Mecánica Computacional Vol. XXIV
A. Larreteguy (Editor)
Buenos Aires, Argentina, Noviembre 2005

FREE SURFACE FLOWS AROUND SHIPS: PROGRESS TOWARD SIMULATION OF HIGH-SPEED FLOWS AND MOTIONS

Pablo M. Carrica, Robert V. Wilson, and Fred Stern

IIHR-Hydroscience and Engineering
The University of Iowa
Iowa City, IA 52242
Pablo-carrica@uiowa.edu

Key words: Free surface flows, overset grids, large amplitude motions.

Abstract. *A steady and unsteady single-phase level set method is being developed for the RANS code CFDSHIP-IOWA to simulate free surface flows around complex geometry with large amplitude motions and maneuvering. A structured overset grid approach is used to allow flexibility in grid generation, local mesh refinement, and to efficiently resolve incident waves. The overset interpolation coefficients are computed with the software packages Pegasus (from NASA) or Suggar (from HPC PET). Dynamic interpolation is achieved through interfacing CFDSHIP-IOWA with Suggar, allowing for the simulation of moving objects and enabling the computation of moving hulls, rudders, etc. while maintaining good grid quality in the far-field.*

We show examples of high speed-flows exhibiting breaking bow and transom waves and a high-amplitude rolling surface combatant. High-speed ships are of increasing interest for diverse applications including fast commuter services, fast ferries, fast cargo ships, and fast combatants requiring innovative hull designs and propulsion systems. The method is demonstrated for two high-speed hulls showing overturning waves: the surface combatant DTMB 5512 and the R/V Athena patrol boat. Comparison of numerical results with measurements shows that the method is able to accurately simulate the overturning bow wave, resulting free-surface vortices, and boundary layer. Several splash-ups can be captured if the grid refinement is appropriate.

Dynamic overset grids are demonstrated for a coupled pitching and heaving bare hull surface combatant at medium and high Froude numbers in head waves.

1 INTRODUCTION

Ship hydrodynamics present many unique challenges due to complex geometry, environment, and operating conditions, which results in many complex physics and modeling issues. Operating conditions with high Froude (Fr) number, bluff geometry with appendages, and/or large amplitude ship motions and maneuvers in waves can lead to unsteady boundary layers and wakes with vortices and separations as well as steep, overturning, spilling, and breaking waves which can produce spray, foam, and bubbles. These affect the performance of the hull and propulsion system and increase air and water signatures.

Free surface modeling is a major issue for the computational solution of these flows. Complex free surface topologies (breaking waves, spray, bubble entrainment), unsteadiness caused by environmental waves and ship motions, and natural instabilities are complications that stress the free surface codes complicating the convergence to a solution. Additional complications arise from the need of enough grid points to resolve the boundary layer and the breaking waves. unsteadiness due to ship motions or incident waves, large differences in fluid properties between water and air, requirement for sufficient resolution of both the turbulent boundary layer and waves, and stability considerations. In addition, problems with complex free surface topologies can be modeled and resolved at a range of scales (e.g., modeling of the effect of the breaking wave on a smoothly resolved free surface or resolution of a spilling breaking wave and modeling of the two-phase flow).

Surface ships typically have complex geometries due to appendages (e.g., shafts, propellers, rudders, bilge keels, water jets) and use of novel hull designs to achieve stealth and/or speed requirements (e.g., tumblehome, trimaran). While unstructured grid approaches have been used to discretize complex ship geometries at low to medium Fr (e.g., Hino *et al*.[1], Lohner and Yang[2] and Burg *et al*.[3]), an alternate approach is the use of structured overset grids. Much of the work in this area was originally developed by the aerospace field and has been used to simulate flow around bodies with relative motion, re-entry vehicles, subsonic vehicles, rotorcraft vehicles, and turbomachinery[4,5,6]. With this method, complex configurations can be discretized using a combination of body-fitted structured grids embedded inside topologically simple background grids, which extend to the far-field. Also, overset grids have been extended for improved resolution of off-body flow features using automatic mesh refinement (AMR) and structured Cartesian refinement blocks[7]. Application of static overset grid techniques to the computation of free surface flows around ships was demonstrated by Orihara and Miyata[8], where the pitch and heave motions for two practical hull forms were predicted using a single-phase density function surface capturing approach. Computation of the fluid flow around the stern region of three practical ship models was also performed by Regnstrom *et al*.[9] using overset grids, although the free surface was not computed but replaced with a symmetry plane. Extension of overset methods to ship hydrodynamics for bodies with relative motion or AMR has not been demonstrated.

The authors of this paper and colleagues have contributed to the development of initially surface tracking and most recently surface capturing methods for ship hydrodynamics. Surface tracking methods were developed using both viscous/inviscid interaction and large-domain RANS approaches[10]. The latter approach was vastly improved for higher-order upwind discretization, PISO method for pressure-velocity coupling, blended k-ε/k-ω two-equation turbulence model, structured multi-block overset grids systems, and high performance computing, as CFDSHIP-IOWA: general purpose high performance computing RANS code for ship hydrodynamics applications[11]. Most recently a surface capturing unsteady single-phase level set method was developed for CFDSHIP-IOWA[12].

The present paper describes implementation of a transient single-phase method with dynamic overset grids to compute ship motions. This method allows, in principle, the computation of different hierarchies of body motions, including a moving ship with moving control surface (rudder, etc.) while crossing another ship also self controlled. Dynamic computation of the interpolation coefficients is achieved using the overset assembler Suggar[13] Since the method is implicit, Suggar is invoked several times per time step until convergence for each time step is attained. This type of approach retains good quality of grid, mainly at the free surface, while the ship moves. The method is demonstrated with a static example, the breaking bow wave in the Athena research vessel and the surface combatant model DTMB 5415, and with a dynamic example, pitch and heave in head waves at two speeds of the surface combatant DTMB 5415.

