COMPUTER SOFTWARE PRODUCT END ITEMS

CFD TOOLS

Submitted to:

SPAWAR Systems Center San Diego
53560 Hull Street, Code 2206
San Diego, California 92152-5001

Attention:
Mr. Mark Tukeman, Code 2712
(619) 553-1510

In fulfillment of the requirements for:

Contract No. N66001-02-D-0039
Development of Port Related Transportation Technologies
to Advance Military Responsiveness to National Needs

Task Order No. 0011
Computational Fluid Dynamic (CFD) Design Tool Development and Validation

CDRL Data Item No. A001BA

Security Classification: Unclassified

Prepared and Submitted by:

Center for the Commercial Deployment of Transportation Technologies
California State University, Long Beach Foundation
6300 State University Drive, Suite 220 • Long Beach, CA 90815 • 562.985.7394

Approved for public release: distribution is unlimited

September 30, 2003

Computer Software Product End Items

Computational Fluid Dynamics (CFD) Tool Development

Deliverable 1

Development of the Three-Dimensional Free Surface Code Based on Navier-Stokes Method

P/E Number: 2.20

Task Order Number: 11

POC

Hamid Hefazi, Professor & Chair

Mechanical and Aerospace Engineering Department

California State University, Long Beach

hefazi@csulb.edu

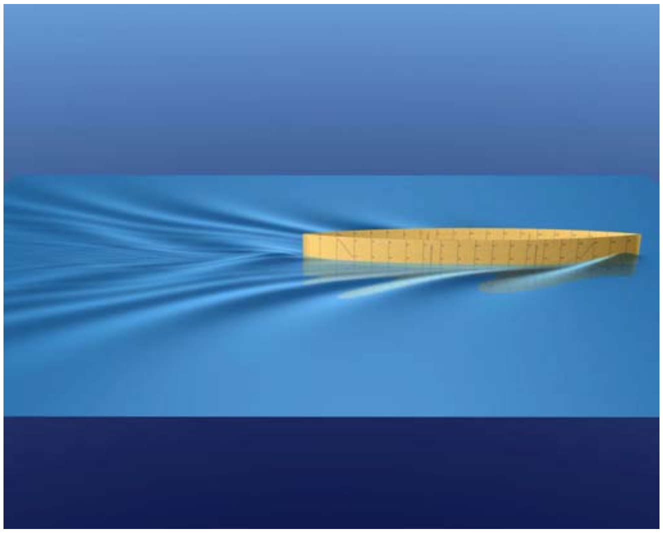

Table of Contents

1. Introduction

This report describes the theoretical development of a three dimensional free surface code based on the Navier Stokes method for hydrodynamic applications. It contains

a) The core information that is needed by a potential user to understand the theory and methodology that has been developed and implemented in this software to model the three dimensional non-linear free surface problem based on Navier Stokes method.

b) Provide information on the application of the method.

Following introduction of section 1, section 2 below describes the mathematical formulation that we have developed and implemented. Section 3 describes the grid system and algorithm that is used for the method. Section 4 briefly outlines the flow solver that is used as part of this algorithm. Finally, explanations of input/output data as well as the source code and validation of the method are addressed in the accompanying Software Test Report (STR).

Hydrofoils or ships moving at constant speed in a calm sea will generate a wave pattern. The waves created will dissipate energy and cause resistance to the moving body, which is referred to as “wave drag”. While the wave drag is small at relatively low speeds, it increases very quickly with speeds and becomes the dominant part of the total resistance. To move near a free surface in a fluid, hydrofoils or ships must overcome the wave drag. In order to accurately predict the wave resistance and minimize it by modifying the body shape in the design stage of hydrofoils and ships, it is necessary to develop capabilities to predict the wave pattern generated by the body.

Theoretical studies on the prediction of wave pattern generated by hydrofoils or ships have been carried out for a long time. However, until recently most studies are based on the linearized solutions of the free surface problem. In these linearized solutions, it is assumed that the disturbance to the incoming uniform flow due to the existence of the body and the free surface is small in magnitude compared with the uniform flow, and the free surface boundary conditions are imposed on the undisturbed water surface, not the real free surface. As the Froude number increases, the nonlinear effects of the free surface become more and more pronounced and the wave resistance predicted by these linearized solutions is not very reliable. Limitations of the linearized methods can be removed and accuracies of the solutions can be improved if the nonlinear effects are taken into account. Therefore, it is desirable to develop a numerical method to solve the fully nonlinear free surface problem.

