21st Australasian Fluid Mechanics Conference
Adelaide, Australia
10-13 December 2018

Time-Accurate Computational Fluid Dynamics Modeling of the LHA Ship Airwake

M. Cvengros[1,2], **D. Linton**[2], **B. Thornber**[2]

[1]School of Engineering Sciences
KTH Royal Institute of Technology, 114 28 Stockholm, Sweden

[2]School of Aeronautical, Mechanical and Mechatronic Engineering
The University of Sydney, New South Wales 2006, Australia

Abstract

Extensive research in the past two decades has shown that flow around the sharp edges of bluff bodies, such as the superstructure of a ship, can cause large scale shear-layer separations with increased turbulence, generating a highly unsteady flow in the airwake region. Such separations affect the control margins and handling loading of a rotorcraft, mainly if operating from the vicinity of a flight deck, and therefore negatively influence the rotorcraft operating limits. Improved Delayed Detached-Eddy Simulations (IDDES) of the ship airwake were conducted with three different grid levels, the highest being the largest computation to date of the US Landing Helicopter Assault (LHA) ship. Computations were undertaken using the open-source unstructured grid Navier-Stokes CFD software OpenFOAM. Results detail the mean flow and discuss highly-detailed vortical structures and inherent unsteadiness of the LHA ship airwake. Understanding these vortical structures is critical to enabling safe helicopter operations around such vessels.

Introduction

The beginning of LHA ship airwake studies dates to January 1999, in which the V-22 aircraft experienced an uncommanded roll while on the deck of the LHA during compatibility trials, with its rotors turning. Thus began the investigation of rotorcraft/ship interaction, and in the past 18 years, substantial time and effort was dedicated to modeling and analysis of unsteady ship airwakes and their influence on helicopter shipboard operations.

The ship airwake is an unsteady, vortical, three-dimensional and highly separated flow created by the superstructure and deck of a ship. The importance of the airwake is closely tied with helicopters present in the near region of the ship, as it has tremendous influence on a dynamic interface (DI) – a coupled relationship between the ship and a rotary aircraft during sea-based launch and recovery operations. The most challenging scenarios are engagement and disengagement of the rotor system (blade-sailing), takeoff, landing and station-keeping (hoovering over a flight deck). Two primary factors impact the aircraft in DI – the wind-over-deck (WOD) condition (wind speed and angle) and the ship motion. Additional problems involve coupling of the ship and rotorcraft wake, changing the fuselage loading, handling loading, aircraft performance and consequently a pilot workload. It is because of these problems that led to development of safe helicopter operating limits (SHOL), which are usually defined in terms of WOD envelopes.

Obtaining the WOD envelope is essentially possible with in-situ full-scale measurements, wind tunnel scaled model measurements and computational fluid dynamics (CFD) simulations. Full-scale experiments can produce highly valuable data, however due to atmospheric conditions, availability of ship personnel, the need of dedicated equipment including specific helicopter type, a pilot trained for DI operations and the need to repeat tests in small numerous increments of wind speed and azimuth, these experiments can prove to be costly, limited and arguably unsafe, while still being bounded to a specific helicopter type. Wind tunnel experiments can provide good quality data and control over the WOD conditions, however they cannot reproduce the intricate coupling effects between the wake of a ship and rotorcraft, with subsequent performance changes. Due to large ship dimensions, the scaled models required to perform these (wind tunnel) tests are in the range of 1/100th. Furthermore, since the dimensions change so much, so do some elements of scaled frequencies of the unsteady flow phenomena. The flow separation and vortex structures are also shown to be of larger sizes, and although it has been argued and to some extent shown that Reynolds number independence applies [3], one should still be careful when interpreting results obtained through such measurements. CFD has proven to be a reliable candidate in the past two decades of ship airwake studies and has the ability to compute both scaled and full-scale solutions, resolve higher unsteady flow frequencies and provide data also for off-body analysis.

However, CFD simulations come with drawbacks of their own, such as mesh generation and turbulence modeling. Complex ship geometries are in many cases the driving factor for their simplification, in order to obtain a satisfactory mesh quality. The mesh resolution is of importance for adequately resolving the smaller vortex structures which are a driving factor for larger eddies, together with the temporal resolution to resolve flow unsteadiness and shedding frequencies. In order to properly resolve higher velocity gradients near walls, an inflation boundary layer has to be defined, which considerably increases the mesh resolution and bounds the upper value of discrete time in order to maintain the Courant-Friedrichs-Lewy (CFL) condition, increasing the computational time considerably. These are the main reasons why CFD simulations of ship airwake are still of concern even today and a very interesting domain of research.

