Computational Fluid Dynamics for Flight Simulator Ship Airwake Modeling

Jeffrey D. Keller,
Glen R. Whitehouse, Alexander H. Boschitsch
Continuum Dynamics, Inc.
Ewing, NJ
jeff@continuum-dynamics.com,
glen@continuum-dynamics.com,
alex@continuum-dynamics.com

Juan Nadal,
Jeff Jeffords, Marty Quire
CAE USA, Inc.
Tampa, FL
Juan.Nadal@cae.com,
Jeff.Jeffords@cae.com,
Marty.Quire@cae.com

ABSTRACT

Modeling and simulation developments have resulted in high fidelity pilot-in-the-loop flight simulators providing realistic training environments. Modeling challenges continue to exist, in particular for accurate simulation of the near-ship environment critical to landing a helicopter onto the flight deck of a moving ship with various wind conditions. Providing an effective simulated environment requires modeling of the highly unsteady airwake resulting from bluff-body aerodynamic interactions of the ship superstructure and hangar near the flight deck and in close proximity to the ship as it passes through the airstream. This paper describes the development of a U.S. Navy rotary wing flight simulation with turbulence effects including high-fidelity representation of the ship airwake environment. The spatially-varying and time-varying flow field around the ship is determined off-line using a hybrid, inviscid CFD methodology that is well-suited for representing the turbulent environment several ship lengths downwind from the flight deck with moderate computational requirements. Results from this off-line analysis are formulated into a ship airwake database for multiple landing platforms and wind-over-deck conditions suitable for real-time pilot-in-the-loop virtual simulation. The paper describes the development of the simulation flight dynamics model, development and validation of the CFD-based ship airwake flow fields, and integration of the ship airwake database within the aerodynamic model. Implementation issues associated with integrating the ship airwake database into the flight dynamics model associated with real-time implementation and memory management are identified, and the approach to overcome these issues are described.

ABOUT THE AUTHORS

Jeffrey D. Keller is an Associate with Continuum Dynamics, Inc., where he has been since 1997. His professional interests include air vehicle aerodynamics and flight dynamics modeling and aircraft flight controls development, and he has led several flight dynamics and simulation projects for unmanned aircraft systems and shipboard dynamic interface environments. He received a Ph.D. from the Department of Mechanical and Aerospace Engineering at Princeton University in 1998 and also received B.A. and M.Eng. degrees in Aeronautical Engineering from Rensselaer Polytechnic Institute in 1990 and 1991, respectively.

Juan Nadal obtained a Bachelors degree in Mechanical Engineering from the University of Puerto Rico in 2003 and later received a Masters degree in Aerospace Engineering from the University of Michigan in 2006 with a specialization in Flight Dynamics and Control. He currently works as a Flight Dynamics Engineer for CAE USA, Inc. where he is responsible for ensuring that all aspects of the flight dynamics environment are simulated and working properly. Juan has held previous positions with Rolls Royce as a Summer CO-OP where he developed a simulation of a turbo fan engine start up sequence, at Stryker Corporation as a Manufacturing Engineer, and at NASA Goddard Space Flight Center as a summer intern where he conducted tests on two miniature Loop Heat Pipes (mLHP) for satellites to characterize its operation. His interests include fixed wing and rotary wing aircraft flight dynamics simulation and rotor dynamics.

Glen R. Whitehouse is an Associate with Continuum Dynamics, Inc., where he specializes in rotorcraft aeromechanics and CFD for flight mechanics applications. At CDI he has led the development of novel Eulerian wake modeling tools for accurately simulating vortical flows and is the principal investigator of an ongoing effort to develop both bio-kinetic/bio-mimetic aeromechanical modeling tools and a prototype unconventional flapping wing MAV for the U.S. Air Force. He received a Ph.D. and D.I.C. from the Department of Aeronautics at Imperial College, London in 2004 and his B.S. in Aeronautical Engineering and Mathematics from Clarkson University in 2000.

Alexander H. Boschitsch obtained his B.S. degree in Aeronautical Engineering at Queen Mary College in London and his Ph.D. from Princeton University in 1989. He is a Senior Associate at CDI, where he has developed several state-of-the-art CFD packages with a focus on aeroelastic applications. His interests also include application of fast boundary element methods in aerospace and biomolecular analyses.

Jeff Jeffords has been the Group Leader of Flight Dynamics at CAE USA, Inc. for the past 5 years. Previous experience in simulation includes flight dynamics for both rotary wing and fixed wing aircraft, including implementation and test of mission critical tasks related to ship board operations. Rotary wing experience includes support of flight test data acquisition, model development, and validation. He has 24 years of experience supporting aircraft simulation projects. Helicopter Airframe experience includes HH-60, UH-60, MH-60S, MH-47G, and SH-2G. Mr. Jeffords graduated from Auburn University with a Bachelors degree in Aerospace Engineering.

Marty Quire is a Project Engineer at CAE USA, Inc. He is currently leading the development and deployment of the Navy's new MH-60R and SH-60B Tactical Operational Flight Trainers, which are the first trainers to incorporate the CFD Ship Airwake modeling techniques described in this paper. He is a 28 year veteran of the simulation industry and received a Bachelors degree in Electrical Engineering from Georgia Tech in 1979.

Computational Fluid Dynamics for Flight Simulator Ship Airwake Modeling

Jeffrey D. Keller,
Glen R. Whitehouse, Alexander H. Boschitsch
Continuum Dynamics, Inc.
Ewing, NJ
jeff@continuum-dynamics.com,
glen@continuum-dynamics.com,
alex@continuum-dynamics.com

Juan Nadal,
Jeff Jeffords, Marty Quire
CAE USA, Inc.
Tampa, FL
Juan.Nadal@cae.com,
Jeff.Jeffords@cae.com,
Marty.Quire@cae.com

INTRODUCTION

One of the most difficult piloting tasks presented to an aviator is the challenge of landing an aircraft (helicopter) onto the flight deck of a moving ship in high sea-states with gusty wind conditions. This task is compounded by the aerodynamic disturbances that result from the flow around the ship superstructure, which is in close proximity to the landing spot. The spatially- and time-varying flow field associated with this ship airwake can lead to significant pilot control activity and workload during approach and recovery, station-keeping, and vertical replenishment (VERTREP) tasks. Determination of launch/recovery and other operational envelopes, as well as pilot training, has depended extensively on flight testing, although testing in this manner is expensive and requires that two assets (ship and aircraft) are available. Furthermore, such tests require that sufficiently variable winds and sea states exist to properly cover the range of operations anticipated for fleet use. One potential alternative, however, is the use of flight simulation (Carico and Madey, 1984). Flight simulation of ship aviation operations, in particular for pilot training, has an obvious benefit of the availability of any sea state, wind condition, and surface ship, provided that the underlying simulation mathematical model has appropriately characterized the complex aerodynamic environment of the ship airwake and its influence on the aircraft flight dynamics.

