Higher Order Accurate Solutions of Ship Airwake Flow Fields Using Parallel Computers

Lyle N. Long †, Jingmei Liu ‡ and Anirudh V. Modi ‡
Department of Aerospace Engineering
The Pennsylvania State University
University Park, PA 16802
lnl@psu.edu

Summary
This paper presents a new method for simulating ship airwake flow fields. These flows are inherently unsteady, and very difficult to predict. The method presented (NLDE) is fourth-order accurate in space and time. In this method we first solve for the steady state flow field, then we solve for the unsteady fluctuations. Steady and unsteady results are presented for a generic frigate shape. Considering the complex geometry of real ship, the unstructured grid approach is most useful. The mean flow results are compared with oil flow visualization photographs. Parallel computational methods are a necessity for ship air wake problems and MPI is used in the NLDE solver. The parallel performance on various computers is presented also.

Introduction
Sharp-edged box-like ship super-structures create numerous aerodynamic and fluid dynamic problems. Unsteady separated flow from sharp edges (and excessive ship motions) make landing helicopters on ships a very hazardous operation. In addition, the strong unsteady flows can cause severe rotor blade deformations. There have been numerous incidences where the helicopter blades have actually impacted the helicopter fuselage, which is called a "tunnel strike". In order to avoid this and other engage/disengage problems, determining safe operating envelopes is very costly and time consuming. On the other hand, many numerical simulation attempts have not been successful due to the inherently unsteady nature of flow and the low-speed character of the flow (which may cause numerical stiffness).

Research on ship airwakes has been conducted using several different approaches. One of the sources of relevant research is building aerodynamics which shows the general features of flow about blunt bodies of different aspect ratios. The simplest model of a ship, admittedly rather crude, is a sharp edged blunt body. The superstructure of most modern ships is very complicated, including towers, antennae, radar dishes, exhaust stacks, etc. The flow around these obstacles is very difficult to predict.

Geometrically precise studies are needed and have been done in wind tunnels. There have also been full scale tests performed by the US Navy, which gives some important information on real ship airwakes. Of course it is difficult to perform very controlled experiments on real ships. It is also difficult to measure the flow field accurately in the harsh ocean environment and in the presence of the strong electromagnetic fields on most ships.

Most wind tunnel tests include measurements made in the wake of a model ship exposed to a uniform velocity profile and almost zero turbulence level. One more realistic test was conducted at NASA Ames in the "Shipboard Simulator" with a neutrally buoyant atmospheric boundary layer.

Another reference for simulations is that by NRC-CNRC. A wind tunnel investigation of the characteristics of the airwake behind a model of a generic frigate was conducted. The wind tunnel simulation incorporated a correctly-scaled atmospheric boundary layer. Measurements of streamwise and vertical components of airwake velocity were made. Time average, standard deviations, spectral densities and time correlations are presented for both velocity components for various position in the airwake. All these experimental tests are crucial for validating numerical models.

Figs. 1 and 2 show a frigate and an LHA, respectively. These are very different ships, and their airwakes are very different also. The frigates typically carry one or two SH-2G Seasprites or SH-60B Seahawks. On the frigate we are mainly interested in studying the hangar deck area (aft portion of the ship), and the separated
flow that effects this region. On the LHA, helicopters can land on many different locations on the deck, and each of these can experience quite different flow fields. The LHA's can carry 9 CH-53D Sea Stallion or 12 CH-46D Sea Knight helicopters, and 6 AV-8B Harriers. The forward portion of the deck is primarily influenced by the separated flow off the deck edge. Very strong vortex sheets emanate from these edges. One of the authors (Long) spent three days on an LHA (U.S.S. Saipan) and helped Kurt Long measure ship airwakes. We found in some cases the flow velocity ranged from 40 knots 12 feet off the deck to zero velocity 3 feet off the deck. In the mid-section of the ship the very large island has a strong effect on the flow and tunnels the flow tangential to the island.