This paper describes the use of CFD methods in prediction of LHA ship airwake, with the highest mesh resolution up to date in the past two decades using 64 million cells, and demostrating specific LHA ship airwake features not observed before.

Previous Airwake Studies

The first wind-tunnel measurements of the LHA ship are described by Polsky et al. [3], performed on a $1/120^{\text{th}}$ test article. Polsky performed both wind tunnel scale and full-scale CFD simulations, with full-scale WOD conditions of 15 knots and 30 knots and various WOD angles. One of the major contributions by Polsky was a demonstration of Reynolds number independence from 15 knots to 30 knots, however only for starboard 330° WOD angles, not including the headwind condition out of concern that reattachment might prove the independence to not be true. This was later disproved in additional paper by Polsky et al. [4] and Reynolds number independence for WOD of 0° was proven to hold, however Polsky mentioned that although flow field remained similar, the velocity traces began to diverge and that vortex shedding frequency is probably dependent to Reynolds scaling to some extent. The only in-situ measurement to date performed was also described in paper by Polsky et al. [4]. Data were taken using ultrasonic anemometers that captured the flow features mainly at the bow and landing spots 2 and 7. The landing spots are shown in Figure 1. Experimental measurements for V-22/Ship/Helicpoter Aerodynamic Interaction Phenomena (VSHAIP) within joint Army/NASA/NAVAIR $1/48^{\text{th}}$ 7-by 10-foot wind tunnel test at Ames Research Center decribed by Silva et al. [6] captured flow features at four planes spanning the beam of LHA at desired landing spots using Particle Image Velocimetry (PIV) for different WOD conditions at full-scale equivalent. CFD calculations were also performed by Sezer-Uzol [5] for different WOD conditions. Thornber et al. [8] modeled high-density structured mesh topology for Type 23 Frigate and Wave Class Auxiliary Oiler and discussed that inclusion of atmospheric boundary layer requires dense mesh in the region in front of the ship, otherwise the turbulent behavior dissipates. Thornber et al. also discussed that boundary layer modeling is perhaps not necessary due to the separation on sharp edges. Forrest et al. [1] discussed the application of DES simulations on ship airwake for the 2^{nd} revision of Simple Frigate Shape (SFS2) geometry.

The layout of this paper starts with description of computational methods used, description of grid generation together with description of boundary conditions and numerical methods. Next, the comparison to wind tunnel measurements and other CFD simulations is conducted. Results are presented, qualitatively discussing field velocity components and finally, a conclusion is drawn.

Computational Methods

Grid, Boundary Conditions and Numerical Methods

Simulations were performed with open-source unstructured grid Navier-Stokes CFD software OpenFOAM, using the incompressible PIMPLE solver for unsteady flow with DES turbulence modeling using under-relaxation for both pressure and velocity components. One equation Spallart-Allmaras turbulence model was applied together with OpenFOAM implementations of turbulence wall models for kinetic eddy viscosity ν at the LHA boundary modeled with a "wall" boundary condition, effectively imposing zero-velocity condition and zero ∇p condition at the surface. The "sea" was modeled with velocity normal to the surface fixed at zero, and the domain boundary was set to OpenFOAM freestream implementation, imposing fixed-value velocity and zero ∇p for flow entering the domain and zero ∇U_i and ∇p for flow leaving the domain, where $i = 1,2,3$ represents the velocity components.

This paper differs from previous LHA studies by employing fully-structured mesh and resolving higher-detail geometry of the ship superstructure, bow and stern features. The ship length was 250 m for full-scale conditions, with the computational domain set to be approximately three ship lengths behind the ship and two lengths in front the ship, according to simulations performed by Thornber et al. [8] and Forrest et al. [1]. The domain shape was chosen to be cylindrical to better accommodate for WOD angle changes, with the center in the middle of the space between the end of superstructure and stern of the ship, in a similar fashion as presented by Thornber. Both the ship geometry, domain and mesh showing the structured topology is shown in Figure 2. In order to properly resolve the vortex structures, an increased attention was payed to the mesh quality in the entire deck region spanning above the ship, which was modeled as highly isotropic and cartesian. Mesh was manually constructed through multi-block technique in ANSYS ICEM CFD.

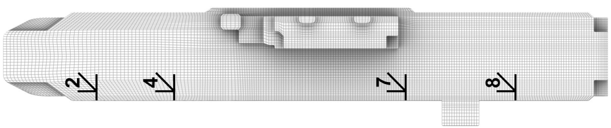

Figure 1: Locations of landing spots $x_2 = 37.6\,\text{m}$, $x_4 = 69.65\,\text{m}$, $x_7 = 165.7\,\text{m}$ and $x_8 = 211.4\,\text{m}$, all at $y = -15\,\text{m}$. Coordinate system begins at the centerline of the ship, with $x = 0\,\text{m}$ at the bow and propagates downwind. Positive y direction faces starboard side.