2 MATHEMATICAL MODEL

The mathematical and numerical formulations for the single-phase level set method were presented elsewhere[14,15] reader is referred to these papers for details. Only a summary of the method is provided here. Application of the single-phase method requires that the following two conditions must be satisfied: (i) the entire liquid-gas interface must be exposed to the ambient pressure since the continuity equation is not enforced in the air phase, (i.e., the method cannot handle bubbles inside the liquid), and (ii) the stresses on the liquid caused by the gas phase must be negligible since the motion in the air phase is not computed. The simplifications allow computation of the flow in the water region only with constant properties and without pressure and velocity oscillations associated with many two-phase approaches. These assumptions should be valid as long as the liquid-gas interface remains a free-boundary, as is the case for a large range of flows around free surface piercing bodies, which is the focus of the present paper. The non-dimensional Reynolds Averaged Navier-Stokes equations for the water phase can be written as:

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\frac{1}{\mathrm{Re}_{eff}}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + s_i \tag{2}$$

The Equations are non-dimensionalized with a characteristic velocity U_0 (free-stream velocity) and a characteristic length L (ship length). The dimensionless piezometric pressure is defined as $p = p_{abs} / \rho U_0^2 + z / Fr^2 + 2k/3$ with p_{abs} the absolute pressure, $Fr = U_o / \sqrt{g\,L}$

the Froude number, and the effective Reynolds number as $1/\mathrm{Re}_{eff} = 1/\mathrm{Re} + \nu_t$. The Reynolds stresses are related to the mean rate of strain through an isotropic eddy viscosity ν_t, which is calculated using Menter's blended *k-ω/k-ε* model without the optional shear-stress transport[16] and without the use of wall functions. The turbulent kinetic energy *k* and the turbulence specific dissipation rate ω are computed from transport equations.

The free surface is computed as the zero level set of the distance function ϕ, obtained by solving the following transport equation

$$\frac{\partial \phi}{\partial t} + u_j \frac{\partial \phi}{\partial x_j} = 0 \tag{3}$$

with appropriate boundary conditions for velocity and pressure at the free surface[17]. A reinitialization procedure is applied every time the free surface changes during the solution process. Since the objects in the computation can be moving, the non-slip condition is replaced by:

$$\vec{U}_S = \frac{\partial \vec{x}_S}{\partial t} \tag{4}$$

Where $\vec{U}_S$ is the no-slip surface velocity and $\vec{x}_S$ is the computed location of the wall point as a function of time.

3 MOTIONS

For details on the implementation of the motions in CFDShip-Iowa, the reader is referred to the work by Wilson et al.[18]. An inertial coordinate system, which moves with the constant forward speed of the ship, is used for the computation of the ship motions. The general 6DOF motion of a rigid body in an inertial reference frame is governed by conservation of linear and angular momentum

$$F_{CG_i} = m \frac{d^2 x_{CG_i}}{dt^2} \tag{5}$$

$$L_{CG_i} = \frac{d}{dt}\left(I_{ij} \frac{d\theta_{CG_j}}{dt} \right) \tag{6}$$

where x_{CG_i} and θ_{CG_i} are the Cartesian translations and rotations of the body, respectively, while F_{CG_i} and L_{CG_i} are the Cartesian resultant forces and moments acting at the center of gravity (CG) of the body along the i^{th} Cartesian coordinate direction. Modes *i*=1-3 represent the translational motions of the CG for surge, sway, and heave in Eq. (5), while *i*=4-6 represent the rotational motions of roll, pitch, and yaw in Eq. (6).

Prediction of motions is accomplished by numerically integrating the equations of motion for a rigid body. This done with a predictor-corrector approach, as shown in Fig. 1. In the predictor step, the translations and rotations are obtained explicitly from a first-order Euler forward difference computation. For example, for roll rotation we have:

$$\begin{aligned}\dot{\phi}^n &= \dot{\phi}^{n-1} + \Delta t \cdot \ddot{\phi}^{n-1} \\ \phi^n &= \phi^{n-1} + \Delta t \cdot \dot{\phi}^{n-1}\end{aligned} \tag{7}$$

where $\dot{\phi}^n$ and $\ddot{\phi}^n = \sum L_R / I_x$ are the angular velocity and acceleration of the ship and Δt is the computational time step. During the corrector step, Simpson's rule is used to integrate the rigid body equations using a third-order approach

$$\begin{aligned}\dot{\phi}^n &= \dot{\phi}^{n-1} + \Delta t \cdot \left(5\ddot{\phi}^n + 8\ddot{\phi}^{n-1} - \ddot{\phi}^{n-2}\right) / 12 \\ \phi^n &= \phi^{n-1} + \Delta t \cdot \left(5\dot{\phi}^n + 8\dot{\phi}^{n-1} - \dot{\phi}^{n-2}\right) / 12\end{aligned} \tag{8}$$

In Eqn. (8), computation of the ship angular velocity and position at time level *n* requires knowledge of the resultant moment L_R at time level *n*, i.e., the equation is implicit and must be solved iteratively as shown in Fig. 1. Arbitrary prescribed motions are possible in all translation and rotation directions that are not predicted.

4 NUMERICAL DETAILS

The general-purpose RANS solver, CFDSHIP-IOWA, has been developed at the University of Iowa for ship hydrodynamics and high-performance computing platforms. A structured, body-fitted, non-orthogonal multiblock approach is used with overset grids. The overall scheme is parallelized using MPI. The continuous governing equations are transformed from the physical domain in Cartesian (x,y,z,t) coordinates into the computational domain in non-orthogonal curvilinear coordinates (ξ, η, ζ, τ). A partial transformation is used in which only the independent variables are transformed, leaving the velocity components in the base coordinates.