The need for numerical simulations comes from the high cost of determining the safe operating envelopes for helicopters in a ship environment (and the huge testing backlog). It would be very useful to have numerical methods that could accurately simulate ship airwakes. There have been other attempts at numerically simulating ship airwakes. The airwake about a DD-963 ship configuration was simulated using a steady-state flow solver based on the 3D multi-zone, thin layer Navier-Stokes method. A US navy destroyer, DDG51 was chosen to validate an unsteady inviscid solver with an unstructured grid and low-order method. No method to-date has been entirely satisfactory for predicting these flow fields.

**Flow Nature of Ship Airwake Simulation**

From previous studies, it has been shown that the key features of ship airwakes are (1) a low Mach number (about 0.05), (2) inherently unsteady flow, and (3) large regions of separated flow. The large separated regions from superstructure sharp edges are quite difficult to capture accurately. In addition, the wind conditions over rough seas have to be considered, such as, the atmospheric turbulent boundary layer and the effect of the wind/ship speed ratio on the turbulence intensity. When this ratio is increased, the turbulence intensity will decrease and its spectrum will shift to a high value in the streamwise direction. The wind direction can vary a great deal, since the air flow can impact the ship at any yaw angle (even 180 degrees). The complex ship geometry makes unstructured grid solvers and parallel computers very attractive. In this paper, preliminary attempts at high order accurate ship airwake predictions have been made by solving a steady flow field with a well-developed CFD method (CFL3D) and a perturbation field with a high-order method. The result is high-order-accurate 3D simulations. We also have some preliminary results for steady flow predictions from an unstructured solver (PUMA).

**Nonlinear Disturbance Equations (NLDE)**

The methodology used here is based on the nonlinear disturbance equations, which is a newly developed numerical method. The general Navier-Stokes equations in a Cartesian coordinate system are:

\[
\frac{\partial q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = \frac{\partial R}{\partial x} + \frac{\partial S}{\partial y} + \frac{\partial E}{\partial z}
\]

(1)

where \(F, G, \) and \(H\) are the inviscid terms and \(R, S, E\) are the viscous terms. The results presented here will
all be inviscid. The flow field is then split into a mean and a fluctuating part:

\[ q = q_0 + q' \]  

(2)

where

\[ q = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ e \end{pmatrix} \]  

(3)

and

\[ q_0 = \lim_{T \to \infty} \frac{1}{T} \int_{t_0}^{t_0+T} q(t) \, dt \]  

(4)

Substitution of equation (2) into (1) and rearranging results in the nonlinear disturbance equations (NLDE):

\[ \frac{\partial q'}{\partial t} + \frac{\partial F'}{\partial x} + \frac{\partial G'}{\partial y} + \frac{\partial H'}{\partial z} = Q \]  

(5)

Where

\[ q' = \begin{pmatrix} \rho' \\ \rho u' + \rho' u_e + \rho' u' \\ \rho v' \\ \rho w' + \rho' w_e + \rho' w' \end{pmatrix} \]  

(6)

On the left hand side of the NLDE are terms related to the perturbation properties and the cross terms (linear and nonlinear), whereas the right hand side contains strictly mean flow terms.

The convective fluxes involving the perturbation quantities \( F', G' \) and \( H' \) are given as

\[ F' = \begin{cases} 
\rho_0 u' + \rho' u_0 + \rho' u' \\
\rho' u'_2 + 2\rho_0 u u' + \rho' u_0 u' + (\rho_0 + \rho') u u' \\
\rho_0 u_0 u' + \rho_0 v u' + \rho_0 w u' + (\rho_0 + \rho') u u' \\
\rho_0 u_0 u + \rho_0 u_w u' + \rho_0 u_0 w + (\rho_0 + \rho') u u' \\
u'(e_0 + p_0) + (u_0 + u')(e' + p') 
\end{cases} \]  

(7)

\[ G' = \begin{cases} 
\rho_0 v' + \rho' v_0 + \rho' v' \\
\rho_0 v_0 v' + \rho_0 u v' + \rho_0 w v' + (\rho_0 + \rho') u v' \\
\rho_0 v_0 u' + \rho_0 u v' + \rho_0 u v + (\rho_0 + \rho') u v' \\
\rho_0 u_0 v' + \rho_0 v w v' + (\rho_0 + \rho') u v' \\
v'(e_0 + p_0) + (u_0 + u')(e' + p') 
\end{cases} \]  

