Progress towards RANS simulation of free-surface flow around modern ships

By I. Senocak AND G. Iaccarino

1. Motivation and objectives

There is a significant effort in the marine industry to integrate computational fluid dynamics (CFD) simulation capability in designing energy efficient ship hull forms while lowering the noise generated from them. Accurate simulation of turbulent free surface flows around surface ships has a central role in the optimal design of such naval vessels. The flow problem to be simulated is rich in complexity and poses many modeling challenges because of the existence of breaking waves around the ship hull, and because of the interaction of the two-phase flow with the turbulent boundary layer.

Generally speaking, there are two distinct approaches in representing the free-surface in a Reynolds-averaged Navier-Stokes (RANS) simulation, namely, interface tracking and interface capturing methods. Both approaches aim to compute the wave profile accurately because the wetted surface area appears in the calculation of the drag acting on ship motion. In interface tracking methods, a kinematic boundary condition is applied at the free-surface, and the governing equations are solved only for the water phase (Li *et al.* 2000; Rhee & Stern 2001). Since the computational grid has to conform to the free-surface shape, this approach is not efficient for high Froude number flows in which breaking waves with high amplitudes are typical. On the other hand, the surface capturing methods are more versatile in handling a variety of free-surface flow conditions, in which the governing equations are solved for both the air and the water phases. Volume of fluid (VOF) (Hirt & Nichols 1981), level-set (Osher & Sethian 1988) and front tracking (Harlow & Welch 1965) are the widely adopted techniques in this category.

The turbulent flow structure in the vicinity of the free-surface can also be much more complex than the turbulent flow structure of single phase flows. For instance, it is known that for turbulent breaking waves, surface tension effects are important at smaller scales, and gravity is effective on larger scales (Brocchini & Peregrine 2001). Existing turbulence models, which have been mainly proposed for single phase flows, may not adequately represent the turbulence structure at the free-surface. The existence of a free-surface interacting with the turbulent boundary layer presents challenges for turbulence modeling. It is therefore necessary to assess carefully the performance of available turbulence models in literature for simulating turbulence flows with free-surface.

The present study is an initial effort towards RANS simulations of free surface flows around modern ship hulls with advanced turbulence models and interface capturing schemes. In this paper, our goal is to demonstrate the feasibility of such a simulation and to identify the issues regarding computational stability such as grid quality and resolution, and turbulence modeling.

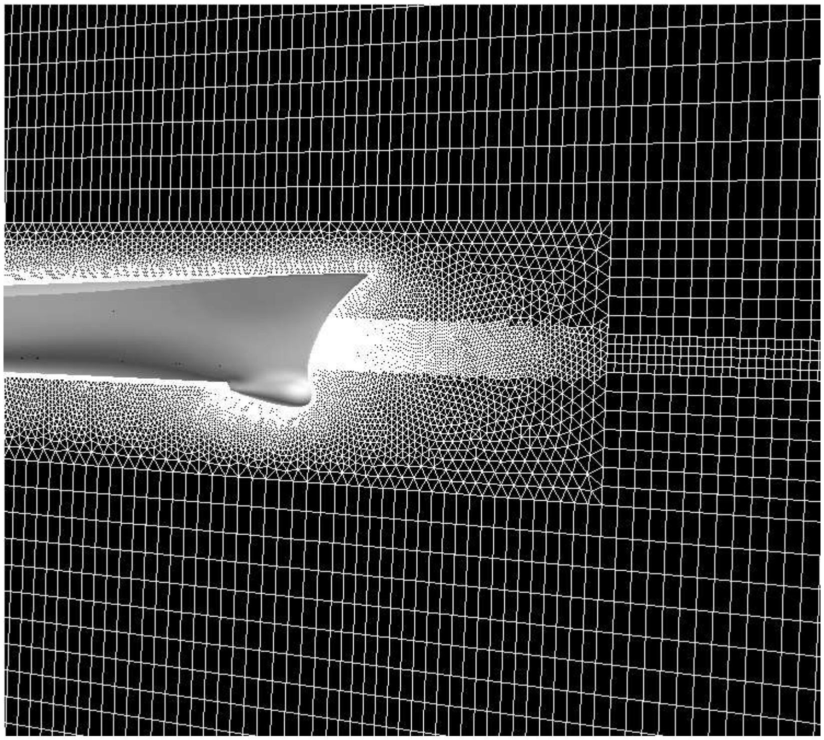

FIGURE 1. The overall view of the unstructured hybrid mesh around the model ship DTMB 5415.

2. Numerical techniques

The commercial CFD software FLUENT version 6.2 is utilized for the computations in the present study. FLUENT solves the RANS equations with a finite-volume approach on hybrid unstructured grids. A variety of pressure-based algorithms are available in FLUENT. For the present steady-state computations, the SIMPLE algorithm (Patankar & Spalding 1972) is adopted, and the volume of fluid technique is employed to simulate the free-surface motion. The second order upwind scheme is used for discretizing the convection terms in the momentum transport equations. A special high resolution interface capturing scheme is used for the convection term of the volume of fluid transport equation in order to avoid excessive smearing of the interface due to numerical diffusion (Muzaferija & Peric 1998). The standard k-ϵ model with wall functions is used for turbulence modeling (Launder & Spalding 1974). The resulting system of equations is solved using an algebraic multigrid method for faster convergence.