A second-order, three-level Euler backward difference is used for discretization of the time derivatives in the level set, turbulence, and momentum equations. All convective terms are discretized using a second-order upwind method. The viscous terms are computed using a second-order central difference scheme. The Pressure Implicit Split Operator (PISO) algorithm[19] is used to couple the momentum and continuity equations by taking the divergence of the momentum equation, using the divergence free condition, and updating the momentum and pressure equations using a predictor-corrector approach.

The CFDSHIP-IOWA flow solver utilizes dynamic overset grids, giving the code the capability to simulate free surface flow over complex geometry, to perform local mesh refinement and to compute moving bodies with no need of remeshing. Hole cutting and connectivity information between overlapping blocks is created whenever required by interfacing dynamically with Suggar[13], developed at PET to serve US DoD HPC users. Once the overset information is generated, CFDShip-Iowa splits the information for parallel computing in a domain decomposition approach.

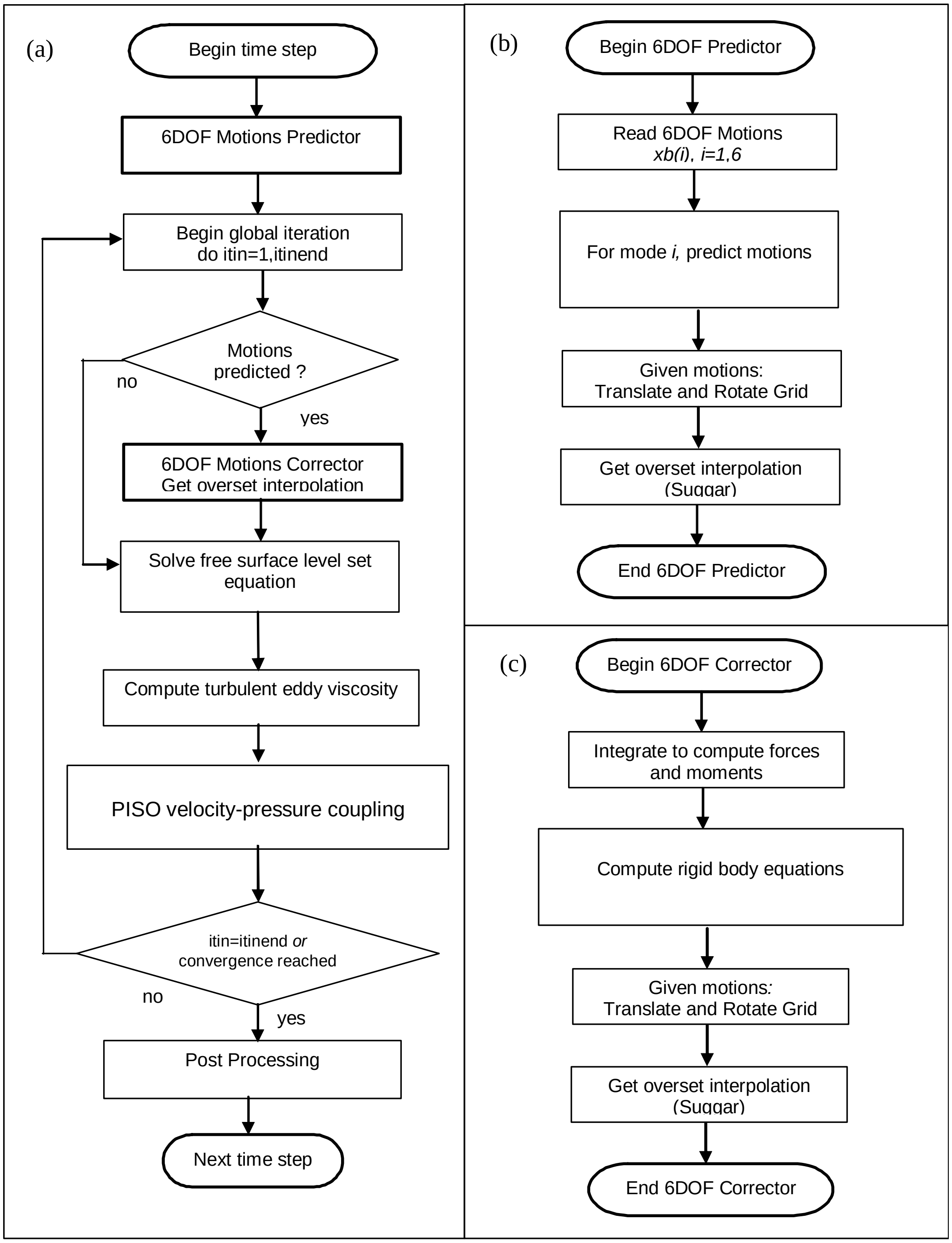

Figure 1: Motions solution procedure

All simulations are performed in a time accurate manner to resolve naturally unsteady flow features, should they exist. A domain decomposition MPI-based approach is used where each decomposed block is mapped to one processor of a parallel machine. Nonlinear and coupled algebraic equations for the turbulence, level set, momentum, and pressure are solved in a sequential manner and a global iteration loop is used to couple and converge the solutions at the unknown time level *n*, thus assuring a time-accurate simulation. Each time step is considered to be globally converged when each of the L2 norms based on the change of the velocity components and pressure between successive global iterations drops below $5x10^{-4}$. On average, 2-5 global iterations are required for convergence of the flow field equations. At the beginning of the global iteration, the turbulence equations are solved followed by the transport and reinitialization of the level set function. Next, the velocity-pressure coupling is enforced by the PISO algorithm, which requires the implicit solution of the momentum equations, followed by the implicit solution of the pressure equation and the explicit update of the velocity. If the problem involves moving grids, the overset coefficients are computed at the beginning of the global iteration.