Computer simulation of the shipboard environment is very complex, however, due to the need to model all aspects of the operational environment. Specifically, considerations must be given to proper modeling of multiple phenomena, including the ship motion in the particular sea-state condition; the atmospheric turbulence seen by the ship superstructure; the airwake over the ship arising from external winds and ship-generated motion; the helicopter response to both pilot inputs and airwake disturbances; the interaction of the helicopter rotorwash or aircraft jet wash with the ship structure and airwake in the landing zone; and the visual, aural and motion cues associated with the piloting of the aircraft in the at-sea environment (Healey, 1987). Individually, modeling the ship airwake represents an extremely challenging problem, and interactions between the ship airwake and these other phenomena (e.g., rotorwash interactions) push the limits of current-generation computational capabilities well beyond what is feasible for real-time implementation.

Historically, real-time flight simulators have used relatively simplistic models for the ship airwake disturbance due to limits of computational power and available airwake data. In recent years, the advent of computational fluid dynamics (CFD) has provided analytical tools suitable for predicting the complex, unsteady flow field associated with the ship airwake. There has been considerable research and development on the application of CFD to model the highly separated bluff-body flow associated with the flow past a ship (e.g., Liu and Long, 1998; Tai, 1998; Polsky and Bruner, 2000; Polsky 2002; Arunajatesan et al., 2004, Alpman et al., 2007). These investigations have examined a range of CFD methods (i.e., inviscid Euler versus viscous Navier-Stokes flow solvers) and modeling parameters, including the effects of ship motion and interactions between the ship airwake and aircraft jet wash or rotorwash. Applications of CFD methods for modeling the ship airwake disturbance environment in real-time, piloted flight simulations has been more limited, however, where CFD is used to pre-compute the flow field as part of a ship airwake database (Bunnell, 2001, Bogstad et al., 2002, Lee et al., 2005). Application of CFD to real-time shipboard flight simulations has typically been limited to modeling of a single ship-helicopter combination, although the latter investigation (Bogstad et al., 2002) has focused on the development of a multi-ship airwake database for a helicopter pilot training simulator. Development of such a database is a substantial undertaking, requiring the balance of modeling fidelity, database flow field discretization, and computational limitations of memory and storage.

This paper describes the development of ship airwake databases from CFD for application in U.S. Navy H-60 piloted training simulators. Analysis of the unsteady flow field due to the flow past multiple ship and landing platforms is performed using a hybrid, inviscid CFD methodology, which has been developed with several features that are well-suited for this application. After a brief overview of the flight simulator development, a description of the CFD analysis is provided, and results are presented that demonstrates its applicability to modeling the disturbance environment in a piloted flight simulation. The approach for the formulation and integration of the ship airwake database is described, which addresses real-time operation and memory management considerations. This paper describes on-going work that will ultimately conclude with pilot evaluations in late-2007 and 2008.

SIMULATION DEVELOPMENT OVERVIEW

Flight Dynamics Simulation Model Overview

The flight dynamic model consists of a nonlinear, total force and moment model with 11 degrees of freedom: six rigid-body, three rotor-flapping, dynamic inflow, and the rotor rotational degrees of freedom. The flight dynamic simulation includes mathematical representations of the fuselage, empennage, main rotor, and tail rotor.

The fuselage model computes the forces and moments due to the surface pressures and skin friction, including external stores, represented by six aerodynamic components which are defined from wind tunnel data. The angle of attack at the fuselage is calculated using the free stream and interference effects of the main rotor, which are based on rotor loading and rotor wake skew angle.

The empennage aerodynamics are modeled separately from the fuselage. The angle of attack at the empennage is developed from the free stream velocity and rotorwash. Additional dynamic pressure effects from the fuselage are accounted for by factoring in the free stream velocity component.

The main rotor model is based on the ARMCOP model (Chen, 1979) with correction factors to fine tune the static and dynamic responses as required by validation criteria. In addition, the model has been enhanced with a dynamic inflow formulation (Schrage et al., 1988). This mathematical representation explicitly accounts for the dynamic effect of rotor modes, such as rotor-blade-flapping. The blade flapping equations of motion contain the primary rotor parameters, specifically: flapping hinge constraint, hinge offset, blade Lock number, and pitch flap coupling.

The dynamic inflow formulation is based on the Pitt-Peters theory (Pitt and Peters, 1981) that relates the airloads of a rotor (C_T, C_L, and C_M) to the induced inflow distribution $\nu(r,\psi)$, where C_T, C_L, and C_M are the aerodynamic perturbations in thrust, roll moment, and pitch moment, respectively (Lee, 2005). The induced inflow is assumed to have the following variation in the wind axis coordinates:

$$\nu(r,\psi) = \nu_0 + \nu_s \frac{r}{R}\sin\psi + \nu_c \frac{r}{R}\cos\psi \tag{1}$$

where r is the blade radial coordinate, R is the rotor radius, and ψ is the rotor azimuth angle. The inflow states ν_0, ν_c, and ν_s are the magnitude of uniform, lateral, and longitudinal variations in induced flow. The time histories of ν_0, ν_c, and ν_s are governed by the following first-order differential equation:

$$[M]\begin{Bmatrix}\dot{\nu}_0\\ \dot{\nu}_s\\ \dot{\nu}_c\end{Bmatrix} + [L]^{-1}\begin{Bmatrix}\nu_0\\ \nu_s\\ \nu_c\end{Bmatrix} = \begin{Bmatrix}C_T\\ C_L\\ C_M\end{Bmatrix} \tag{2}$$

where $[M]$ is the matrix of the apparent mass terms, $[L]$ is the nonlinear version of the inflow gains matrix.