The motion of the free-surface flow is governed by gravitational and inertial forces. Hence, the boundary conditions to be imposed must take into account the gravity effects. For this purpose, the computational domain is modeled as an open channel flow, which is also consistent with the experimental setup.

3. Results

In the present study, we simulate a model surface ship (DTMB 5415) advancing in calm water under steady conditions. DTMB 5415 is a model of a modern US Navy

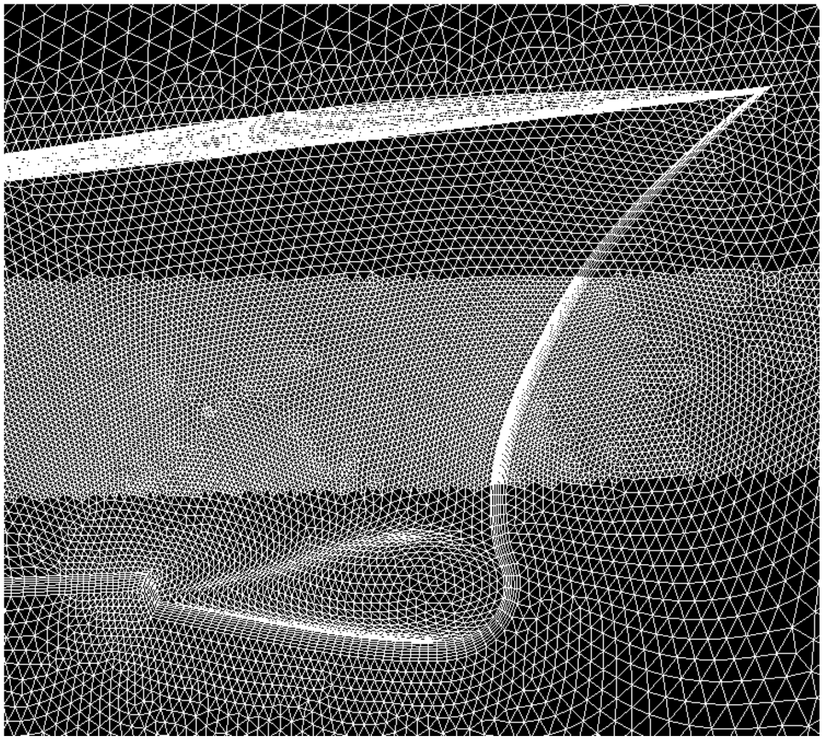

FIGURE 2. Close-up view of the mesh displaying the resolution at the boundary layer and around the free-surface.

combatant surface ship and has been selected as a benchmark case in the Gothenburg 2000 workshop on numerical ship hydrodynamics (Larsson *et al.* 2000). This specific model is also recommended by the 1996 International Towing Tank Conference as a benchmark case for CFD computations of ship resistance and propulsion. Olivieri *et al.* (2001) provide towing tank experiments on resistance, sinkage, trim, wave profile and the wake field under various conditions.

The important dimensionless numbers defining the flow conditions for the free-surface flow around ships are the Froude and the Reynolds numbers, respectively. The Froude number is representative of the ratio of inertial forces to gravitational forces, whereas the Reynolds number represents the ratio of inertial forces to the viscous forces. A Froude number of 0.28 and a Reynolds number of 1.2×10^7 defines the flow conditions for the present case with the model ship in fixed orientation. The length scale is the distance measured between the perpendiculars of the model ship ($l = 5.72m$).

An important issue in free-surface flows is the grid quality. A fine grid around the interface is desirable to minimize the smearing of the interface due to numerical diffusion. In addition, the present VOF technique implemented in FLUENT also benefits from adopting grid cells of hexahedral shape. Furthermore, the turbulent boundary layer, which is very thin due to the high Reynolds number, needs to be resolved with a mesh having a not too large aspect ratio. Although these requirements can be met easily for a relatively simple geometry, such as a free-surface piercing hydrofoil, the grid generation process for the current geometry proved to be a tedious one because of the existence of a bulbous sonar dome, which can be seen in Fig. 1. A hybrid unstructured mesh consisting

