig. 7 shows that PowerFLOW accurately captures the unsteady portion of the airwake including its apparent asymmetry.

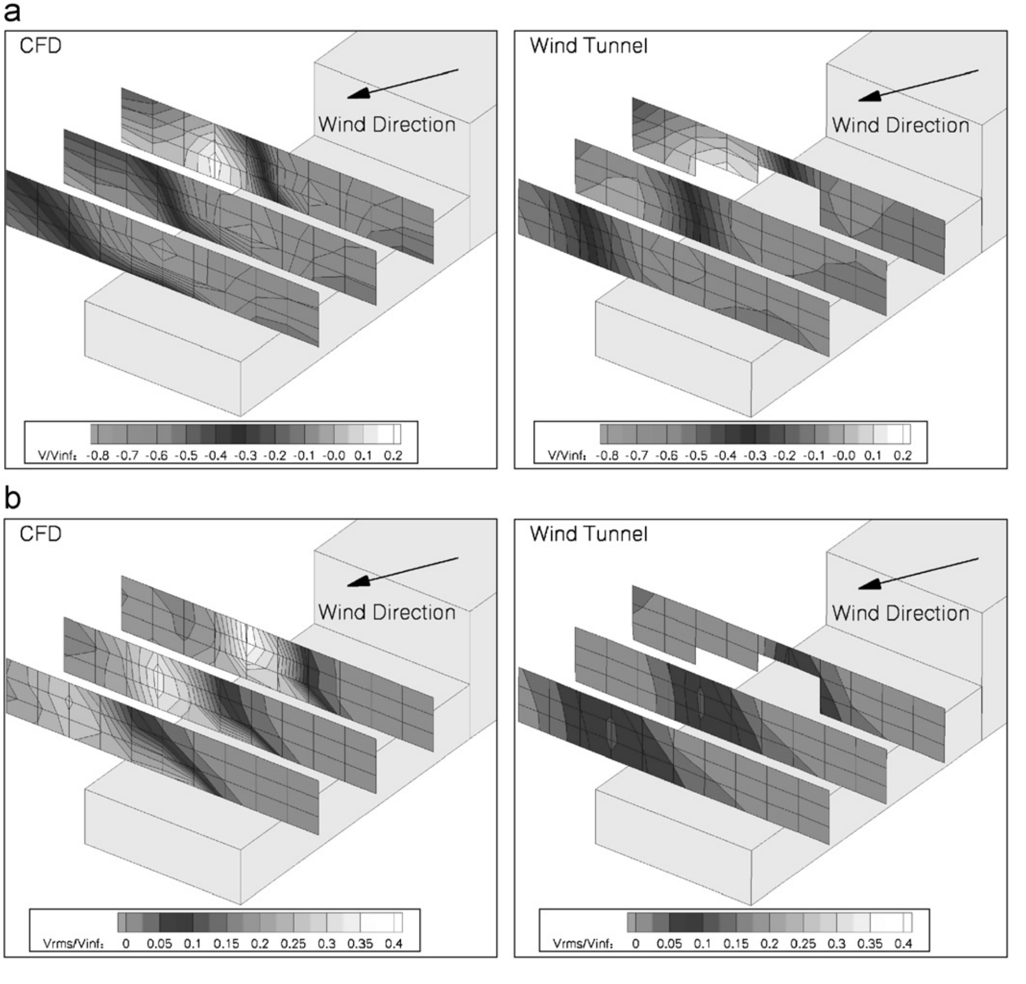

Fig. 8. Cross-deck velocity over flight deck of SFS 2 (45° yaw): (a) mean velocity; (b) RMS velocity.

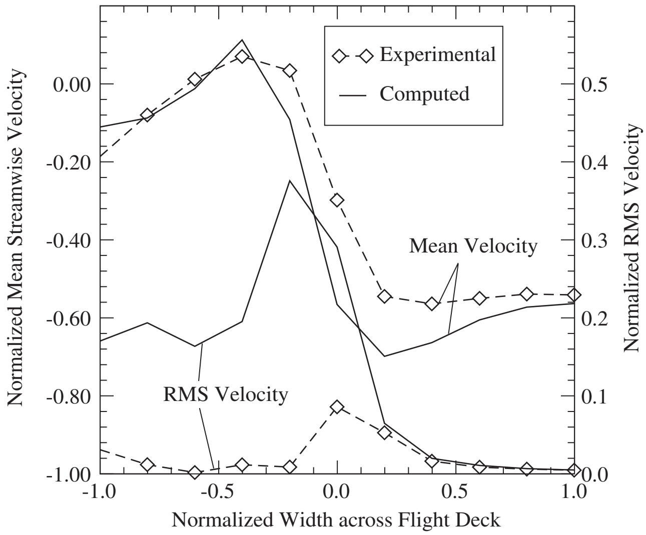

Fig. 9. Mean and RMS cross-deck velocities across flight deck of SFS 2 (45° yaw).