The tail rotor model is modeled as a teetering rotor without cyclic pitch. For this case, the forces in the wind-hub system are obtained from the expressions derived for the main rotor by setting the lateral and longitudinal cyclic pitch terms equal to zero. Furthermore, since the tail rotor flapping frequency is much higher than that of the main rotor system, the tip-path-plane dynamics are neglected. Thus, the first and second derivatives of the blade flapping non-rotating coordinates are set equal to zero in the force equations. The result is a set of basic quasi-static force expressions similar to those in classical work. The local flow at the tail rotor includes the effect of downwash from the main rotor system (Talbot et al., 1982).

SHIP AIRWAKE CFD MODELING

The primary objective of this simulation development has been to model the unsteady flow field using CFD that permits efficient calculation of the ship airwake database inputs. As discussed previously, CFD methods are finding widespread application in modeling the unsteady flow field for complex aerospace systems, and in particular, for ship airwake applications. Fundamentally, the spatially- and time-varying flow field within the ship airwake is determined by the shedding and transport of vortical structures from the ship hull, flight deck, and superstructure. These structures convect downstream and may introduce potentially adverse interactions with operating aircraft.

Several factors must be considered in the application of conventional volumetric CFD methods to the analysis of this unsteady flow field. First, results are sensitive to the computational mesh, and conventional CFD methods require fine meshes to predict the flow structures of interest and to minimize the numerical diffusion of vorticity. Resolving the vortical structures in the ship airwake several ship lengths downstream would require large CFD meshes using conventional methods and would incur enormous data management and CPU requirements. Second, the generation of the computational grid for conventional CFD methods is non-trivial, typically requiring a very high resolution volumetric mesh near the ship hull and superstructure. Conventional boundary conforming meshes require considerable user-intervention, and while unstructured (tetrahedral) meshes are more amenable to automation, the time required to generate the mesh is lengthier. Poor quality cells (e.g., slivers) are invariably present in a complex geometry mesh and usually must be fixed before embarking on the fluid computation.

The approach used here combines two analyses that are complementary and address the considerations described above. The CFD methodology that is used for the development of the ship airwake database is a hybrid method that solves the inviscid (Euler) flow equations while preserving the vorticity transport processes in the ship airwake, which is critical for capturing turbulent velocity fluctuations in the unsteady flow field. This hybrid methodology combines a novel, vorticity transport formulation (VorTran-M) with a hierarchical, Cartesian grid Euler solver (CGE). A brief description of the combined methodology is provided below.

CFD Methodology Description

The CFD methodology used for the ship airwake database is derived from prior research and development work. The first analysis, the CGE model (McKillip et al., 2002), determines the unsteady flow field by solving the compressible (3D) Euler equations[†] upon a Cartesian grid structure consisting of a hierarchical collection of nested cube-shaped cells (an octree). A central element in the Cartesian grid concept is reliance upon intersection methods to generate the cell volumes and areas at the surfaces rather than attempting to align the mesh with the complex surfaces. Once a surface (ship) geometry definition is provided, the subsequent grid generation and flow computation can proceed autonomously since the Cartesian mesh does not need to be boundary conforming. Furthermore, flow separation points, which initialize the propagation of the large-scale flow structures in the ship airwake, do not need to be explicitly specified since the preset computational scheme ensures that Kutta conditions are implicitly enforced at sharp corners and edges. Thus, the critical flow features of the ship airwake, which are dominated by bluff body separation and vortex shedding, can be captured by an inviscid Euler formulation, and enormous reductions in CPU time can be gained over alternative CFD approaches (e.g., Reynolds-averaged Navier-Stokes or RANS solvers).

The VorTran-M methodology is a specialized time accurate CFD solver that accurately models unsteady vorticity transport processes (i.e., evolution, coalescence, and rupture of vortical structures in the flow field) using an inviscid vorticity-velocity formulation of the Navier-Stokes equations (Brown, 2000; Whitehouse et al., 2007). This approach has demonstrated good conservation of the strength and structure of wakes and addresses limitations in traditional Eulerian-based primitive variable (PV) (i.e., velocity and pressure) CFD methods by controlling numerical diffusion using a carefully constructed flux formula and selecting an appropriate flux limiter. To solve for the vorticity evolution, and hence, flow field, it is necessary to initialize the vorticity at grid locations with a distribution provided by traditional CFD flow field solvers and/or other methods. Thus, the CGE and VorTran-M models are complementary and provide a hybrid CFD approach, where the CGE model provides the near field solution and the VorTran-M model provides the far wake evolution and velocity field prediction.

This hybrid methodology provides a low-dissipation, first-principles methodology for application to wake-

[†] While primarily intended to solve compressible flows, the CGE analysis has also been shown to behave well for low speed flows with Mach number less than 0.1, so can be readily applied to ship airwake calculations.

dominated flow field predictions that capture and sustain strong vortical structures over time and length scales of interest. Furthermore, the procedure for generation of the computational mesh around the complex ship geometries can be automated with essentially no user involvement required.

An additional comment on the use of inviscid CFD methods is required. A critical observation in the development of the ship airwake database is that viscous flow effects are significant only in localized regions near to the ship surface. Outside this boundary layer region, small-scale flow features, for which full, viscous Navier-Stokes solvers would be required, do not induce aerodynamic forces on the helicopter that are relevant to piloted flight simulation applications. The aerodynamic forces resulting from these small-scale flow features are effectively filtered by the flight dynamic response of the helicopter. The CFD methodology used here is inviscid and strikes the correct balance between completeness in physical modeling with the computational effort to produce flow field solutions. The primary flow features in the ship airwake are controlled by inviscid processes (unsteady vortex wake propagation) and flow separation from sharp edges is recovered indirectly by enforcement of the Kutta condition in the solution methodology. While viscous effects, such as boundary layer separation and reattachment, may be significant in a limited class of interactions, the advantages of this methodology to provide robust, efficient solutions out-weigh the significant computational overhead of a “brute force” approach using conventional, volumetric viscous flow solvers on highly refined meshes.

