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

Jingmei Liu $^{\dagger}$, Lyle N. Long $^{\dagger}$ and Anirudh Modi $^{\dagger}$
Department of Aerospace Engineering
The Pennsylvania State University
University Park, PA 16802
lnl@psu.edu

Abstract

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. The parallel method is a necessity for ship airwake problems and MPI is used in the NLDE solver. The parallel performance on various computers is compared 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 of this flow field 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 NRCC. 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.

The wind tunnel tests have to suitably scale the environment and structure to model size, make the appropriate measurements in the wake of the model and then rescale the results back to full size. All these experimental tests are crucial for validating numerical models. Wind tunnel tests can be quite costly, but flow measurements on real Naval ships are very difficult and costly to obtain.

Figs. 1 and 2 show a frigate and an LHA, respec-
tively. These are very different ships, and their airwakes are very different also. The frigates typically carry one or two SH-2G Seaspites 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 Stallions 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 very 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, DDG 51 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

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:

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

**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\textsuperscript{18}) and a perturbation field with a high-order method. The result is high-order-accurate 3D simulations. We will only show structured grid results herein, but we are also pursuing higher-order accurate unstructured solvers (e.g. based on PUMA\textsuperscript{1}).

**Nonlinear Disturbance Equations (NLDE)**

The general Navier-Stokes equations in a Cartesian coordinate system are:

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

**Figure 1:** Oliver Hazard Perry Class Guided Missile Frigate (length=445 feet, beam=45 feet)

**Figure 2:** Tarawa Class LHA (length = 820 feet, beam = 132 feet)
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:

\[
q = q_o + q
\]  

(2)

where

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

(3)

and

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

(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

\[
q' = \begin{bmatrix}
\rho' u' + \rho' v_o + \rho' w_o \\
\rho' u_o + \rho' v_o + \rho' w_o \\
\rho' u_o + \rho' v_o + \rho' w_o \\
e'
\end{bmatrix}
\]  

(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

\[
F' = \begin{bmatrix}
\rho' u' + \rho' u_o + \rho' w_o \\
\rho' u_o + \rho' v_o + \rho' w_o + \rho' v_o + \rho' w_v + \rho' w_o + (\rho_o + \rho') u' w' \\
\rho' u_o + \rho' v_o + \rho' w_v + \rho' u_v + \rho' w_v + (\rho_o + \rho') u' w' \\
u'_o (\rho_o + p_o) + (u_o + u')(e' + p')
\end{bmatrix}
\]  

(7)

\[
G' = \begin{bmatrix}
\rho' v' + \rho' v_o + \rho' w_o \\
\rho' v_o + \rho' v_o + \rho' w_o + \rho' u_v + \rho' u_v + \rho' w_v + (\rho_o + \rho') v' w' \\
\rho' u_v + \rho' u_v + \rho' w_v + \rho' u_v + \rho' w_v + (\rho_o + \rho') u' w' \\
u'_v (\rho_o + p_o) + (u_o + u')(e' + p')
\end{bmatrix}
\]  

(8)

\[
H' = \begin{bmatrix}
\rho' w_o + \rho' u_w + \rho' w_v \\
\rho' w_o + \rho' w_v + \rho' u_v + \rho' w_v + \rho' u_v + \rho' w_v + (\rho_o + \rho') u' w' \\
\rho' w_v + \rho' u_v + \rho' w_v + \rho' u_v + \rho' w_v + (\rho_o + \rho') u' w' \\
u'_w (\rho_o + p_o) + (u_o + u')(e' + p')
\end{bmatrix}
\]  

(9)

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

\[
Q = - \left( \frac{\partial F_o}{\partial x} + \frac{\partial G_o}{\partial y} + \frac{\partial H_o}{\partial z} + \frac{\partial R_o}{\partial x} + \frac{\partial S_o}{\partial y} + \frac{\partial E_o}{\partial z} \right)
\]  

(10)

If the NLDE is time averaged, it becomes the Reynolds-averaged Navier-Stokes equation, where the Reynolds’s stresses are on the left hand side. Thus, for a laminar flow \( Q = 0 \).