The solution of the nonlinear free surface problem is much more difficult than the linearized solution due to the nonlinear boundary conditions and the presence of the moving boundary, i.e. the free surface. According to Froude's hypothesis, the resistance of hydrofoils and ships can be divided into a viscous part and an inviscid part, i.e., the wave drag. Therefore most traditional numerical methods make assumptions that there is only weak interaction between viscous and inviscid flows so they can be treated separately. In past years, panel method has been used widely to solve the inviscid nonlinear free surface flows. A drawback of treating viscous and inviscid flows separately is that interactions between viscous and free surface phenomena are not accounted for and accurate prediction of wave resistance cannot be provided. Another problem associated with this lies in the treatment of transom stern flows. In order to accurately predict the friction and form resistances, and the flow separation at the stern, new methods must be developed. With advances in computational fluid dynamics (CFD), attempts [1-4] have been made to simulate the nonlinear free surface flows by solving the Navier-Stokes equations so the viscous effect and its interaction with waves are taken into account in the analysis.

To solve the nonlinear free surface problem, an iterative procedure must be used. It has been found that the iterative procedure tends to be instable, and many researchers have experienced severe convergence problems in solving the nonlinear free surface problem. Therefore, techniques need to be incorporated into numerical schemes to suppress the instability.

In the present study, such a general, robust method using a Navier-Stokes solver has been developed to compute the free surface flows over submerged three-dimensional bodies. The method will be used to predict both the wave patterns and the frictional and wave drag components.

2. Mathematical Formulation

Consider a three-dimensional body moving with constant speed in a calm sea. A coordinate system moving with the body is adopted. The xy-plane is horizontal and the z-axis is positive upward. The governing equations are the incompressible Reynolds-averaged Navier-Stokes (RANS) equations. All length and velocities are non-dimensionalized by the freestream velocity, V_∞, and the reference length L, respectively. A pseudo-compressibility formulation was introduced by Chorin [5] to remove the difficulties associated with the prediction of incompressible flows due to the loss of an evolution equation for the density. Following Chorin's formulation, the time derivative of pressure is added to the continuity equation as follows:

$$\frac{\partial p}{\partial t}+\beta\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)=0 \tag{1}$$

where u, v and w are velocity components in x, y and z directions, respectively and β is a positive constant referred as the pseudo-compressibility parameter. Equation (1) and the momentum equations form a hyperbolic system of differential equations.

$$\frac{\partial}{\partial t}D+\frac{\partial}{\partial x}(E-E_v)+\frac{\partial}{\partial y}(F-F_v)+\frac{\partial}{\partial z}(G-G_v)=0 \tag{2}$$

where

$$D=\begin{bmatrix}p\\u\\v\\w\end{bmatrix}\quad E=\begin{bmatrix}\beta u\\u^2+p\\uv\\uw\end{bmatrix}\quad F=\begin{bmatrix}\beta v\\vu\\v^2+p\\vw\end{bmatrix}\quad G=\begin{bmatrix}\beta w\\wu\\wv\\w^2+p\end{bmatrix}$$

$$E_v = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \end{bmatrix} \qquad F_v = \begin{bmatrix} 0 \\ \tau_{yx} \\ \tau_{yy} \\ \tau_{yz} \end{bmatrix} \qquad G_v = \begin{bmatrix} 0 \\ \tau_{zx} \\ \tau_{zy} \\ \tau_{zz} \end{bmatrix} \tag{3}$$

where $\tau_{xx}, \tau_{xy}, \tau_{xz}, \tau_{yx}, \tau_{yy}, \tau_{yz}, \tau_{zx}, \tau_{zy}$, and τ_{zz} are the viscous stresses. Gravity force, which plays an important role in free surface flows, are taken into account by excluding the hydrostatic component from p in equations (1)-(3), i.e. $p = \hat{p} + z / F_r^2$ where $\hat{p}$ is the static pressure and F_r is the Froude number defined as $F_r \equiv \frac{V_\infty}{\sqrt{gL}}$.