CFD Input and Computational Parameters

The primary input to the CFD methodology is the surface geometry representation for the ship and landing platforms. The ship airwake database will be formulated to include several ships and landing platforms that comprise the range for helicopter operations. Surface geometry descriptions are provided from the geometries used in the simulation visual database 3D models, which are specified in OpenFlight™ format. Representative surface geometries are shown in Figure 1 for LHA- and DDG-class ships.

Note that geometries used for visualization are generally not well suited for CFD analysis, since visualization geometries tend to have higher geometric detail (i.e., smaller polygons) and are generally not watertight (i.e., the surface mesh includes small gaps and overlapping panels). The geometries used for CFD analysis (illustrated in Figure 1) correspond to the lowest level of detail (LOD) for far distant visualization. To address watertightness of the corresponding visualization geometries, an extension to the computational mesh generation algorithm within the CGE model was developed and implemented. In addition, the ability to model the sea surface was also incorporated into the CGE/VorTran-M CFD methodology.

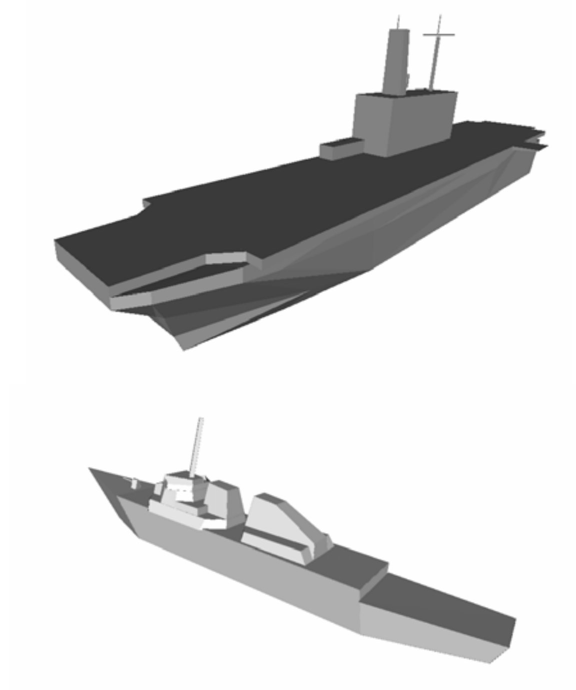
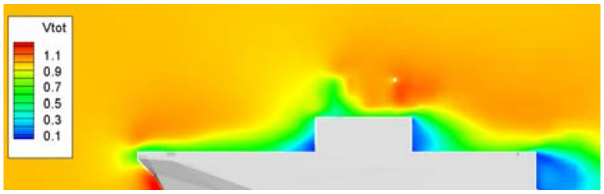


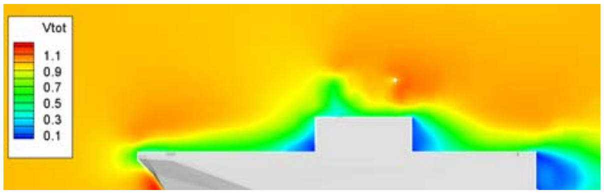
Figure 1. Representative Ship Surface Geometry Inputs for CFD Analysis

Results from the CFD analysis are provided as velocity components referenced to a ship-fixed coordinate system. The velocity components can be separated into steady and time-varying components, where the steady components represent the time-averaged flow field containing only spatial variations (gradients). Representation of the unsteady velocity components will be discussed in a subsequent section of this paper. CFD results are tabulated as a function of ship-referenced position and the wind over deck (WOD) angle suitable for interpolation for integration with the flight simulation.

Note that the unsteady flow field due to the ship airwake depends on the WOD velocity magnitude, but it is not necessary to explicitly include the WOD velocity in the tabulated results from the CFD analysis. It can be shown that for inviscid analysis of the ship airwake, numerical results scale with the WOD velocity (i.e., the computed velocity components and time can be non-dimensionalized by the WOD velocity). This observation permits flow field calculations to be performed for a single WOD velocity condition and to scale these results to a user-specified velocity in the simulation. This result has been verified by performing calculations for two different WOD velocities (same WOD angle), as illustrated in Figure 2. In this result, the instantaneous flow field, normalized by the free stream velocity, is shown for the LHA-class ship as contours in a plane aligned with the ship axis but offset to the starboard of the ship centerline by approximately 5 meters. The normalized velocity contours are nearly identical (very minor differences can be observed near the bow and behind the island masts), confirming this result. Note that this result has also been confirmed by independent analysis using a viscous flow Navier-Stokes model (Polsky, 2002).



30 m/s, 330 degree WOD condition



20 m/s, 330 degree WOD condition

Figure 2. Comparison of Instantaneous Normalized Flow Field for Two WOD Velocity Conditions

CFD Results and Methodology Validation

Calculations, using the geometries shown in Figure 1, have been performed using the hybrid CGE/VorTran-M methodology to validate its application to formulating the ship airwake database. Validation calculations have been performed to permit correlation of the CFD results against experimental data, including subscale wind tunnel data and at-sea measurements. Validation of the CFD methodology has primarily focused on the LHA-class ship configuration.

Results from the CFD methodology have been compared against unpublished wind tunnel data made available by U.S. Navy personnel (Long, 2001). The wind tunnel data were obtained from a 1/120th scale test program performed at NASA Ames Research Center. Time-averaged measurements of the flow field were made using a seven-hole pressure probe, providing flow surveys for three components of the local flow velocity. Flow surveys were made at measurement locations aligned with several landing spots (see Figure 3) for different WOD directions. The wind tunnel velocity was 170 ft/sec, and measured velocity components were normalized based on the wind tunnel velocity. Note that the CFD calculations have been performed using the geometry shown in Figure 1, which differs from the wind tunnel geometry.