(8)

\[ H' = \begin{cases} 
\rho_0 w' + \rho' w_0 + \rho' w' \\
\rho_0 w_0 w' + \rho_0 u w' + \rho_0 w v' + (\rho_0 + \rho') u w' \\
\rho_0 w_0 u' + \rho_0 u w' + \rho_0 w w + (\rho_0 + \rho') u w' \\
\rho_0 u_0 w + \rho_0 w w + (\rho_0 + \rho') u w' \\
w'(e_0 + p_0) + (u_0 + u')(e' + p') 
\end{cases} \]  

(9)

The mean flow source term \( Q \) is time independent:

\[ Q = -\left( \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} \right) + \frac{\partial R}{\partial x} + \frac{\partial S}{\partial y} + \frac{\partial E}{\partial z} \]  

(10)

The boundary conditions for the NLDE are developed by applying Thompson's characteristic method to the nonlinear disturbance equations. This methodology allows us to use the most effective algorithms for the steady and unsteady portions of field, respectively. It also minimizes round-off error since we are only computing perturbations. We can even use different grids for the steady and unsteady solution. More discussion on this new method is in the reference.15

**Characteristic Boundary Conditions for NLDE**

The boundary conditions for the NLDE are developed by applying Thompson's characteristic method to the nonlinear disturbance equations. This type of boundary condition treatment allows one to easily introduce a disturbance at the incoming boundary by deriving an expression for one of the incoming
characteristics with a source term. Atmospheric boundary layer conditions can be incorporated in this manner at incoming boundaries. At the outflow boundaries, the boundary conditions are essentially non-reflecting. The ship superstructure and ocean surface are both treated as hard wall boundary conditions.

Numerical Method and Parallel Methodology

The NLDE are cast in a generalized coordinate system and solved numerically using a finite difference based scheme. The discretized equations are solved in a time accurate manner by taking advantage of computational aeroacoustics (CAA) methods. The spatial flux derivatives are calculated using seven point stencils of the fourth order optimized Dispersion Relation Preserving (DRP) scheme of Tam and Webb. The time integration is a fourth order accurate Runge-Kutta method.

Efficient computing performance is achieved by using a three dimensional domain decomposition strategy. Efficient computing performance is achieved by using a three dimensional domain decomposition strategy. The code is written in Fortran 77 plus Message Passing Interface (MPI) and is scalable in three dimensions. As mentioned early, the ship geometry is very complicated, even for a generic frigate test model. This makes multi-block grid simulations and domain decomposition very difficult. In order to make the code scalable and flexible, a three dimensional single-block grid is used. The whole computational domain is divided into many three dimensional zones. The grid points are evenly distributed across each processor.

The NLDE solver is implemented portably on parallel computers, such as the IBM SP2 (e.g. Penn State, Npaci, MHPCC), SGI Power Challenge and Pentium II Cluster. A comparison of code performance for the ship airwake run on various machines is shown in Fig. 3. While a 24-processor IBM SP2 is 8.4 times faster than eight 260 MHz Pentium III's networked together, the SP2 costs roughly 28 times more than the PC cluster. Fig. 4 gives the wall clock time for a ship air wake case with 1.86 million grid points using various number of processors. A 64-processor SP2 is roughly 2.6 time faster than a 16-processor SP2 (when problem size is kept fixed).

Results and Discussions

In the helicopter/ship interface problem, the most important data for coupling the airwake solution to the dynamics analysis of the main rotor blades of helicopter are the mean flow field, the intensity of flow perturbations, and its dominant frequencies. Such an approach is presented in this paper. So far we have been concentrating on two types of ships: (1) frigates with helicopter landing pads on the deck behind the hangar and (2) aircraft carriers and LHA's the deck leading edge vortex and separation are the key flow phenomena.