The algebraic equations arising from discretization of the turbulence, level set, and momentum equations are solved using a line-ADI scheme with penta-diagonal solvers and a local iteration loop is used to converge multi-block and overset boundary points, which are lagged from the previous local iteration. The local iteration loops for the turbulence, level set, and momentum equations are run until the L2 norm based on the change in successive iterates drops below $1x10^{-5}$, $1x10^{-6}$, and $1x10^{-5}$, respectively, requiring on average, 25, 10, and 5, local iterations to reach convergence. For the Poisson equation, a matrix is built and solved using a biconjugate-gradient-squared algorithm available in the Portable Extensible Toolkit for Scientific Computation (PETSc) toolkit[20] and equations for multi-block and overset boundary points are built into the matrix so that a local iteration loop is not required. Convergence of the pressure equation is reached when the normalized residual imbalance of Poisson equation drops below $5x10^{-4}$ (i.e., the residual drops roughly four orders of magnitude from its initial value). Tests show that the solution does not change when the convergence criteria for the local and global iteration loops is reduced by a factor of two.

5 BREAKING BOW WAVES

Breaking bow waves are presented for two different ships, DTMB 5415 and Athena R/V. The DTMB 5415 is a slightly modified model scale version of the US Navy surface combatant DDG-51. At Fr=0.35, breaking waves have been measured by Olivieri *et al.*[21]. For this speed the Reynolds number is $Re=1.57x10^{7}$. An overset refinement block has been used to resolve the bow breaking wave, resulting in a grid with $2.43x10^{6}$ grid points ran in 32 processors. The grid is shown in Fig. 2. Figure 3 shows a comparison of the predicted and measured wave elevations for this case. The agreement with the experimental data is strikingly good. The experimentally observed “scars” are captured by the CFD. These scars are lines observed experimentally where the free surface is deflected due to *x*-vorticity generated by breaking waves. Predicted and measured resistance coefficients are 0.00479 and 0.00483, respectively.

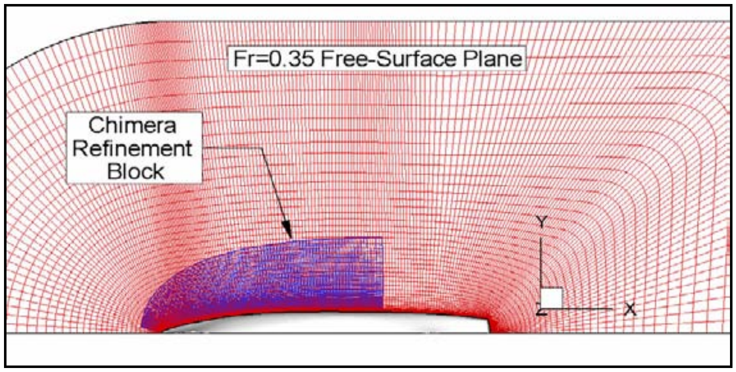

Figure 2: View of the computational grid for DTMB 5415 at *Fr*=0.35

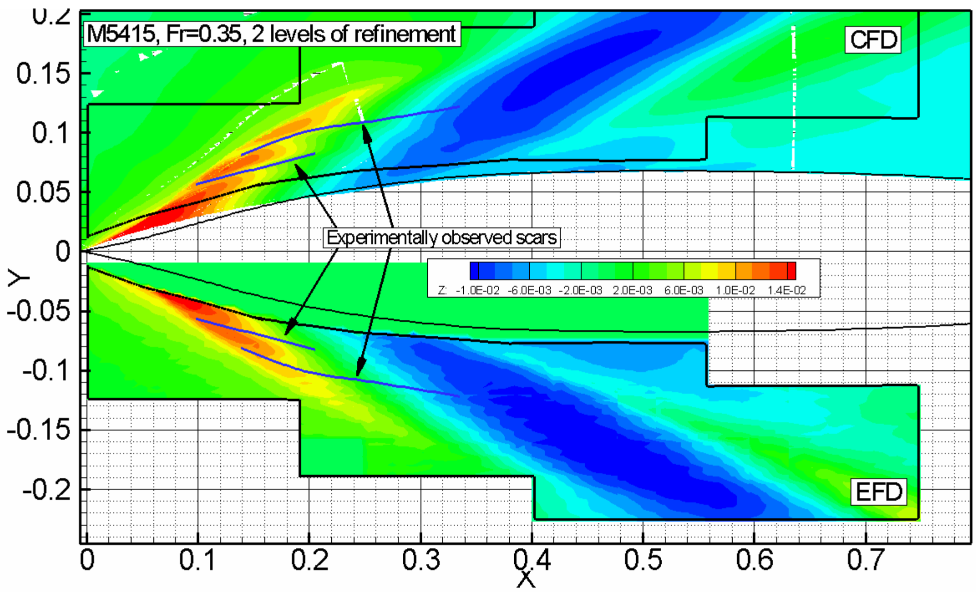

Figure 3: free surface elevation comparison with experimental data for DTMB 5415 at *Fr*=0.35

The Athena research vessel DTMB 5365 is a 8.25 geosym of a 47 *m* long multipurpose boat of the US Navy. Extensive experiments have been performed at David Taylor Model Basin. This geometry was selected as a test case for the 4 April 2005 ONR Ship Wavebreaking Workshop to identify the capability of RANS codes to predict bow and transom wave breaking for high speed ships and workshop results were submitted in a blind fashion. We present here the results for *Fr*=0.43. Two different levels of discretization have been used to evaluate the effect of grid refinement on the resolution of the free surface, as shown in Fig. 4.