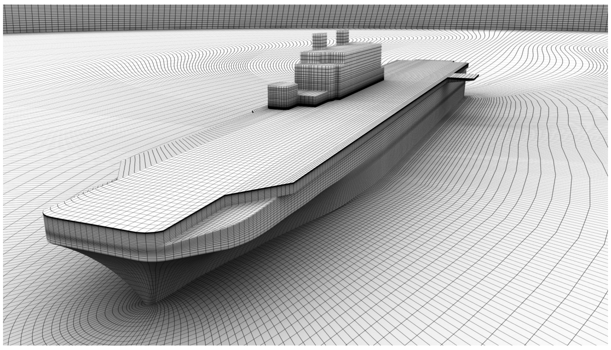

Figure 2: Structured mesh and ship geometry for 4.69 million cells.

Three meshes of 6.3 million, 12.24 million and 64.24 million cells were used to perform the CFD simulations. The cell length at the deck ranged from approximately 1.5 m from the coarser mesh up to 20 cm in the finest mesh. Polsky et al. [4] demonstrated that reattachment happens at the deck region in the case of headwind condition, and so boundary layer growth was included. For wall modeling, the highest acceptable value of y^+ was chosen to be 300 at the stern of the ship. For all DES simulations, the boundary layer cell growth was performed as described by Spalart [7]. Only the deck of the ship was given a boundary layer growth, with the rest of the ship using regular, although highly cartesian cell spacing due to the nature of the structured mesh.

To test the grid independence, a separate steady-state calculations using Reynolds-Averaged Navier-Stokes (RANS) turbulence modeling were performed in order to provide additional comparable data-points. Polsky et al. [3] has shown that time-averaged values do not compare well to Reynolds-averaged values in ship airwake studies, due to the highly unsteady nature of the wake, which produced complex vortical structures that cannot be captured by Reynolds-averaging. Indeed, comparison of time-averaged and RANS results showed that although velocity magnitudes preserved some similar patterns, they were too different to be successfully used for grid comparison. RANS simulations used two equation Menter's Shear Stress Transport $k-\omega$ turbulence model [2], with appropriate OpenFOAM implementation of boundary conditions at the ship surface for both eddy kinetic energy k and energy dissipation rate ω. The freestream boundary conditions for k and ω were defined as in Menter. Grid-convergence study was performed with 600 thousand, 1.65 million and 4.69 million cells. Due to the structured mesh topology and ship geometry, cells with highly non-orthogonal value were identified at the bow of the ship. These regions were enough to cause major convergence difficulties for the steady-state simulations, and numerous mesh adaptations and repeated simulations were required to generate results with

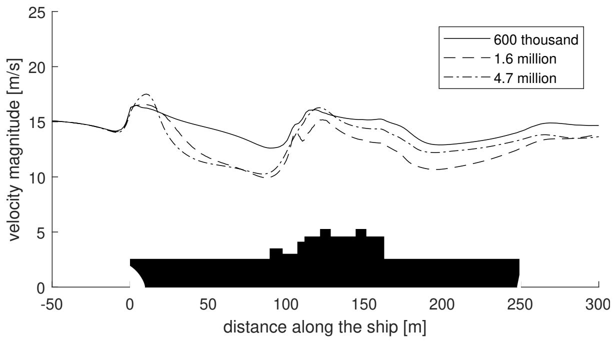

Figure 3: Velocity magnitudes from RANS simulations used for convergence study for deck centerline at $z = 6$ m.

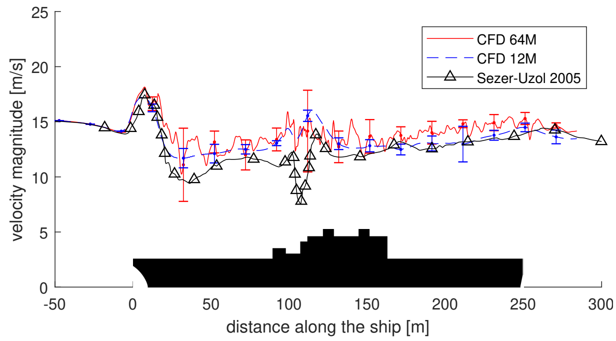

Figure 4: Time-averaged mean velocity magnitudes along ship deck at $z = 6$ m .

sufficient residual convergence. The numerical schemes used were an OpenFOAM implementation of second-order "linear upwind" scheme for gradient terms and second-order "linear" scheme for diffusive terms. Cell limiting was used for the gradient terms, limiting the gradient such that the extrapolated cell values to faces never fell outside of the bounds of values in the surrounding cells. The diffusive terms used correctors with limiters, adding explicit non-orthogonal correction to orthogonal components. The results of grid convergence study are in the Figure 3.