The physical domain of the free surface problem is bounded by the inflow, outflow, body surface, free surface, plane of symmetry, and the far field boundaries. Figure 1 shows a cut of the physical domain in the xz-plane. Boundary conditions used in the present analysis are given as follows:

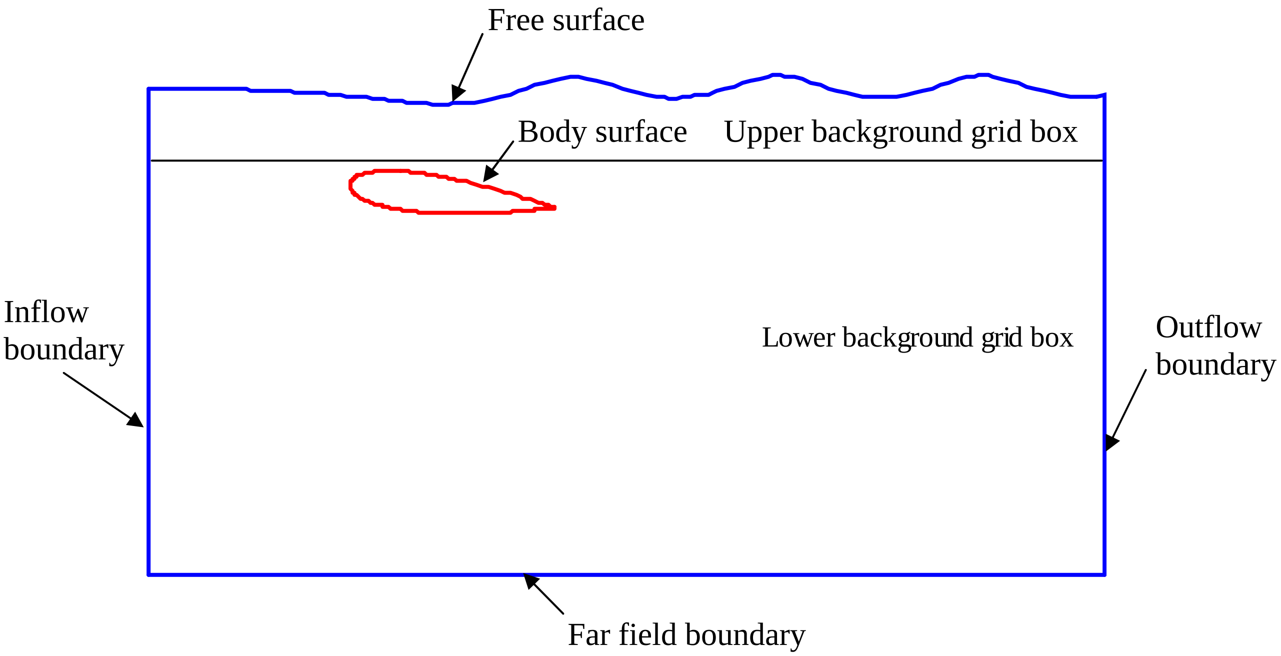

Figure 1. Cut of physical domain of free surface problem

1. Inflow boundary condition: $u = U_\infty, \quad v = 0, \quad w = 0, \quad p = p_\infty$ (4)
2. Outflow boundary condition: u, v, w and p extrapolated (zero-normal-gradient condition)
3. No-slip boundary condition on the body surface: $u = v = w = 0$ (5)

4. Free surface boundary condition:

Kinematic: $\frac{\partial h}{\partial t}+u\frac{\partial h}{\partial x}+v\frac{\partial h}{\partial y}-w=0$ (6)

Dynamic: $\hat{p} = p_0$ or $p = p_0 + h/F_r^2$ (7)

Where p_0 is the atmospheric pressure and $z = h(x, y, t)$ is the free surface coordinate.

5. Boundary condition on plane of symmetry: $v = 0, \quad \frac{\partial u}{\partial y} = 0, \quad \frac{\partial w}{\partial y} = 0, \quad \frac{\partial p}{\partial y} = 0$ (8)
6. Far field boundary condition: $u = U_\infty, \quad v = 0, \quad w = 0, \quad p = p_\infty$ (9)

The kinematic boundary condition given by equation (6) states that fluid can not penetrate through the free surface and the dynamic boundary condition given by equation (7) requires the static pressure to be constant and equal to the atmospheric pressure everywhere on the free surface. The inflow and outflow boundary conditions assume that flow is undisturbed far upstream, while a trailing wave pattern is propagated downstream. This implies that the waves do not propagated upstream and is referred to as the “radiation condition” of the free surface problem. The symmetry boundary condition is applied on the plane of symmetry where $y=0$. The far field boundary condition assumes infinite depth and breadth of the water so that no disturbances will be felt at the far field.