In this paper some preliminary simulations have been done for a generic ship shape (TTCF ship). In fact, this is a generic frigate model and it is shown in Fig. 5 in a computational mesh. It is 240 feet long, 45 feet wide and 55 feet high. It was chosen because there are some experimental investigations using the same configuration. It is acknowledged that the ship superstructure does not resemble a typical frigate superstructure in detail. However, from an aerodynamic point of view the airwake should still be representative of that for an ac-
tual frigate since this study is concerned with the macro-
scopic flow properties and large scale phenomena in the
hangar wake.

Special attention is given to the helicopter landing
area which is the square, aft section of the ship. There is
a 20 feet drop down to the landing deck from the hangar
structure, which will lead to vortex shedding over the
deck causing landing approach hazards.

The computational grid for this problem is 201 x 109
x 85 which results in a grid resolution of two feet or
less in each direction around the ship, grid stretching
was used to enlarge the domain. So far both mean flow
based on the structured grids and NLDE simulations
were based on the same grid in order to avoid three di-
mensional interpolation. In fact, the NLDE needs much
fewer grid points than CFL3D.

Mean flow simulation based on
structured grids

NASA Langley and Ames research centers have de-
voted significant resources in the past decades to develop-
ning modern CFD technology. The CFL3D 5.0 pack-
age from NASA Langley is used here to simulate the
mean flow which will be given as a background flow
to the unsteady flow computation of NLDE. The code
is a Reynolds-Averaged thin-layer Navier-Stokes flow
solver for structured grids. A finite volume algorithm
with a spatial-factored diagonalised, implicit scheme is
used in discretization of the partial differential equations.
The upwind-biased-differencing using the flux-difference-
splitting technique is employed.

From the experimental results, it is known that the
flow is mostly separated, with free vortices originating
from the sharp corners. There are two types of separa-
tion: one due to viscosity and the other due to sharp
corners of the blocked structures. The former is heavily
influenced by the Reynolds number. The latter is purely
an inviscid phenomenon, independent of Reynolds num-
ber. The air wake is greatly influenced by both of them.
In this mean flow simulation, we are concerned primarily
with the inviscid phenomenon and used the Euler solver
of CFL3D. The TTCP ship computational domain is di-
vided into 10 blocks.

The Mach number chosen for the simulation is a high
wind case. The incoming flow speed is 41 knots. The wa-
ter surface is assumed to be a hard wall boundary. Fig. 6
and fig. 7 show the contour plots of velocity magnitude
on the surface of the TTCP ship for the zero and 40
degree yaw angle wind cases from CFL3D results. The
symmetry property of zero yaw angle flow is captured
very well. The flow is accelerated around the sharp cor-
ners and there are several reverse flow regions near the
walls close to each corner. After the blocked structures
there is massive flow separation; the separation line is
clearly shown after each block.

Of importance to the landing operation is the flow
condition over the flight deck. Fig. 8 depicts the contour
of velocity magnitude at the ship's center plane. It is
shown that the large region of recirculating flow extends
over the flight deck and rises higher than the hangar.
This flow region is in the landing path.

Fig. 9 shows velocity vectors in two horizontal planes
4.75 feet and 8.75 feet above the flight deck. Our numer-
ical results are compared with a flow pattern obtained
from an experimental study\textsuperscript{18} in fig. 10. It shows the
flow pattern from experiments, where four distinct flow
regions are behind the hangar. This three dimensional
vortex and reverse flow has very low speed but generally
is very unsteady and yaw-dependent. Comparing to the
experiment, the physical flow features are well captured
by the simulation. In the numerical plots, the vortex
pair in the higher plane is much close to the center line
and the hangar. This indicates that there is a horse shoe
vortex as shown by the topological drawing in fig. 10.

In fig. 11 the velocity vector on the flight deck floor is
compared with flow visualization results for the TTCP
ship\textsuperscript{23} The figure (a) is the results from CFL3D us-
ing structured grids and the figure (c) is from another
solver (PUMA) using unstructured grids, which will be
discussed in the next section. The photograph in the
middle is from oil flow visualization\textsuperscript{24} There are dif-
ferences in the attachment line and the position of the
vortex center. This is probably due to our inviscid ap-
proach.

Figure 6: Flow speed contours on the surface of TTCP
ship with 0 degree yaw angle wind (CFL3D).