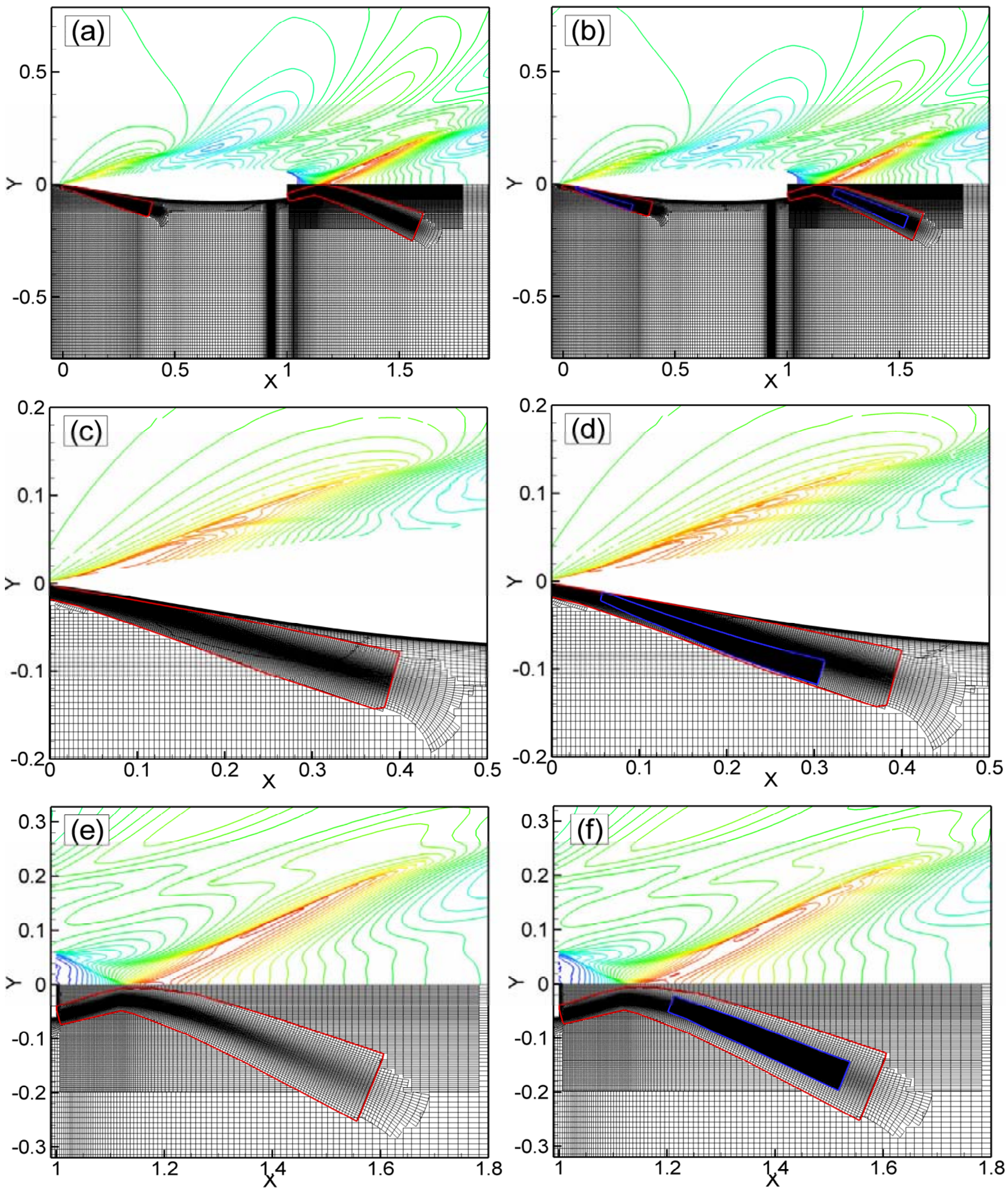

Figure 4: Wave elevation contours and grid using one level (a),(c),(e) and two levels (b),(d),(f) of overset grid refinement: (a),(b) overall, (c),(d) bow, and (e),(f) stern regions (Athena R/V).

The effect on the free surface is more clearly shown in Fig. 5, where cross section at different x/L are shown for both refinement levels.

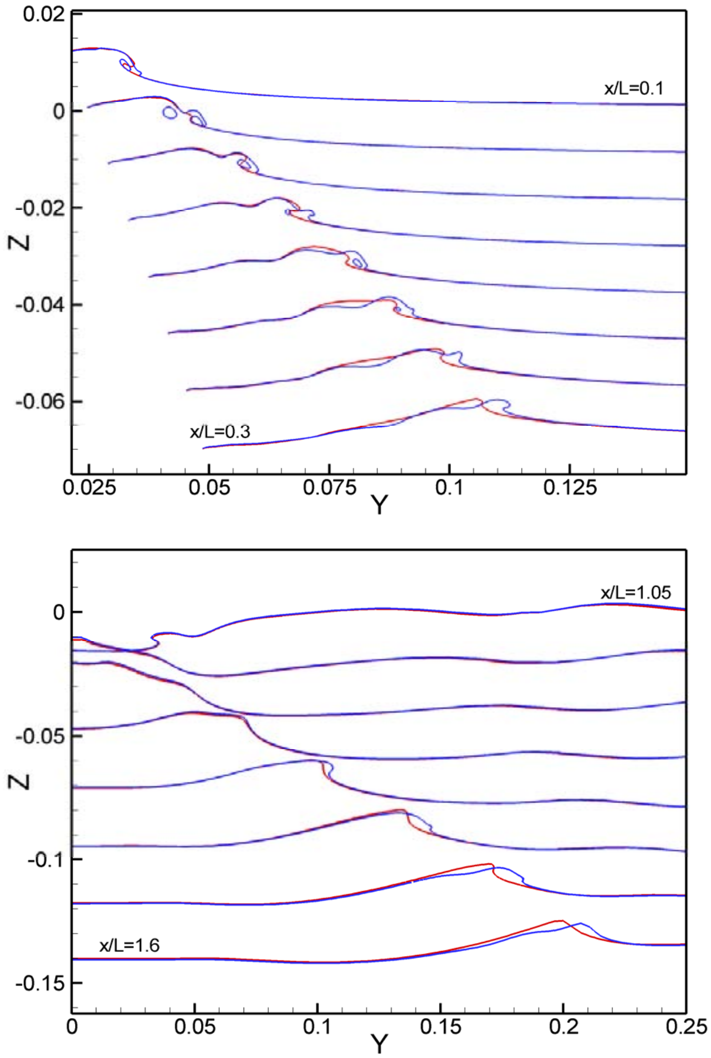

Figure 5: Effect of local grid refinement on breaking bow (top) and transom (bottom) wave systems: free surface predictions at eight cross sections using one (red) and two (blue) levels of grid refinement.