The free surface problem is difficult to solve since the location and shape of the free surface are unknown. Iterative method must be used in order to solve this problem with a moving boundary. The free surface location and shape must be updated in successive iterations until solution gets converged. Since the kinematic and dynamic boundary conditions, equations (6) and (7), cannot be satisfied simultaneously with specified location and shape of the free surface, only one condition can be imposed for each iteration. The other condition will be used to update the free surface location and shape between successive iterations.

Two different approaches can be adopted in the iteration procedure. The first approach is to impose the kinematic boundary condition on the specified free surface for each iteration to require that fluid cannot penetrate through the free surface. A new wave elevation is then calculated to satisfy the dynamic boundary condition, equation (7), for the next iteration. Although this iterative procedure is simple and straightforward, it is well known that it will generate a solution without a trailing wave pattern and the solution usually diverges.

The second approach is to impose the dynamic boundary condition on the specified free surface for each iteration and use the kinematic boundary condition, equation (6), to update the wave elevation for the next iteration. This approach is likely to have better convergence properties and has been used to solve the free surface flow over submerged bodies in previous studies [3, 6]. Therefore, this approach is adopted in our iterative procedure.

In his panel method, Dawson [7] found that it is possible to enforce the radiation condition indirectly by approximating the derivatives of velocities by upwind finite difference scheme. The use of upwind finite difference scheme adds a dissipative numerical mechanism to the free surface waves and helps the satisfaction of the radiation condition automatically. It is found that the stability of the iterative procedure is quite robust using this numerical technique. This approach was adopted in the present studies. With a central difference scheme for the time discretization, a three-point upwind finite difference scheme in x-direction and a central difference scheme in y-direction for the spatial discretization, equation (6) can be written as

$$\begin{aligned}&\frac{h_{i,j}^{n+1}-h_{i,j}^{n}}{\Delta t}+u_{i,j}^{n+1/2}(Ah_{i,j}^{n+1/2}+Bh_{i-1,j}^{n+1/2}+Ch_{i-2,j}^{n+1/2})+\\&\qquad v_{i,j}^{n+1/2}(Dh_{i,j-1}^{n+1/2}+Eh_{i,j}^{n+1/2}+Fh_{i,j+1}^{n+1/2})-w_{i,j}^{n+1/2}=0\end{aligned} \tag{10}$$

$$\begin{aligned}&A=\frac{2x_i-x_{i-1}-x_{i-2}}{(x_i-x_{i-1})(x_i-x_{i-2})},B=\frac{x_i-x_{i-2}}{(x_{i-1}-x_i)(x_{i-1}-x_{i-2})},C=\frac{x_i-x_{i-1}}{(x_{i-2}-x_i)(x_{i-2}-x_{i-1})}\\&D=\frac{y_j-y_{j+1}}{(y_{j-1}-y_j)(y_{j-1}-y_{j+1})},E=\frac{2y_j-y_{j-1}-y_{j+1}}{(y_j-y_{j-1})(y_j-y_{j+1})},F=\frac{y_j-y_{j-1}}{(y_{j+1}-y_{j-1})(y_{j+1}-y_j)}\end{aligned} \tag{11}$$

where Δt is the time step, i is the index in the streamwise direction, j is the index in the spanwise direction and n is the index for the time step. Starting from the inflow boundary and marching in the streamwise direction, equation (10) forms a linear, tridiagonal system which can be solved by the Thomas algorithm. Since the wave elevation is not known a priori, the free surface shape is assumed to be the undisturbed water surface initially. The flow is calculated at each time step with the dynamic boundary condition given by equation (7) imposed on the specified free surface. Before solution gets converged, the kinematic boundary condition is not satisfied and the flow is allowed to leak through the free surface. This leakage, in effect, drives the evolution of the free surface. The movement of the free surface is calculated and the wave elevation is updated using equation (10). The dynamic boundary condition is then imposed on the new free surface for the next time step. This procedure is repeated until solution gets converged and the flow reaches the final steady state. According to the kinematic boundary condition, the flow must be tangent to the free surface in the final steady state. Figure 2 shows the flow chart of this iterative procedure.

3. Grid system and algorithm

The overset grid system is adopted in our analysis to calculate three-dimensional flows over a submerged body. Three individual grids, a grid around the solid body and two background box grids, need to be generated (see Figure 1). A hyperbolic grid generation code, HYPGEN, was adopted to generate the volume grid. Both the upper and lower background grids are of H-H type. The primary purpose to split the background grid to two blocks is to reduce the grid size and save the computational time. The top surface of the upper background grid coincides with the free surface. It has fairly fine grid spacing in order to capture the movement of the free surface. The lower background grid covers the rest of the computational domain. Its grid spacing is relatively coarse comparing to that of the upper background grid.