Solution Strategy

The freestream velocity was chosen to be 15.43 m/s (30 knots) according to simulations performed by Polsky [3], Sezer-Uzol et al. [5] and wind tunnel measurements performed by Silva et al. [6]. Due to the smaller cell sizes of the boundary layer, a variable time-step with an initial value of 1 ms was chosen in order to maintain the CFL condition. According to Polsky, at least two ship "flow-over times" were required before the unsteady flow patterns could be deemed as developed. This suggests approximately 30 s of physical time, after which the sampling of frequencies was started. For the 64 million case, considerable amount of CPU time was required, with 3120 cores running in parallel. However, it was only possible with current computational facilities to run for 30 s of total time. Therefore, the results presented for this case are quantified by evaluated uncertainties, and it can be assumed that the single time-step at the end of initialisation period represents the flow well, as will be discussed later. For validation purposes, the 12 million case was used, which ran the full initialisation period of 30 s, after which the flow patterns were sampled for 10.7 s with sampling frequency of 10 Hz. The 6.4 million case was run as a posteriori simulation in order to provide additional validation data with a much higher sampling rate of 100 Hz, however due to the lack of CPU resources, only 12.19 s of physical time were calculated, and so the results from these data are not presented.

Results and Discussion

Simulations of 12 million and 64 million cases were compared to CFD simulations of Polsky et al. [4] and Sezer-Uzol et al. [5] for 3 m (approximately 10 ft) and 6 m above the ship deck and WOD conditions of 30 knots and 0° (headwind). It was found that while time-averaged velocity magnitude at 6 m compared to results presented by Sezer-Uzol rather well, velocity magnitudes at 3 m experienced more disturbance and variation. Interesting was the observation that velocity magnitudes presented by Polsky and Sezer-Uzol did not compare well to each other and correlated only qualitatively. Velocity magnitudes for 6 m above the ship deck are shown in Figure 4. While the 64 million case obtained only 5 time-averaged samples from the beginning of the initialisation period compared to 107 values of sampled time from 12 million case, the mean values were still within the reasonable bounds of uncertainties, with the time-accurate velocities at $t = 30$ s of 64 million and 12 million case comparing extremely well.

Mean vertical velocity behind spot 7 and spot 8 at height $z = 6$ m above deck was compared to wind tunnel values presented by Silva et al. [6]. The comparison is within uncertainty bounds for spot 8, however does not compare well with spot 7, although the patterns are preserved, as can be seen in Figure 5. It was observed that the mean vertical velocity was extremely sensitive to the position of the sampling plane. The model used in the wind tunnel measurements was roughly 16.7 ft long, which translates to 244.33 m for full-scale. Since the CFD model used was 250 m in length, corrections needed to be used. It was assumed that for mean vertical velocity behind superstructure, exact locations and dimensions of both the ship and superstructure must be matched in order to expect viable results. Results are shown in Figure 5.

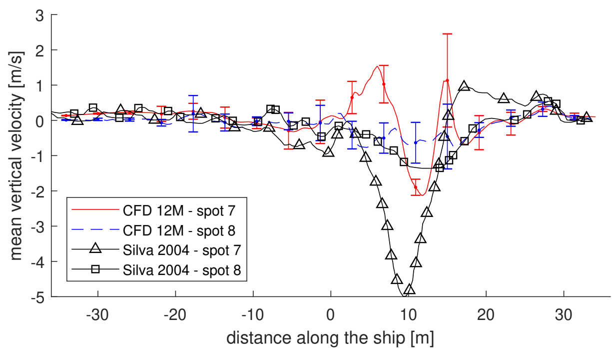

Figure 5: Mean vertical velocities at spot 7 and spot 8 for $z = 6$ m.

Highly detailed pressure contours of the ship deck are shown in Figure 6. Volumetric contour plots representing the vorticity magnitude can be seen in Figure 8.