A comparison of the CFD free surface prediction with the experimental measurements of Fu *et al.*[22] are shown in Fig. 6. Again, the agreement is excellent. The experimental data grid was not focused on detecting the breaking waves, so much of the detail is lost in the experimental data due to a too coarse resolution. However the elevation, location of wave fronts and angles on the Kelvin and breaking patterns are clear and agree well with the CFD.

Figure 7 shows a view of the overall free surface compued with CFDShip-Iowa.

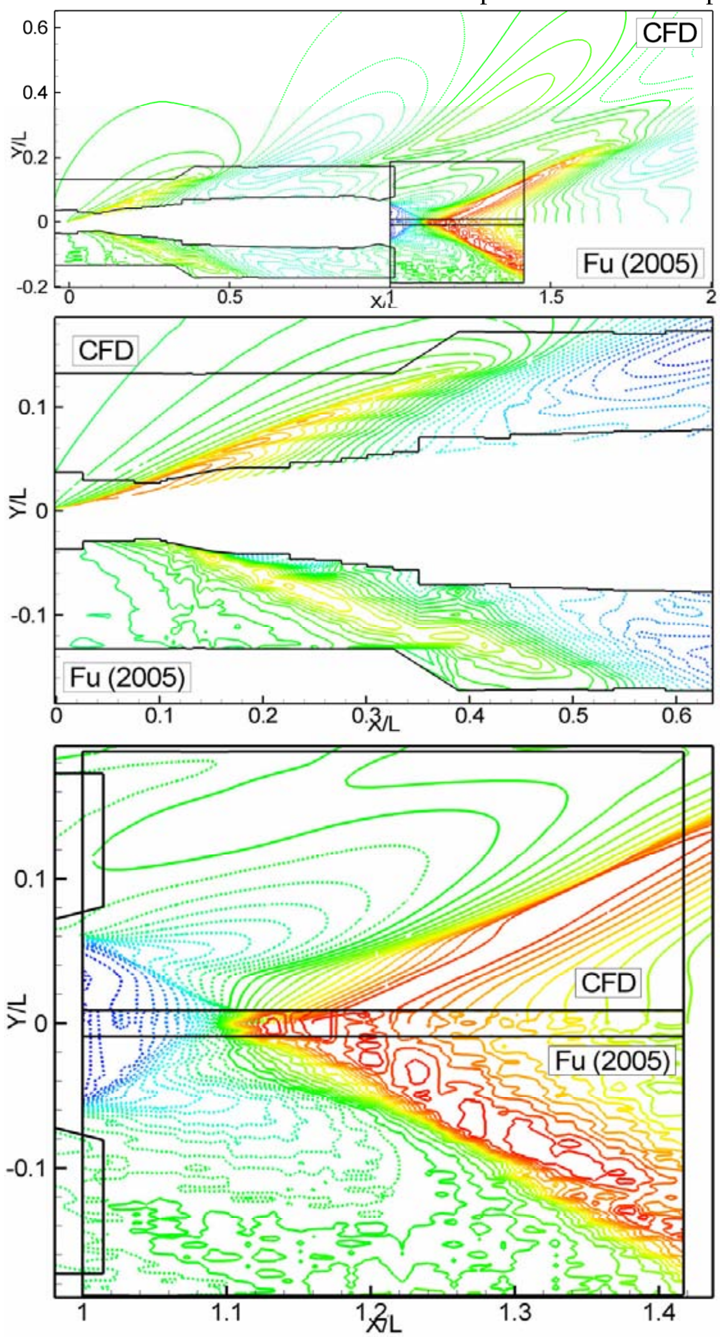

Figure 6: Comparison of predicted and measured wave elevation contours: global (top), bow (middle) and transom (bottom), for Athena R/V

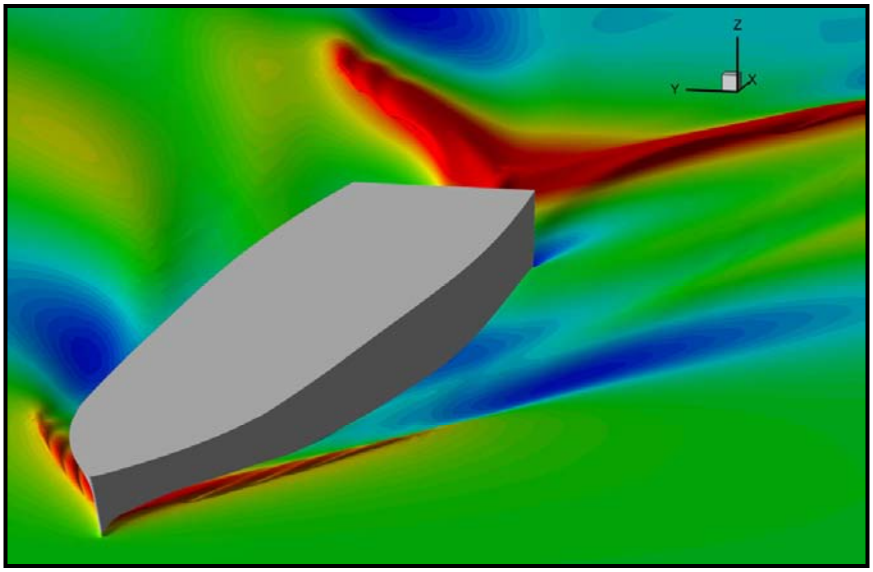

Figure 7: Predicted free surface for high speed case: 3D perspective.