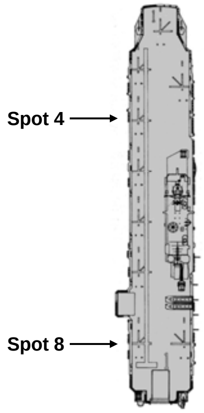
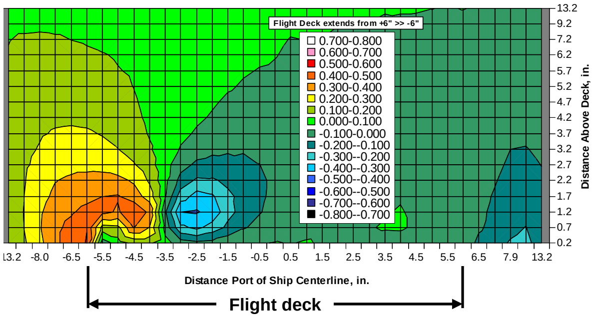


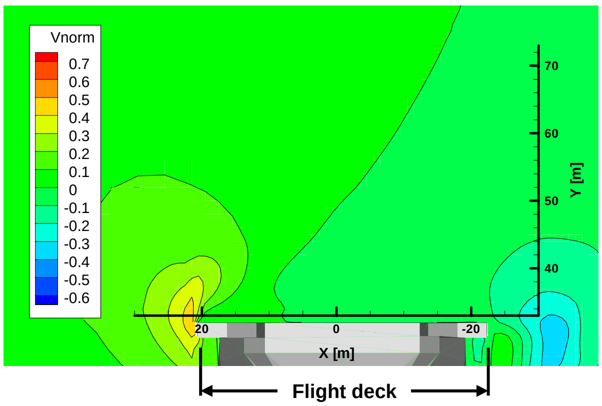
Figure 3. Measurement Locations for CFD Methodology Validation Calculations

Figures 4 and 5 illustrate the comparison between experimental data and CFD for a 20° WOD condition at two measurement locations. Correlation is shown for the vertical velocity component (approximately normal to the rotor tip path plane for typical flight operations) since this velocity component will directly affect the rotor inflow and thrust/moment generation. Comparisons have also been made with the in-plane velocity components with reasonable results. The measured flow field indicates the presence of a shed vortical structure near the starboard side of the flight deck at the Spot 4 location (Figure 4), which is captured qualitatively in the CFD prediction. Good comparison is also seen in the magnitude of the upflow and downflow around the flight deck. Near Spot 8 (Figure 5), the flow field is characterized by a large region of swirling flow, which is well predicted in magnitude and location by the CGE/VorTran-M methodology.

In general, correlation of the CFD methodology with 1/120th scale wind tunnel measurements demonstrates good qualitative and quantitative comparison. Better correlation is seen at locations behind the island where the flow field is characterized by bluff body separation and vortex shedding. Near the forward landing spots, where the velocity field is governed by the separation

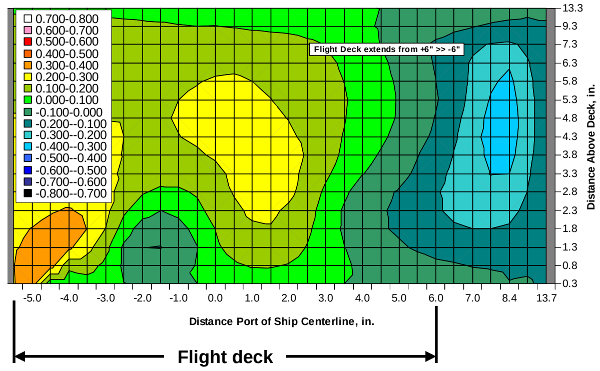


Time Averaged Wind Tunnel Data

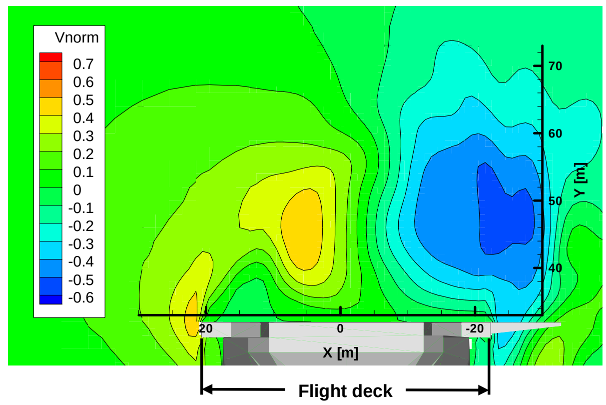


Instantaneous CFD Prediction

Figure 4. Comparison of Normalized Vertical Velocity for 1/120th Scale LHA Wind Tunnel Data and CFD Prediction, Spot 4, 20° WOD Condition



Time Averaged Wind Tunnel Data



Instantaneous CFD Prediction

Figure 5. Comparison of Normalized Vertical Velocity for 1/120th Scale LHA Wind Tunnel Data and CFD Prediction, Spot 8, 20° WOD Condition

and reattachment of the flow near the flight deck leading edge, the CFD methodology has been found to overpredict the velocity deficit. This result is not surprising given that the flow field in this region is more sensitive to localized viscous effects that are neglected in the analysis. Recent CFD results using Navier-Stokes analysis (Polsky, 2007) have also indicated a sensitivity of the flow field in this region to the atmospheric boundary layer, which is not modeled in this study.

The CFD methodology has also been validated using at-sea velocity measurements obtained as part of the JSHIP program (Polsky and Bruner, 2000, Polsky, 2002). This experimental database includes measurements of the three velocity components using anemometers located 20-feet above the flight deck of an LHA at several locations. The unsteady flow field was sampled at a single point, and data from four three-axis anemometers were recorded simultaneously, permitting correlation with unsteady velocity predictions for the CFD methodology.

Predictions of the unsteady flow field were performed using the CGE/VorTran-M methodology by initially converging the CGE model to a steady-state condition. Time-accurate calculations were then performed for a finite simulation time. For validation calculations presented here, the WOD velocity was 30 m/s, and calculations were performed for 30 seconds of simulated time. During the initial part of the CFD simulation, the VorTran-M airwake was evolving, and quasi-steady state conditions were achieved after approximately 10 seconds. Analysis of the predicted unsteady flow field was performed using CFD data between 10 and 30 seconds.