Since the location and shape of the free surface change with iterations, regriding after each iteration is needed. The regriding is carried out using the grid perturbation technique to avoid lengthy computational time in grid generation for each iteration. The new wave elevation, $h_{i,j}^{n+1}$, is first calculated using equation (10), the movement of grid points is then calculated based on the following equation:

$$z_{i,j}^{n+1} = \begin{cases} z_{i,j}^{n} + \dfrac{h_{i,j}^{n+1} - z_0}{h_{i,j}^{n} - z_0}(z_{i,j}^{n} - z_0) & z_{i,j}^{n} > z_0 \\ z_{i,j}^{n} & z_{i,j}^{n} \le z_0 \end{cases} \qquad (12)$$

It is seen from equation (12) that only grid points lying above the plane $z = z_0$ are allowed to move vertically following the movement of the free surface. The movement is linearly distributed between the free surface and the plane $z = z_0$. The value of z_0 is chosen carefully such that none of the fringe points (where interpolation for the overset grid system takes place) will be moved during the regrinding process. This makes sure that redistribution and interpolation processes in the iterative procedure are avoided. Therefore, the computational efforts to re-run the Pegasus code for obtaining interpolation information for the overset grid system are eliminated.

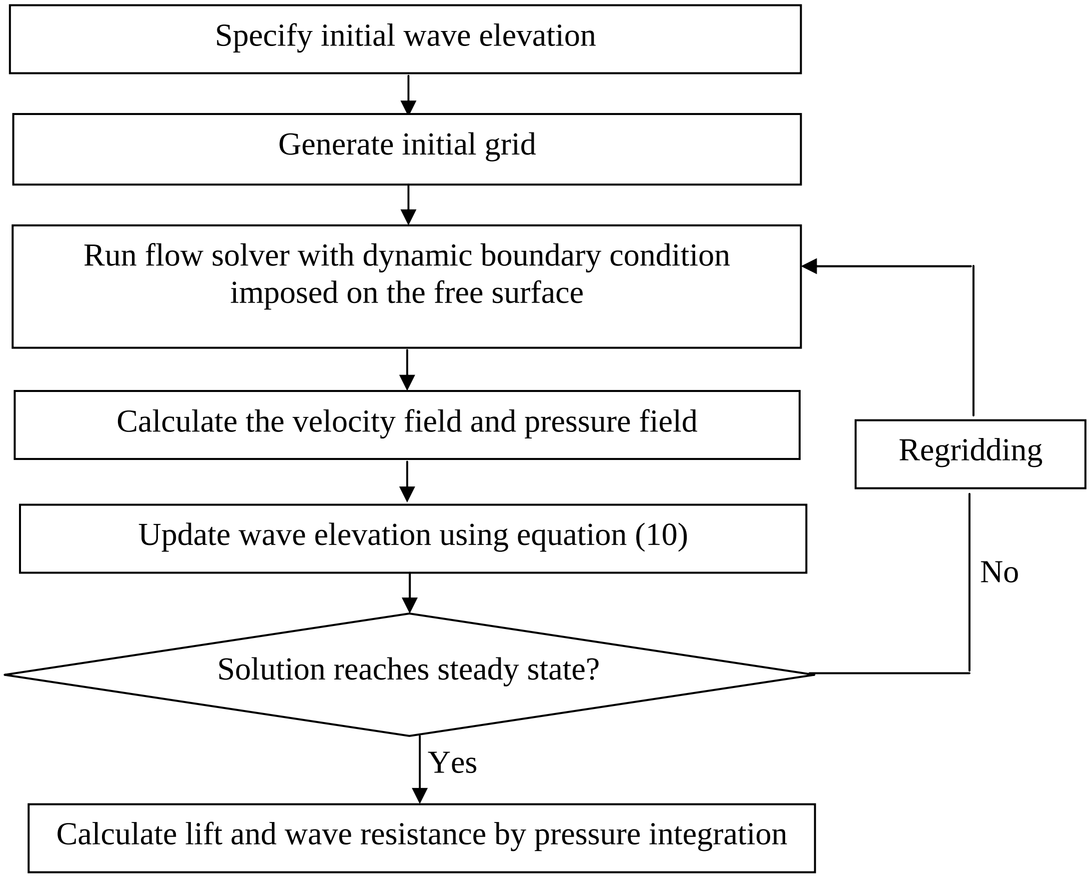

Figure 2. Flow chart of the iterative procedure

4. Flow solver

The existing three-dimensional incompressible Navier-Stokes code, INS3D [8, 9], was chosen as the flow solver to calculate the free surface flows. The INS3D code is a finite-difference, structured-grid flow solver developed by NASA for Aerodynamic applications. It is capable of solving both steady state and time-dependent problems. The convective terms are differentiated using an upwind biased flux-difference splitting. The equations are solved using an implicit line-relaxation scheme. The code is written for either single or multiple-zone calculations. The flow solver contains pre-coded boundary conditions for slip and no slip walls, symmetry planes, inflow and outflow boundaries, and far-field boundaries as described in the subsequent Software Test Report (STR). The new dynamic boundary condition on the free surface given by equation (1.2.7) has been implemented into the code as a user-provided subroutine. The source code for this boundary condition is also given in that accompanying STR.

Summary/Conclusion

In the present study, a general method has been developed to calculate three-dimensional flows over submerged bodies, by imposing a fully non-linear boundary condition on the free surface. The non-linear free surface condition is implemented through iterative procedures and has been incorporate into an existing incompressible Navier Stokes code (INS3D). Validation of this method as well as more detailed information on the algorithm is presented in the accompanying software test report (STR).