We seek a solution of the perturbation variables \( q' \) with a known mean flow field which can be obtained from existing well-developed CFD codes (e.g. CFL3D, \textsc{INS3D}, \textsc{OVERFLOW}, \textsc{PUMA}, \textsc{SST}, ...) for steady flow. 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.\textsuperscript{14}

**Characteristic Boundary Conditions for NLDE**

The boundary conditions for the NLDE are developed by applying Thompson’s characteristic method\textsuperscript{21} to the nonlinear disturbance equations. Instead of using the conservative form of the equations, the boundary conditions are derived based on the nonconservative form:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial u_j}{\partial x_j} + u_j \frac{\partial \rho}{\partial x_j} = 0
\]  

(11)
\[ \frac{\partial p}{\partial t} + \gamma p \frac{\partial u_j}{\partial x_j} + u_j \frac{\partial p}{\partial x_j} - V I S_p = 0 \] (12)

\[ \rho \frac{\partial u_i}{\partial t} + p \frac{\partial u_i}{\partial x_j} + \frac{\partial p}{\partial x_j} - V I S_{vel} = 0 \] (13)

Here \( V I S_p \) and \( V I S_{vel} \) are viscous terms. Substitution of equation (2) into (11) - (13) and rearranging the results gives the boundary conditions in nonlinear perturbation form:

\[ \frac{\partial u_i'}{\partial t} + (\rho_0 + p') \frac{\partial u_i'}{\partial x_j} + (u_{0j} + u_j') \frac{\partial u_i'}{\partial x_j} = \] 
\[ -(p' \frac{\partial u_{0j}}{\partial x_j} + u_j \frac{\partial p_0}{\partial x_j}) - (\rho_0 \frac{\partial u_{0j}}{\partial x_j} + u_{0j} \frac{\partial p_0}{\partial x_j}) \] (14)

\[ \frac{\partial p'}{\partial t} + \frac{1}{2} p' \frac{\partial p'}{\partial x_j} + \rho_0 \frac{\partial p_0}{\partial x_j} = \] 
\[ -(\rho_0 \frac{\partial u_{0j}}{\partial x_j} + u_j \frac{\partial p_0}{\partial x_j}) - (\rho_0 \frac{\partial u_{0j}}{\partial x_j} + u_{0j} \frac{\partial p_0}{\partial x_j}) \] (15)

Here we apply a characteristic analysis to the perturbation variables and move the mean flow terms to the right hand side, which is different than the approach employed by others. Now considering a boundary located at \( x = x_0 \) and using the characteristic analysis\(^2\) to modify the hyperbolic terms of Eqs. (14) - (15) corresponding to waves propagating in the \( x \) direction, we can recast this system as:

\[ \frac{\partial u_i'}{\partial t} + (u_{0j} + u_j') \frac{\partial u_i'}{\partial x_j} + \frac{1}{\rho_0 + p'} \frac{\partial p'}{\partial x_j} = \] 
\[ -(u_j \frac{\partial u_{0j}}{\partial x_j} + \frac{1}{\rho_0 + p'} \frac{\partial p_0}{\partial x_j}) - u_{0j} \frac{\partial u_{0j}}{\partial x_j} \] (16)

\[ \frac{\partial u_i'}{\partial t} + \frac{1}{c^2} [L_2 + \frac{1}{2}(L_1 + L_5)] + (\rho_0 + p') \frac{\partial u_i'}{\partial y} + \frac{\partial u_i'}{\partial z} = -[\gamma p \frac{\partial u_{0j}}{\partial x} + \frac{\partial u_{0j}}{\partial y} + \frac{\partial u_{0j}}{\partial z}] + \] 
\[ (v_0 + v') \frac{\partial u_i'}{\partial y} + (w_0 + w') \frac{\partial u_i'}{\partial z} = \] 
\[ -(\rho_0 \frac{\partial u_{0j}}{\partial x} + \frac{\partial u_{0j}}{\partial y} + \frac{\partial u_{0j}}{\partial z}) - u_{0j} \frac{\partial u_{0j}}{\partial x} + \frac{\partial u_{0j}}{\partial y} + \frac{\partial u_{0j}}{\partial z} \] (17)