Comparison of the vertical velocity autospectra for the predicted and measured time histories is shown in Figure 6 for a measurement location above Spot 5. The measured and predicted velocity components are normalized by the WOD velocity, and the frequency axis is normalized by the velocity and ship beam width to account for differences in the WOD (free stream velocity) conditions. Good agreement can be seen in

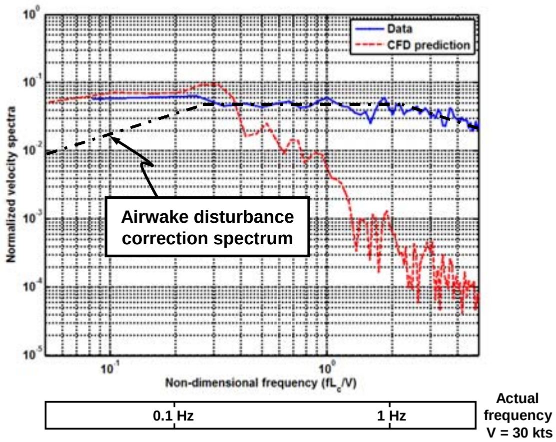


Figure 6. Comparison of the Predicted and Measured Vertical Velocity 20-feet for LHA-class Ship above Spot 5

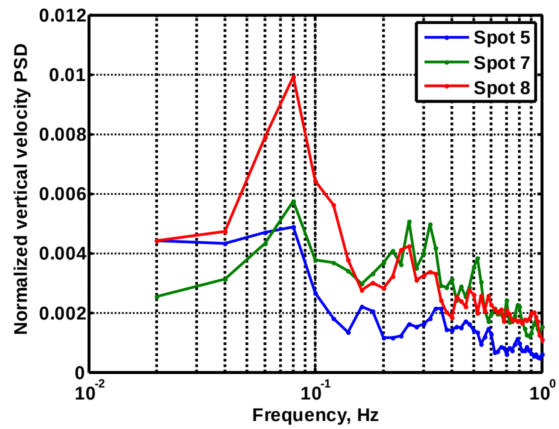
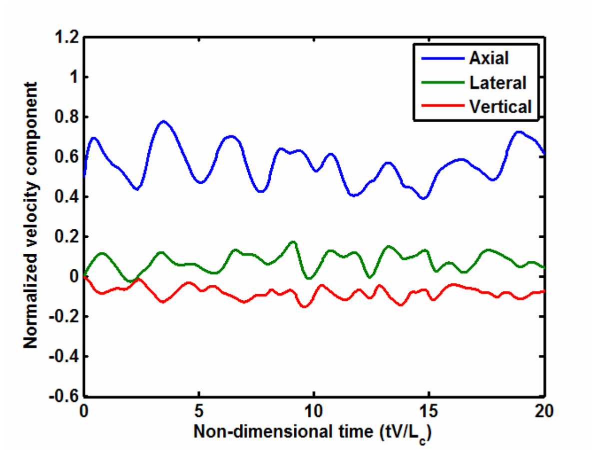


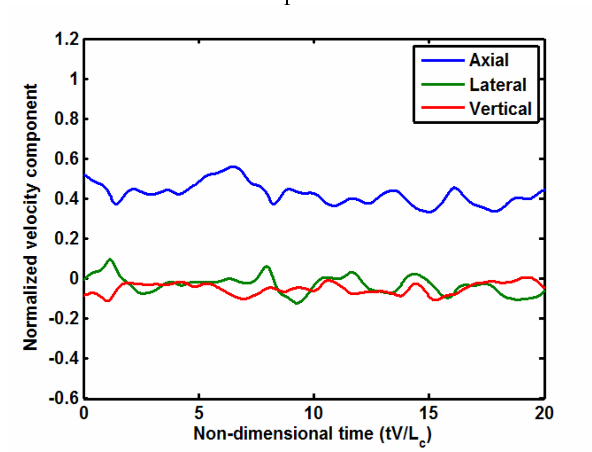
Figure 7. Normalized Velocity Autospectra at Different Landing Spots for Measured At Sea Data (25 knots, 20-degree WOD condition)

the low frequency range, but the predicted spectrum is observed to roll-off faster than the data, which is similar to previous ship airwake flow simulations (Lee et al., 2005) and is believed to be a result of neglecting viscous effects. Free-air atmospheric turbulence, which is present in the at sea measurements, but neglected in the CFD simulations, may also contribute to this discrepancy. Correction for this discrepancy is possible by including an additional disturbance generated by filtering random noise through an empirically-determined band-pass filter, indicated by the correction shown in Figure 6. It is anticipated that this high-frequency disturbance correction should have minimal impact on the helicopter response.

Analysis of the predicted and measured time histories have been performed to determine the fundamental characteristic shedding frequency. Examination of the autospectra for the measured velocity components from at-sea testing (Figure 7) indicates that the characteristic frequency is between 0.08 and 0.1 Hz for the WOD condition of 25 knots and 20 degrees. This frequency corresponds to a Strouhal number between 0.23 and 0.28 based on a characteristic length of 120 feet, which is the approximate beam width of the LHA. Estimation of the predicted shedding frequency from the velocity fluctuations shown in Figure 8 and scaling this frequency by the ratio of the free stream velocities from the at-sea data and computed results, the predicted and measured characteristic frequencies can be compared, as shown in Table 1. The agreement of the measured and predicted (scaled) frequencies is good, indicating that the CGE/VorTran-M methodology is able to predict this fundamental shedding frequency behavior, which will be more significant to the helicopter flight dynamics response.



Spot 7



Spot 8

Figure 8. Predicted Normalized Velocity Component at Several Locations for LHA

Table 1. Comparison of Fundamental Shedding Frequency for LHA-class Ship

	Shedding Frequency in Hz		
	Measured	**CFD estimated**	**Scaled CFD**
Spot 7	0.1	0.19	0.08
Spot 8	0.08	0.13	0.06

AERODYNAMIC MODEL CFD INTEGRATION

Evaluation points are provided to the CFD airwake database in ship coordinates corresponding to the fuselage center of pressure, the tail rotor hub, and five points on the rotor disk.