3.2.2. 45° *Yaw*

The flow topology over the flight deck of the SFS 2 with starboard-bow winds is similar to that of the SFS 1 with two dominant vortices present. The edge vortex formed along the windward deck edge lies low to the deck and its presence is for the most part not seen in the experimental measurement planes. The leeward vortex, however, is very much apparent as can be seen in Fig. 8. In this figure, the mean and RMS of the cross-deck velocity, that is the V-velocity in body axes, is presented for both the computed and measured flow fields. To further develop the comparison, the two flow fields are detailed along a line at hangar roof height on the first measurement plane downstream of the hangar in Fig. 9. The simulated flow field of Figs. 8 and 9 shows a slightly more compact vortex resulting in higher mean velocity gradients and a generally larger RMS velocity component. The core of the vortex present in the computed results is slightly higher above the flight deck also contributing to the larger RMS velocity seen in Fig. 9. The above observations suggest that the simulated flow field contains less dissipation that is seen in the experiment. However, the data presented in Figs. 8 and 9 do show that PowerFLOW is able to accurately simulate the flow topology for the 45° yaw case.

4. Conclusions

The flow around two simplified frigate shapes was studied to understand better the complex unsteady flow field existing around the superstructure of the ship. Time accurate unsteady simulations were performed using the lattice-Boltzmann flow solver PowerFLOW. A comparison of the surface flow using computed surface streaklines and experimental oil flow visualization shows that the LBM method captures all the flow features of the airwake for both bow and starboard-bow winds. At both wind angles, the dominant vortical structures are present in the simulated results. An analysis of the off-body flow shows that PowerFLOW recreates the mean and RMS velocity field accurately for the wind angles presented. For bow winds, slight errors in the position of the shear layer emanating from the hangar roof were seen. The starboard-bow wind vortical flow structures were more compact with higher fluctuating velocities. Examination of the computed flow fields suggest that the PowerFLOW simulations contain less dissipation than that seen in the wind tunnel. However, the presented results do demonstrate that a lattice-Boltzmann algorithm can accurately predict the mean and unsteady components flow field of a frigate-like shape.

References

Chen, S., Doolen, G., 1998. Lattice Boltzmann method for fluid flows. Ann. Rev. Fluid Mech. 30, 329–364.

Chen, H., Teixeira, C., Molvig, K., 1997. Digital physics approach to computational fluid dynamics: some basic theoretical features. Int. J. Mod. Phys. C (4), 675–684.

Cheney, B., Zan, S., 1999. CFD code validation data and flow topology for the Technical Co-operation Program AER-TP2 simple frigate shape. Technical Report LTR-A-035, National Research Council Canada/Institute for Aerospace Research.

Lee, R., 2007. SFS 2 Code Validation Data Update. Presented at the TTCP AER TP 2 Dynamic Interface Workshop, 30 April 2003, Patuxent River, USA.

Liu, J., Long, L., 1998. Higher order accurate ship airwake predictions for the helicopter/ship interface problem. In: American Helicopter Society 54th Annual Forum, Washington.

Reddy, K., Toffoletto, R., Jones, K., 1999. Numerical simulation of ship airwake. Comput. Fluids 29, 451–465.

Syms, G., 2003. Simulation of ship airwakes using a lattice-Boltzmann method. In: Proceedings of the 11th Annual Conference of the CFD Society of Canada, Vancouver, pp. 183–190.

Syms, G., 2004. Numerical simulation of frigate airwakes. Int. J. Comput. Fluid Dyn. 18 (2), 199–207.

Tulin, M., 1998. Technical evaluation report. In: Proceedings of the RTO AVT Symposium of Fluid Dynamics Problems of Vehicles Operating Near or in the Air–Sea Interface, RTO MP-15, Amsterdam.

Wakefield, N., Newman, S., Wilson, P., 2002. Helicopter flight around a ship's superstructure. Proc. Inst. Mech. Eng. Part G: J. Aerosp. Eng. 216, 13–28.

Wilkinson, C., Zan, S., Gilbert, N., Funk, J., 1998. Modelling and simulation of ship airwakes for helicopter operations—a collaborative venture. In: Proceedings of the RTO AVT Symposium of Fluid Dynamics Problems of Vehicles Operating Near or in the Air–Sea Interface, RTO-MP-15, Amsterdam, pp. 8.1–8.12.

Zan, S., 2001. Surface flow topology for a simple frigate shape. Can. Aeronaut. Space J. 47 (1), 33–43.