\[ \frac{\partial u_i'}{\partial t} + \frac{1}{c^2} [L_2 + \frac{1}{2}(L_1 + L_5)] + (\rho_0 + p') \frac{\partial u_i'}{\partial y} + \frac{\partial u_i'}{\partial z} = \] 
\[ -(\rho_0 \frac{\partial u_{0j}}{\partial x} + \frac{\partial u_{0j}}{\partial y} + \frac{\partial u_{0j}}{\partial z}) - u_{0j} \frac{\partial u_{0j}}{\partial x} + \frac{\partial u_{0j}}{\partial y} + \frac{\partial u_{0j}}{\partial z} \] (18)

\[ \frac{\partial u_i'}{\partial t} + \frac{1}{2} (\rho_0 + p') v (L_5 - L_1) + (v_0 + v') \frac{\partial u_i'}{\partial y} + \] 
\[ (w_0 + w') \frac{\partial u_i'}{\partial z} = \] 
\[ -(u_0 \frac{\partial u_{0j}}{\partial x} + \frac{\partial u_{0j}}{\partial y} + \frac{\partial u_{0j}}{\partial z}) + \frac{1}{\rho_0 + p'} \frac{\partial p_0}{\partial x} \] (19)

\[ \frac{\partial u_i'}{\partial t} + L_3 + (v_0 + v') \frac{\partial u_i'}{\partial y} + (w_0 + w') \frac{\partial u_i'}{\partial z} = \] 
\[ -(u_0 \frac{\partial u_{0j}}{\partial x} + \frac{\partial u_{0j}}{\partial y} + \frac{\partial u_{0j}}{\partial z}) + \frac{1}{\rho_0 + p'} \frac{\partial p_0}{\partial y} \] (20)

\[ \frac{\partial u_i'}{\partial t} + L_4 + (v_0 + v') \frac{\partial u_i'}{\partial y} + (w_0 + w') \frac{\partial u_i'}{\partial z} = \] 
\[ -(u_0 \frac{\partial u_{0j}}{\partial x} + \frac{\partial u_{0j}}{\partial y} + \frac{\partial u_{0j}}{\partial z}) + \frac{1}{\rho_0 + p'} \frac{\partial p_0}{\partial z} \] (21)

where the \( L_i \)'s are the amplitudes of characteristic waves associated with each characteristic velocity \( \lambda_i \). These velocities are given by:

\[ \lambda_1 = (u_0 + u') - c \] (22)

\[ \lambda_2 = \lambda_3 = \lambda_4 = u_0 + u' \] (23)

\[ \lambda_5 = (u_0 + u') + c \] (24)

where \( c \) is the speed of sound:

\[ c^2 = \gamma \frac{\rho_0 + p'}{\rho_0 + p'} \] (25)

\( \lambda_1 \) and \( \lambda_5 \) are the speed of acoustic waves moving in the negative and positive \( x \) direction; \( \lambda_2 \) is the convection
velocity (the speed at which entropy waves will travel) while $\lambda_3$ and $\lambda_4$ are the velocities at which $v$ and $w$ are advected in the $x$ direction. The $L_i$'s are given by:

$$L_1 = \lambda_1 \left( \frac{\partial \rho'}{\partial x} - (\rho_0 + \rho')c \frac{\partial u'}{\partial x} \right)$$

$$L_2 = \lambda_2 \left( \frac{\partial \rho'}{\partial x} - \frac{\partial p}{\partial x} \right)$$

$$L_3 = \lambda_3 \frac{\partial u'}{\partial x}$$

$$L_4 = \lambda_4 \frac{\partial u'}{\partial x}$$

$$L_5 = \lambda_5 \left( \frac{\partial \rho'}{\partial x} + (\rho_0 + \rho')c \frac{\partial u'}{\partial x} \right)$$

We have shown here the characteristic boundary conditions in the $x$ direction. The equations for the $y$ and $z$ direction are derived in a similar manner. For edges and corners in three dimensional situations a simple extension is used by combining the equations for the various directions into one approximate equation.

This type of boundary condition treatment also 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.\(^{20}\) The time integration is a fourth order accurate Runge-Kutta method.