The evaluation points for the fuselage and the tail rotor hub are then transformed from the body coordinate system to the ship coordinate system using the following coordinate transformation:

$$\vec{X}_{ship} = [R]^{s,i}[R]^{i,b}\vec{X}_{body} \tag{3}$$

where $\vec{X}_{ship}$ is the evaluation point in ship coordinates, $\vec{X}_{body}$ is the evaluation point in body coordinates, $[R]^{i,b}$ is the rotation matrix from body to inertial coordinate system, and $[R]^{s,i}$ is the rotation matrix from inertial to ship coordinate system. The steady and unsteady ship airwake velocity components at each evaluation point are summed in the ship coordinate system and then resolved in the aircraft body coordinate system. The airwake velocity components in body coordinate system are obtained by the following transformation:

$$\vec{V}_{body} = [R]^{b,i}[R]^{i,s}\vec{V}_{ship} \tag{4}$$

where $\vec{V}_{body}$ and $\vec{V}_{ship}$ are the airwake velocity components in the body and ship coordinate system respectively. $[R]^{i,s}$ and $[R]^{b,i}$ are the rotation matrices from ship to inertial coordinate system and from inertial to body coordinate system respectively. The airwake velocity components are added in the fuselage, vertical tail, and horizontal tail dynamic pressure calculation and to the tail rotor inflow calculation.

To integrate the ship airwake disturbance into the ARMCOP rotor disk model the ship airwake velocity perturbation is evaluated at five points over the rotor disk plane, corresponding to the rotor hub center and the 75% radius at the 0, 90, 180, and 270-degree azimuth locations. The main rotor evaluation points are transformed from disk-path plane coordinate system to ship coordinate system as follow:

$$\vec{X}_{ship} = [R]^{s,i}[R]^{i,b}[R]^{b,h}[R]^{h,d}\vec{X}_{disk} \tag{5}$$

where $\vec{X}_{disk}$ is the location of the rotor evaluation point in tip-path-plane coordinate system, $[R]^{b,h}$ is the rotation matrix from rotor hub coordinate system to body coordinate system, and $[R]^{h,d}$ is the rotation matrix from tip-path-plane to rotor hub coordinate system.

The velocity perturbations at these five locations are used to determine an equivalent rotor inflow perturbation $v_{sh}(r,\psi_a)$ in terms of the uniform and harmonic inflow components. The average of all five rotor evaluation points yield the uniform inflow perturbation $v_{0,sh}$ and the fore-aft and side-to-side gradients are used to determine the harmonic inflow perturbations $v_{c,sh}$ and $v_{s,sh}$. The inflow perturbations are included directly into the main rotor dynamic inflow calculation (equation 2).

DATABASE IMPLEMENTATION ISSUES

The CGE/VorTran-M methodology provides predictions of the unsteady flow field at locations relative to the ship for a finite time period. This time period is limited by computational and data storage requirements. Similarly, storage requirements for the ship airwake database files must also be considered to avoid using all available resources for the ship airwake look-up tables or to require implementation of a memory paging schemes. To this end, an approach to representing the unsteady velocity components with a minimal parameter set has been examined to reduce storage/memory requirements. This approach uses a stochastic representation of the unsteady velocity components and a data reconstruction algorithm that preserves the spatial correlation relationships due to the vortex structures in the ship airwake. A similar approach to the representation of (spatially-correlated) atmospheric disturbances has been previously developed at NASA Ames Research Center

(McFarland, 1997). More recently, stochastic modeling approaches have been examined for application to helicopter shipboard operations (Gaonkar, 2007; Xin and He, 2007).

The approach used to reconstruct the time histories in the ship airwake database is given as follow. First, CFD computations produce time histories for the flow field velocity components at each output grid point. The mean value at a given location is determined and subtracted from the time histories, resulting in a turbulent fluctuation time history. Next, time histories for the turbulent fluctuation velocity components are used to determine an equivalent spectral filter. Finally, spatial correlation effects are represented by correlation coefficients that are determined from turbulent fluctuation time histories between neighboring spatial locations. Correlation coefficients are non-zero in general since the velocity at neighboring locations will be related due to the shed vortical structures in the wake.

The above velocity reconstruction approach leverages the observation that the predicted (and measured) velocity spectra do not vary significantly over a range of spatial locations relative to the ship (see Figure 9, where ΔR is distance downrange from Spot 7 in meters). Therefore, it is not necessary to store the complete spectra or time histories at each output grid point to retain the physically-observed frequency content in the unsteady flow field. Ideally, the filter structure and coefficients are relatively insensitive to position in the airwake, although multiple filters can be used as needed to account for the slow evolution of the airwake structure (i.e., to an observer moving with the airwake at the speed and direction of the relative wind). A comparison of the original and reconstructed time histories at two representative measurement locations is shown in Figure 10. These locations differ spatially by approximately one H-60 rotor radius, and examination of Figure 10 indicates that spatial correlations are also qualitatively captured in the reconstructed data.

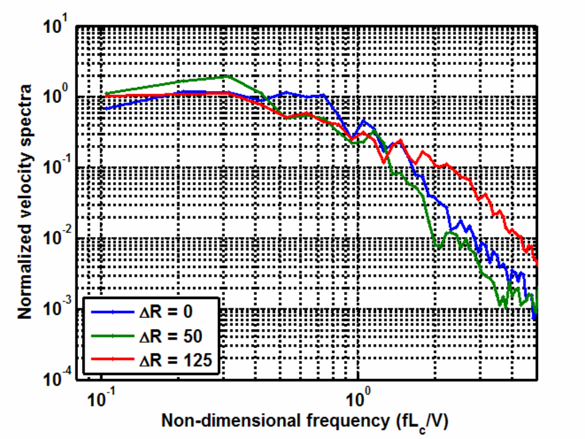


Figure 9. Normalized Predicted Velocity Spectra at Locations Relative to Spot 7

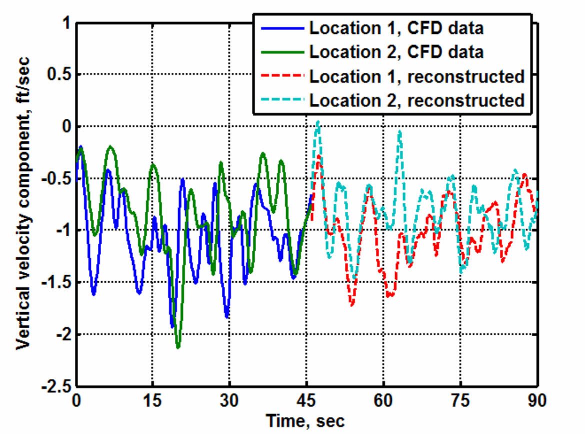


Figure 10. Comparison of Original CFD Results and Reconstructed Time Histories at Two Locations for LHA near Spot 7

CONCLUDING REMARKS

At the time of writing, the initial training device to receive this solution has completed detailed design, including technical review of the described CFD airwake solution. The program is entering the implementation and integration phase, during which the CFD database loading and infrastructure software will be developed and the real-time flight dynamics software will be modified to incorporate the CFD ship airwake effects into the flight model. Navy preliminary evaluation of the resulting flight performance is soon to follow, along with corresponding tuning activities. In parallel and off-line, approximately 15 landing platform CFD airwake databases will be generated using the CFD methodology described above.