6 PITCH AND HEAVE IN HEAD WAVES

Pitch and heave computations were carried out for DTMB 5512 at medium (*Fr*=0.28) and high (*Fr*=0.41) speeds. The model 5512 is similar but smaller than 5415. The wavelength was λ=1.5 *L*, and the wave slope *ak*=0.025. This case has been experimentally studied by Irvine *et al*.[23] The grid consisted of three overset blocks. The boundary layer block was allowed to move, while the refinement across the free surface and the background block were stationary.

Figures 8 and 9 show time histories for pitch and heave as compared to the experimental measurements, for Froude numbers *Fr*=0.28 and *Fr*=0.41, respectively. Since the numerical model is started with impulsive acceleration to full speed, there are significant low frequency transients that are slowly dying. Otherwise, the phase, amplitude and zero-level agree well with the experimental data.

7 CONCLUSIONS

A dynamic overset free surface solver has been presented. Overset grids have been used to refine regions of interest or to compute body motions. Three examples have been shown: the breaking bow wave in the DTMB 5415 model, the breaking bow and transom waves in the Athena R/V, and predicted pitch and heave in the model DTMB 5512. The solver has proved robust and accurate for this type of calculations.

Future developments include full 6-DOF implementation, arbitrary heading waves, automatic grid refinement and propeller modeling. A significant number of test cases are being pursued now, including fully appended ships, rolling ships and extreme motions.

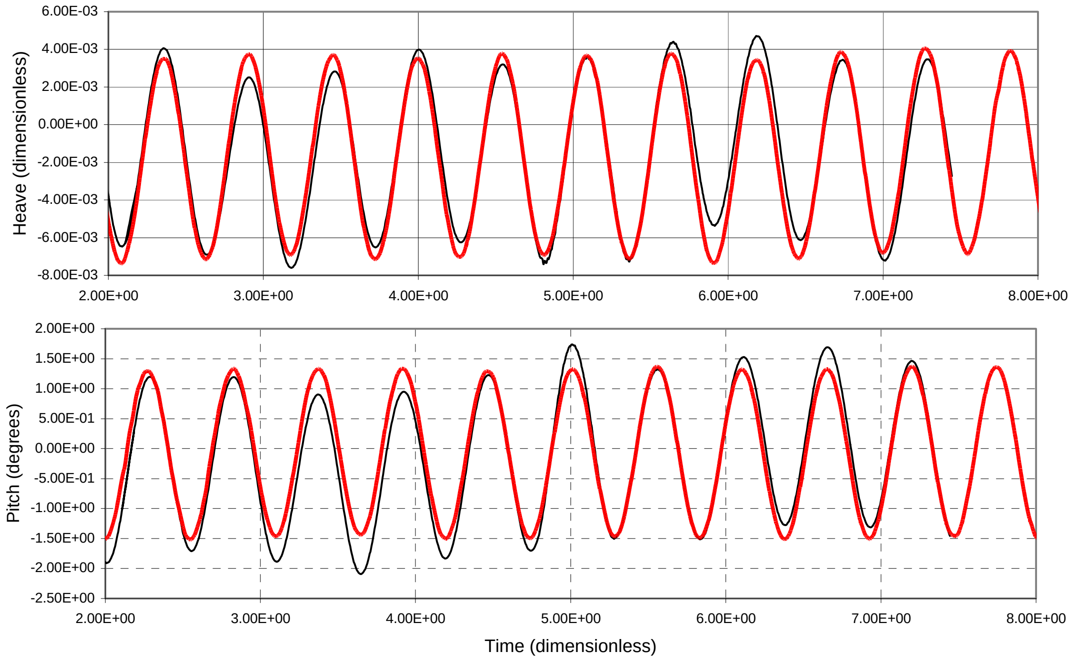

Figure 8: Predicted (blach) and measured (red) pitch and heave for DTMB 5512 in head waves at medium speed.

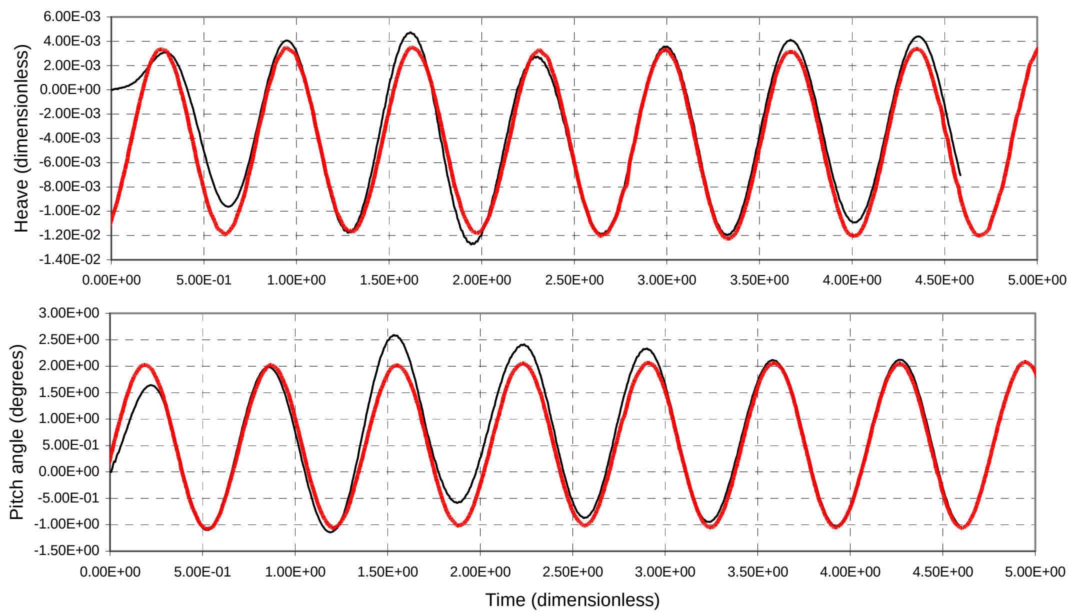

Figure 9: Predicted (blach) and measured (red) pitch and heave for DTMB 5512 in head waves at high speed.