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)\(^{15}\) 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 8 Pentium II's networked together, the SP2 costs roughly 14 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).

Figure 3: Timings for a ship air wake case on several parallel computers

Figure 4: Timings for a ship air wake case on SP2
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 with several helicopter landing spots on the deck around the control tower. The airwake influences on the helicopter are quite different in these two cases. For frigates the flow separation area behind the hangar cube has a strong effect on a landing helicopter, while on LHA’s the deck leading edge vortex and separation are the key flow phenomena. Fig. 5 is some of the experimental data obtain by one of the authors (L. Long) with K. Long (PAX River) on the U.S.S Salpam (an LHA). It shows the flow velocity at different heights from the deck in the center plane. This was obtained from a cup anemometer and only includes the effects of velocity in the longitudinal and normal (to deck) directions. The wind was at 36 knots and had a 90 degree yaw angle. These data will eventually be used to validate numerical results for the LHA case.

![Figure 5: Field measurement of LHA](image)

In this paper some preliminary simulations have been done for a generic ship shape (TTCP ship). In fact, this is a generic frigate model and it is shown in Fig. 6 in a computational mesh. It is 240 feet long, 45 feet wide and 55 feet high. It was chosen because there will be 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 actual frigate since this study is concerned with the macroscopic 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 $201 \times 109 \times 85$ 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 and NLDE simulations were based on the same grid in order to avoid three dimensional interpolation. In fact, the NLDE needs much fewer grid points than CFL3D.

Mean flow simulation

NASA Langley and Ames research centers have devoted significant resources in the past decades to developing modern CFD technology. The CFL3D 5.0 package 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 separation: 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 number. 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 divided into 10 blocks.

The Mach number chosen for the simulation is a high wind case. The incoming flow speed is 41 knots. The water surface is assumed to be a hard wall boundary. Fig. 7 shows the contour plot of velocity magnitude on the surface of TTCP ship. This is a zero yaw angle case from CFL3D results. The asymmetry property of zero yaw angle flow is captured very well. The flow is accelerated around the sharp corners 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.

Figure 6: Configuration of TTCP ship and computational mesh

Figure 8: Flow speed contour on center plane of the TTCP ship

Fig. 9 shows velocity vectors in two horizontal planes 4.75 feet and 8.75 feet above the flight deck. Our numerical results are compared with a flow pattern obtained from an experimental study\(^\text{17}\) 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 deck floor is compared with flow visualization results for the TTCP ship.\(^\text{23}\) There are differences in the attachment point and the position of vortex center. This is probably due to our inviscid approach.

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 wild 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.

Finally in fig. 12 the performance of CFL3D (a serial code) is shown. For this TTCP ship configuration it takes days to get the steady state results.
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 freestream airflow are (1) the mean windspeed, 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 windspeed, elevation and roughness length scale.3

Incoming characteristics with source terms are introduced from the inflow boundary as it is expressed as \( L_i \) in equation (17) - (21). 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. 13 and fig. 14 give contour plots of longitudinal...
velocity perturbations $u'$ in the center plane at two different timesteps. Fig. 15 gives contour plots of the longitudinal velocity intensity $\sqrt{u'^2}$. From this unsteady result, 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. 16 and fig. 17, contour plots of vertical and transverse perturbation velocity $v'$, $w'$ are shown. In fig. 18 the vertical perturbation intensity $\sqrt{w'^2}$ 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 perturbation is quite weak over the flight deck. By comparing the instantaneous perturbations in fig. 13 and 17, the perturbation length scale of longitudinal perturbations is different from that of vertical perturbations.

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. 19 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 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. Our present results are qualitatively correct, and show that the key flow phenomena can be captured by using a steady-state code followed by the NLDE code. Future work will concentrate on more detailed comparisons to experiment, the inclusion of more geometrical features of the ships, and the inclusion 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: Contour of longitudinal velocity intensity

Figure 16: Contour of instantaneous transverse velocity perturbations


Figure 17: Contour of instantaneous vertical velocity perturbations

Figure 18: Contour of vertical velocity intensity


Figure 19: Contour plots of perturbation velocity intensity in the rotor plane of helicopter