A case with 40 degree yaw angle wind was also sim-
ulated. The numerical results on the front and middle
bridge-deck, and flight-deck are compared with oil vi-
sualization photographs in fig. 12, fig. 13 and fig. 14.
The three dimensional separation lines are clearly shown.
Comparing fig. 12 (a) and (b) the flow patterns on the
front part of bridge deck are very similar and close to each other since this part of the flow is less influenced by the viscosity and our nonviscous simulation is more capable of accurately capturing the flow. Comparing fig. 13 (a) and (b) and fig. 14 (a) and (b), the flow nature is partially captured since the viscous effects become stronger downstream. In the next section, the same comparisons are made for a 30 degree yaw angle case using the unstructured grid solver PUMA. Finally in fig. 15 the performance of CFL3D (a serial code) is shown. We use 1.86 million grid points and for a typical run it needs about 10,000 time iterations. For this TTCP ship configuration it takes days to get the steady state results using an SGI Power Challenge (Single Processor).

**Mean flow simulation based on the unstructured grids**

While we have simulated relatively simple geometries here, more complex geometries (such as the 'island' or control tower on the LHA) are important. Preliminary results using unstructured grids are discussed in this section.

For the unstructured grid flow field predictions we are using the PUMA code¹ from Dr. Chris Bruner (NAWC). PUMA (Parallel Unstructured Maritime Aerodynamics) is a computer program for the analysis of internal and external non-reacting compressible flows over arbitrary geometries. PUMA is written entirely in ANSI C and uses MPI (Message Passing Interface) to ensure high portability and good performance.

PUMA is based on FVM (Finite Volume Method) and supports mixed-topology unstructured grids composed of tetrahedra, wedges, pyramids and hexahedra. The code may be run so as to preserve time accuracy for unsteady problems, or may be run using a pseudo-unsteady formulation to enhance convergence to steady-state. Either explicit or implicit time integration may be used. Primitive flow quantities (density, velocity, and pressure) are computed at the cell centers and saved on exit.

Because PUMA uses DMA (Dynamic Memory Allocation), problem size is limited only by the amount of memory available on the machine. Since PUMA is targeted for distributed-memory parallel computers, which usually have an abundance of memory, little effort has been

---

¹. PUMA code is available from Dr. Chris Bruner, Naval Air Warfare Center (NAWC).
placed on reducing PUMA's memory requirements. Currently, with double precision floating point variables used throughout the code, PUMA needs 582 bytes/cell and 624 bytes/face, not including message passing overhead. For tetrahedral grids, this amounts to 2002 bytes/cell (or 250 words/cell). This requirement can be reduced significantly by compiling the code using single precision floating point variables for which an option is provided.

We ran the TTCP ship in PUMA by converting the structured grids (used in CFL3D and NLDE code) to unstructured tetrahedral grids for PUMA. There are 1,718,080 cells and 1,769,837 nodes used in PUMA. A case with 30 degree yaw angle wind is simulated using PUMA. The numerical results on the front and middle part bridge-deck, and flight-deck are compared with oil flow visualization photographs in fig. 16, 17 and 18. The flow pattern at all three locations are similar to the experimental results, such as the vortex location after the island in fig. 17 and the three dimensional separation line is very similar in fig. 18.

Based on the discussion above, in fact, the flow pattern shown represents a very rough mean of the flow. They are intended only to give approximate envelopes for the different regions and provide the background flow for NLDE simulations. There are large fluctuations about this mean flow. The flow field is generally very unsteady. In the following section, the results from NLDE simulations are discussed based on this mean flow.

**Perturbation simulation**

As mentioned before, NLDE has been developed for solving more complex geometries, such as the TTCP ship, which allow multi-solid-boxes inside the computational domain. This makes the boundary condition implementation difficult, especially combined with a domain decomposition parallel technique. To overcome this difficulty, a single block domain was chosen for the NLDE code. The TTCP ship is divided into 88 solid boxes. Characteristic boundary conditions are used at the surfaces, edges and corners of these boxes. At each time step, after the single block computation is finished, the solid box wall boundary conditions are applied to update the value at wall grid points.