8 ACKNOWLEDGEMENTS

This work has been supported by the Office of Naval Research under Grant N00014-01-1-0073. Dr. Patrick Purtell was the program director.

9 REFERENCES

[1] T. Hino, K. Ohashi and Y. Ukon, "Flow Computations Around a Ship with Appendages by an Unstructured Grid Based NS Solver", 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, S. Korea, (2003)

[2] R. Lohner and C. Yang, "Viscous Free Surface Hydrodynamics Using Unstructured Grids", 22nd ONR Symp. on Naval Hydrodynamics, Washington DC, USA (1998)

[3] C.O.E. Burg, K. Sreenivas, D. Hyams and B. Mitchell, "Unstructured Nonlinear Free Surface Solutions: Verification and Validation", 24th ONR Symp. Naval Hydrodynamics, Fukuoka, Japan (2002)

[4] R. Meakin, "Moving Body Overset Grid Methods for Complete Aircraft Tiltrotor Simulations", AIAA-93-3350-CP, 11th AIAA Computational Fluid Dynamics Conf., Orlando, FL (1993)

[5] W.M. Chan, R.J. Gomez,, S.E. Rogers and P.G. Buning,, "Best Practices in Overset Grid Generation," AIAA Paper 2002-3191, 32nd AIAA Fluid Dynamics Conf., St. Louis, MO (2002)

[6] N.E. Suhs, S.E. Rogers and W.E. Dietz, "PEGASUS 5: An Automated Pre-processor for Overset-Grid CFD," AIAA Paper 2002-3186, 32nd AIAA Fluid Dynamics Conf., St. Louis, MO (2002)

[7] R. Meakin, "An Efficient Means of Adaptive Refinement within Systems of Overset Grids", AIAA paper 95-1722 (1995)

[8] H. Orihara and H. Myata, "A Numerical Method for Arbitrary Ship Motions in Arbitrary Wave Conditions using Overlapping Grid Systems," 8th Int. Conf. Numerical Ship Hydrodynamics, Busan, S. Korea (2003)

[9] B. Regnstrom, L. Broberg, and L. Larsson, "Ship Stern Flow Calculations on Overlapping Composite Grids", 23rd ONR Symp. Naval Hydrodynamics, Val de Reuil, France, (2000)

[10] Y. Tahara and F. Stern, "Large-Domain Approach for Calculating Ship Boundary Layers and Wakes and Wave Fields for Nonzero Froude Number", *J. Comp. Phys.* **127**, 398-411 (1996)

[11] E.G. Paterson, R.V. Wilson and F. Stern, "General-Purpose Parallel Unsteady RANS Ship Hydrodynamics Code: CFDShip-Iowa", IIHR report 432, Iowa Institute of Hydraulic Research, The University of Iowa (2003)

[12] P.M. Carrica, R.V. Wilson and F. Stern, "An Unsteady Single-Phase Level Set Method for Viscous Free Surface Flows", submitted to *Int. J. Num. Meth. Fluids* (2004).

[13] R. Noack, "SUGGAR: a General Capability for Moving Body Overset Grid Assembly", 17th AIAA Computational Fluid Dynamics Conf., Toronto, Ontario, Canada (2005)

[14] R.V. Wilson, P.M. Carrica, M. Hyman and F. Stern, 2004, "An Unsteady Single-Phase Level-Set Method with Application to Breaking Waves and Forward Speed Diffraction

Problem", 25th ONR Symposium on Naval Hydrodynamics, St Johns, Canada, (2004)
[15] P.M. Carrica, R.V. Wilson and F. Stern, "Unsteady RANS Simulation of the Ship Forward Speed Diffraction Problem," IIHR-Hydroscience and Engineering Report No.444, April (2005)
[16] F.R. Menter, "Two-Equation Eddy Viscosity Turbulence Models for Engineering Applications", *AIAA J.* **32**, 1598-1605 (1994)
[17] P.M. Carrica, R.V. Wilson and F. Stern, "Unsteady RANS Simulation for Forward Speed Diffraction of a Surface Combatant," *Comp. Fluids*, in press (2005)
[18] R. V. Wilson, P. M. Carrica and F. Stern, "Six-Degree-of-Freedom Unsteady RANS Method with Application to Roll Motion for a Surface Combatant", *Comp. Fluids*, in press (2005).
[19] R.I. Issa, "Solution of the Implicitly Discretized Fluid Flow Equations by Operator Splitting", *J. Comp. Phys*. 62, 40-65 (1985)
[20] S. Balay, K. Buschelman, W. Gropp, D. Kaushik, M. Knepley, L. Curfman, B. Smith and H. Zhang, "PETSc User Manual", ANL-95/11-Revision 2.1.5, Argonne National Laboratory (2002).
[21] A. Olivieri, F. Pistani, E. Campana, R. Benedetti, F. LaGala, R.V. Wilson and F. Stern, "Froude Number and Scale Effects and Froude Number 0.35 Wave Elevations and Mean-Velocity Measurements for Bow and Shoulder Wave Breaking of Surface Combatant DTMB 5415," IIHR–Hydroscience and Engineering Report No.441, The University of Iowa, November (2004)
[22] T. Fu, A. Karion, A. Pence, J. Rice, D. Walker and T. Ratcliffe, "Characterization of the Steady Wave Field of a High Speed Transom Stern Ship - Model 5365 Hull Form", Naval Surface Warfare Center, Carderock Division, Hydromechanics Directorate R&D Report, NSWCCD-50-TR-2005 (2005)
[23] M. Irvine, J. Longo and F. Stern, "Towing-Tank Test for Surface Combatant for Free Roll Decay and Coupled Pitch and Heave Motions", 25th ONR Symp. on Naval Hydrodynamics, St. John, Newfoundland, Canada (2004)