Glossary

Symbols

V_∞	Free Stream Velocity
P_∞	Free Stream Pressure
L	Reference Length
β	Pseudo-Compressibility Parameter
$\frac{\partial}{\partial x}$	Partial Derivatives
u, v, w	Velocity Components
τ	Viscous Stress

$F_r \equiv \frac{V_\infty}{\sqrt{qL}}$	Froude Number

Subscripts/Superscripts

∞	Free Stream
O	Reference Value
i, j, k	Discretization indeces x, y, z direction
n	Iterative index

Acronyms

CFD	Computational Fluid Dynamics
RANS	Reynolds Averaged Navier Stokes
HYPGEN	Hyperbolic Grid Generation
STR	Software Test Report

Acknowledgement:

The CCDoTT project is a collaborative effort involving several faculty and students at California State University, Long Beach. Staffs primarily responsible for this portion of the work are Drs. H.H Chen and H Hefazi Professors, Dr. Lie Mine Gea, consultant and Ryosuke Uda, gradate student in the Mechanical and Aerospace Engineering department.

References

1. T. Hino, “Computation of a Free Surface Flow Around an Advancing Ship by the Navier-Stokes Equations”, Proc. 5th International Conf. Numerical Ship Hydrodynamics, pp. 103,-117, Hiroshima, Japan, 1989.

2. H. C. Chen, V. C. Patel, and S. Ju, “Solutions of Reynolds-averaged Navier-Stokes Equations for Three-Dimensional Incompressible Flows”, J. Computational Physics, Vol. 88, pp. 305-336, 1990.

3. H. Miyata, M. Zhu, and O. Watanabe, “Numerical Study on a Viscous Flow With Free-Surface Waves About a Ship in Steady Straight Course by a Finite-Volume Method”, J. Ship Research, Vol. 36, No.4, pp. 332-345, 1992.

4. Y. Tahara, F. Stern, and B. Rosen, “An Interactive Approach for calculating Ship Boundary Layers and Wakes for Nonzero Froude Number”, J. Computational Physics, Vol. 98, 1992.

5. A. J. Chorin, “A Numerical Method for Solving Incompressible Viscous Flow Problems”, J. Computational Physics, Vol. 2, No. 1, pp. 12-26, 1967.

6. J. Farmer, L. Martinelli, and A. Jameson, “Fast Multigrid Method for Solving Incompressible Hydrodynamic Problems with Free Surfaces”, AIAA Journal, Vol. 32, No. 6, pp. 1175-1182, 1994.

7. C. W. Dawson, “A Practical Computer Method for Solving Ship-wave Problems”, Proc. Second Int. Conf. Numerical Ship Hydrodynamics, Berkeley, 1977.

8. S. E. Rogers, and D. Kwak, “An Upwind Differencing Scheme for the Steady-State Incompressible Navier-Stokes Equations”, J. Applied Numerical Mathematics, Vol. 8, pp. 43-64, 1991.

9. S. E. Rogers, and D. Kwak, “An Upwind Differencing Scheme for the Time-Accurate Incompressible Navier-Stokes Equations”, AIAA Journal, Vol. 28, No. 2, pp. 253-262, 1990.