Areas of further research include methods for handling dynamically changing wind over deck angles, effects due to sea state, and addition of unique discrete flow generators (e.g. exhaust vent thrust). In addition, it is also of interest to tailor the methodology for use with ship-based fixed-wing and VSTOL aircraft.

ACKNOWLEDGEMENTS

The authors would like to thank NAVAIR for their support, including providing valuable inputs from the Fleet H-60 helicopter pilots, the Manned Flight

Simulator laboratory, and from the engineering teams involved in previous Navy sponsored CFD Ship Airwake research. The authors would also like to acknowledge Richard Brown of Glasgow University for his contributions to the VorTran-M development and application.

REFERENCES

Alpman, E., Long, L. N., Bridges, D. O., and Horn, J. F. (2007). "Fully-Coupled Simulations of the Rotorcraft / Ship Dynamic Interface," American Helicopter Society 63rd Annual Forum, Virginia Beach, VA.

Arunajatesan, S., Shipman, J., and Sinha, N. (2004). "Numerical Modeling of Coupled VSTOL-Ship Airwake Flowfields," Proceedings of FEDSM'04: 2004 ASME Heat Transfer/Fluids Engineering Summer Conference, Charlotte, NC.

Bogstad, M. C., Habashi, W. G., Akel, I., Ait-Ali-Yahia, D., Giannias, N., and Longo, V. (2002). "Computational Fluid Dynamics Based Advanced Ship Airwake Database for Helicopter Flight Simulators," *Journal of Aircraft*, *39*, 830-838.

Brown, R. E. (2000). "Rotor Wake Modeling for Flight Dynamic Simulation of Helicopters," *AIAA Journal*, *38*, 57-63.

Bunnell, J. W. (2001). "An Integrated Time-varying Airwake in a UH-60 Black Hawk Shipboard Landing Simulation," AIAA 2001-4065, AIAA Modeling and Simulation Technologies Conference and Exhibit, Montreal, Canada.

Carico, D. and Madey, S. L., Jr. (1984). "Dynamic Interface – Conventional Flight Testing Plus A New Analytical Approach," Proceedings of the American Helicopter Society Conference on Helicopter Testing Technology, Williamsburg, VA.

Chen, R. (1979). A Simplified Rotor System Mathematical Model for Piloted Fight Dynamics Simulation. *NASA Technical Memorandum*, 78575.

Gaonkar, G. H. (2007). "Toward a Complete Stochastic Model of Airwake Turbulence for Helicopter Shipboard Operation," American Helicopter Society 63rd Annual Forum, Virginia Beach, VA.

Healy, J. Val. (1987). "The Prospects for Simulating the Helicopter/Ship Interface," *Naval Engineers Journal*, *99*, 45-63.

Lee, D., Sezer-Uzol, N., Horn, J. F., and Long, L. N. (2005). "Simulation of Helicopter Shipboard Launch and Recovery with Time-Accurate Airwakes," *Journal of Aircraft*, *42*, 448-461.

Lee, D. (2005). Simulation and Control of a Helicopter Operating In a Ship Airwake. Ph.D. thesis, Georgia Institute of Technology, August 2005.

Liu, J. and Long, L. (1998). "Higher Order Accurate Ship Airwake Predictions for the Helicopter/Ship Interface Problem," American Helicopter Society 54th Annual Forum, Washington DC.

Long, K. (2001). Private communications.

McFarland, R.E. (1997). "Finite Element Aircraft Simulation of Turbulence," NASA TM 110437.

McKillip, R. M., Jr., Boschitsch, A. H., Keller, J. D., and Wachspress, D. A. (2002). "Dynamic Interface Simulation Using a Coupled Vortex-based Ship Airwake and Rotor Wake Model," American Helicopter Society 58th Annual Forum, Montreal, Canada.

Pitt, M, and Peters, D. (1981). Theoretical Prediction of Dynamic-Inflow Derivatives. *Vertica*, *5*, 21-24.

Polsky, S. and Bruner, C. (2000), "Time-accurate Computational Simulations of an LHA Ship Airwake," AIAA 2000-4126.

Polsky, S. (2002). "A Computational Study of Unsteady Ship Airwake," AIAA 2002-1022.

Polsky, S. (2007). Private communication.

Schrage, D. P.; Peters, D. A.; Prasad, J. V. R.; Stumpf, W. F. and He, C. (1988). "Helicopter stability and control modeling improvements and verification on two helicopters," 14th European Rotorcraft Forum, Milan, Italy.

Tai, T. (1998). "Simulation and Analysis of LHD Ship Air Wake by Navier-Stokes Method," Proceedings of Fluid Dynamics Problems of Vehicles Operating near or in the Air-Sea Interface, RTO/AVT Symposium, Amsterdam.

Talbot, P, Tinling, B, Decker, W, and Chen, R. (1982). A Mathematical Model of a Single Main Rotor Helicopter for Piloted Simulation. *NASA Technical Memorandum*, 84281.

Whitehouse, G. R., Boschitsch, A. H., Quackenbush, T. R., Wachspress, D. A., and Brown, R. E. (2007). "Novel Eulerian Vorticity Transport Wake Module for Rotorcraft Flow Analysis," American Helicopter Society 63rd Annual Forum, Virginia Beach, VA.

Xin, H. and He, C. (2007). "A Statistical Turbulence Model for Shipboard Rotorcraft Simulations," American Helicopter Society 63rd Annual Forum, Virginia Beach, VA.