The high wind speeds relevant to the ship/helicopter interface problem arise from storm centers far from the actual ship and are called neutrally stratified. This wind condition is considered at our inflow boundary. The
principal parameters of the free stream airflow are (1) the mean wind speed, time-averaged over an appropriate scale; (2) the turbulence intensity; (3) the longitudinal (or integral) length scale of the turbulent velocity fluctuations. Empirical relationships are available (ESDU data items 74030, 74031) for the above four parameters as a function of the mean wind speed, elevation and roughness length scale.3

Incoming characteristics with source terms are introduced from the inflow boundary. The magnitude of the incoming disturbance is determined by the turbulence intensity. Its spectrum is obtained from the wind spectrum by using a random walk (random phase) technique.

Fig. 19 and 20 give contour plots of longitudinal velocity perturbations $u'$ in the center plane at two different time steps. Fig. 21 gives contour plots of the longitudinal velocity intensity $\sqrt{u'^2}$. From these unsteady results, it is shown that large perturbations occur around the TTCP ship structure, especially in the area after the hangar and after the leading edge. In the field far from the ship the flow is quite steady. Vortex shedding from the hangar can be clearly observed.

In fig. 22 and 23, contour plots of vertical and transverse perturbation velocity $v'$, $w'$ are shown. In fig. 24 the vertical perturbation intensity $\sqrt{v'^2}$ is given in the center plane. High instantaneous vertical perturbations are found in the region just after the hangar trailing edge. Since this is a zero yaw angle case, the transverse pertur-
Figure 13: Predicted surface flow velocities on the middle part of bridge-deck compared to surface oil flow images with 40 degree yaw angle wind.\(^1\) (a) CFL3D, (b) oil flow visualization. The unsteady three-dimensional flow is of interest throughout the domain but in particular the flow unsteadiness is important around the helicopter landing deck. Fig. 25 presents a contour plot of perturbation intensity in a horizontal plane 17 feet above the deck, where the helicopter rotor would be.

From those preliminary results, the unsteady features of a TTCP ship air wake are captured qualitatively. However, detailed experimental data is not yet available. In the meantime, the NLDE code is being improved and prepared for quantitative evaluation and analysis.

**Concluding Remarks**

This paper presents steady and unsteady flow field predictions for frigate class ships. A nonlinear disturbance equation solver has been developed using parallel computers. The parallel performance of the code has been compared on various computers. By comparing to experimental results, our present results are qualitatively correct, and show that the key flow phenomena can be captured using a steady-state code followed by the NLDE code.

The unstructured grid approach can be used for more complex geometry and practical simulations, such as a real ship shape.

Future work will concentrate on more detailed comparisons to experiment, the inclusion of more geometrical features of the ships, and the inclusion of viscous effects.

**Acknowledgments**

We gratefully acknowledge ONR Grant No. N00014-97-1-0530. We would also like to thank Steve Zan (NRC Canada) and Kurt Long (USN Pax River).

**References**


Figure 15: Timing for CFL3D on several computers.


Figure 16: Predicted surface flow velocities on the front part of bridge-deck compared to surface oil flow images with 30 degree yaw angle wind.24 (a) PUMA, (b) oil flow visualization


Figure 17: Predicted surface flow velocities on the middle part of bridge-deck compared to surface oil flow images with 30 degree yaw angle wind. (a) PUMA, (b) oil flow visualization.

Figure 18: Predicted surface flow velocities on the flight-deck compared to surface oil flow images with 30 degree yaw angle wind. (a) PUMA, (b) oil flow visualization.


Figure 19: Contour of instantaneous longitudinal velocity perturbations at $t = t_1$ (NLDE).

Figure 20: Contour of instantaneous longitudinal velocity perturbations at $t = t_2$ (NLDE).

Figure 21: Contour of longitudinal velocity intensity.
Figure 22: Contour of instantaneous transverse velocity perturbations (NLDE).

Figure 23: Contour of instantaneous vertical velocity perturbations (NLDE).

Figure 24: Contour of vertical velocity intensity (NLDE).
Figure 25: Contour plots of perturbation velocity intensity in the rotor plane of helicopter (NLDE).