The unsteady nature of the flow is immediately apparent, showing mainly the large separation at the bow of the ship. Vortical structures crawling along the sides are typical for the 0° WOD condition and were observed both by Polsky et al. [4] and Sezer-uzol et al. [5]. Polsky also described a presence of "burbles" – gusts of vortical islands of wind separating off the bow of the ship. These structures can be better seen in Figure 7. Interesting is also the strong deflection of the flow field after the ship superstructure in Figure 8, which was previously not observed in LHA ship airwake studies for 0° WOD condition. Since the

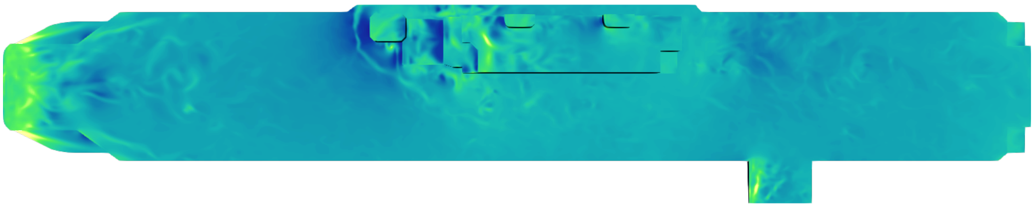

Figure 6: Contours of pressure at the ship deck. Contours show pressure divide by density difference, with lighter colors representing lowering of pressure and darker colors pressure increase.

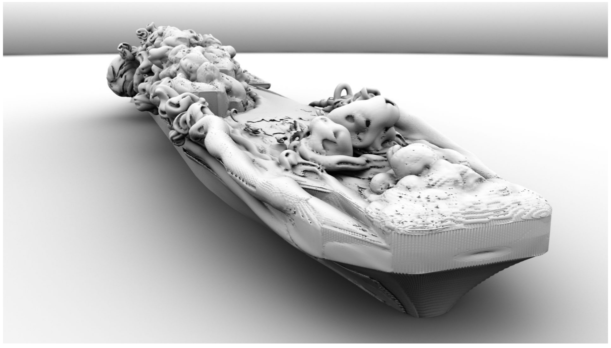

Figure 7: Start of strong separation off the ship bow at $t = 5\,\mathrm{s}$.

highly unsteady and turbulent flow is the main concern for aircraft performance in DI, these flow field features are a good candidate for future work.

Conclusions

CFD simulations of LHA ship were performed using the open-source CFD sotware OpenFOAM producing time-accurate data. Resulting velocities were compared both to CFD and wind tunnel measurements. Data compared relatively well to previous CFD analysis of velocity magnitudes at 6 m above the deck, however comparison was worse if closer to areas with higher unsteadiness. Due to these reasons, comparison with wind tunnel data proved to be difficult, as even a small difference in sampling position produced different flow patterns and velocity values. It was demonstrated that OpenFOAM has the ability to capture the highly unsteady characteristics of the LHA ship airwake and specific flow field structures unique only for the $0°$ WOD case were successfully presented. These structures agreed with the observations made by previous studies, such as strong bow separation and periodic shedding of vortical flow islands. Refined resolution of vortical structures shed from the ship sides and the bow were observed.

References

[1] Forrest S. J., Owen I., An Investigation of Ship Airwakes Using Detached-Eddy Simulation, *Computers & Fluids*, 39 (2010) 656-673.

[2] Menter, F. R., Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications, *AIAA Journal*, Vol. 32, No. 8, August 1994.

[3] Polsky, A. S., Bruner, W. S. C., Time-Accurate Computational Simulations of an LHA Ship Airwake, *AIAA-2000-4126*.

[4] Polsky, A. S., Bruner, W. S. C., A Computational Study of Unsteady Ship Airwake, *AIAA 2002-1022*.

[5] Sezer-Uzol, N., Sharma A., Long N. L., Computational fluid dynamics simulations of ship airwake, *Proceedings of the Institution of Mechanical Engineers*, Oct 2005, 219, 5, pg. 369.

[6] Silva, J. M., Yamauchi K. G., Wadcock J. A., Long R. K., Wind Tunnel Investigation of the Aerodynamic Interactions Between Helicopters and Tiltrotors in a Shipboard Environment, *American Helicopter Society 4th Decennial Specialist's Conference on Aeromechanics*, San Francisco, CA, January 2004.

[7] Spalart, R. P., Young-Person's Guide to Detached-Eddy Simulation Grids, NASA/CR-2001-211032.

[8] Thornber B., Starr M., Drikakis D., Implicit Large Eddy Simulation of Ship Airwakes, *Aeronautical Journal*, Volume 114, Issue 1162, pp. 715-736, 2010.

Figure 8: Iso-surfaces showing the vorticity flow field with $\|\mathbf{rot}\,\mathbf{U}\| = 2.5\,\mathrm{Hz}$ for $0°$ WOD condition and freestream velocity of 30 knots.