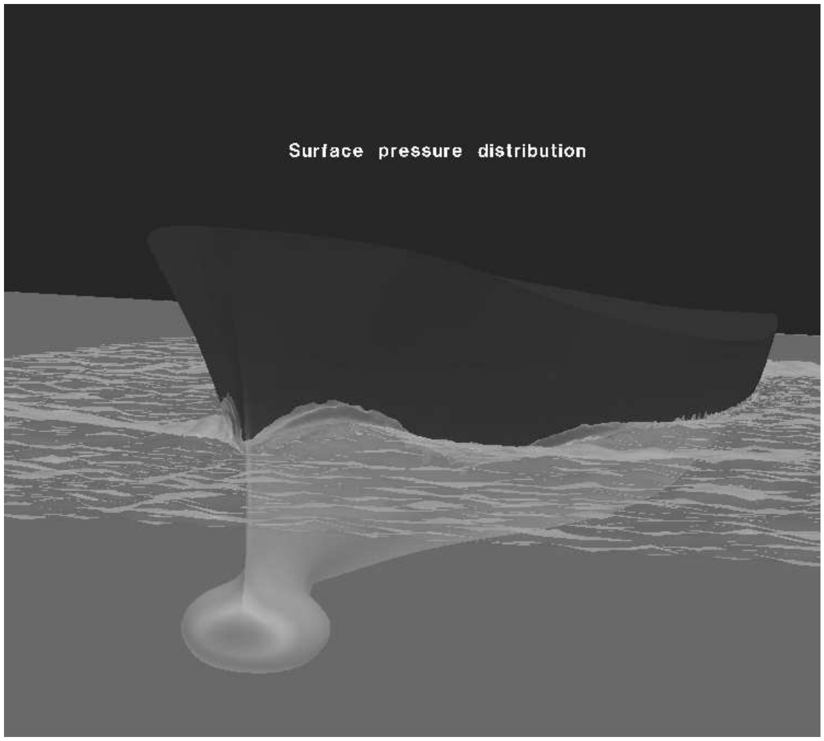

FIGURE 3. The pressure distribution on the ship hull, and free-surface deformation around it.

of about 2 million cells was generated for the present simulations. Figure1 shows the overall grid structure adopted in the simulation, and Fig. 2 presents a close up view of the grid near the boundary. The grid generation process is as follows. First, the surface of the ship is triangulated, and then a boundary layer mesh is fitted all around it. The spacing of the nearest grid cells are such that it is consistent with the wall function formulation. The computational domain is then meshed with tetrahedral shaped cells in the core, which is surrounded by hexahedral cells in the rest of the domain. Finally, a local grid refinement (*e.g.* each cell is divided into 2 cells) is applied around the free-surface to provide reasonable resolution of the interface.

The pressure distribution on the model ship is shown in Fig. 3. As can be seen from the figure, the stagnation pressure has developed realistically on the sonar dome. The free-surface deformation is also visible in this figure. Although we do not present any direct comparisons with experiments in this study, the free-surface deformation is in qualitative agreement with the experiments (Olivieri *et al.* 2001). We also note that the wrinkles on the free-surface are due to post processing of the data obtained from the unstructured tetrahedral cells.

Figure 4 shows the wave pattern generated due to the advancement of the model ship in the calm water. The forward velocity is projected onto the free-surface to display the wave pattern. Two distinct scars on the free surface, originating from the side of the ship, are visible in this figure. There is also a distinct scar in the wake field, which originates from the back of the ship. These dominant features on the wave pattern are in qualitative agreement with the experiments.

FIGURE 4. Wave pattern on the free surface. The projection of forward velocity onto the free surface is shown.

4. Future plans

We have presented a preliminary RANS simulation of free-surface flow around a model ship (DTMB 5415). The commercial CFD software FLUENT version 6.2 is utilized for the computations. The VOF technique is used for capturing the free-surface, and the standard k-ϵ model is adopted for the turbulence closure. The mesh quality has proven to be an important issue in the computations.

Future work should focus on direct comparisons with the experiments while employing a finer resolution grid than the one employed in the present study. The standard k-ϵ has well known deficiencies for complex flows. Therefore, different turbulence models should be tested in the simulations. In that respect, the $v^2 - f$ of Durbin (1995) is a promising choice.

There are various schemes for the interface reconstruction within the VOF technique. These schemes are designed to minimize the smearing of the interface and can be very useful for the present case. However, most of these schemes are for time-dependent problems. Hence, unsteady simulations should be performed in order to utilize them. Furthermore, unsteady computations are also necessary for simulating a surface ship advancing towards a regular head wave, which represents a more realistic condition.

REFERENCES

BROCCHINI, M. & PEREGRINE, D. H. 2001 The dynamics of strong turbulence at free surfaces Part 1: Description. *J. Fluid Mech.* **449**, 225.

DURBIN, P. A. 1995 Separated flow computations with the k-ϵ-v^2 model. *AIAA J.* **33**, 659.

HARLOW, F. H. & WELCH, E. J. 1965 Numerical calculation of time-dependent viscous incompressible flows of fluid with free surface. *Phys. Fluids* **8**, 2182.

HIRT, C. W. & NICHOLS, B. D. 1981 Volume of fluid method for the dynamics of free boundaries. *J. Comp. Phys.* **39**, 201.

LARSSON, L., STERN, F. & VERTRAM, V. 2000 A workshop on numerical ship hydrodynamics. *Proceedings of Gothenburg 2000*, Chalmers University of Technology, Gothenburg, Sweden.

Launder, B. E. & Spalding, D. B. 1974 The numerical computation of turbulent flows. *Comp. Methods App. Mech. Eng.* **3**, 269.

Li, T., Matusiak, J. & Lehtimaki, R. 2000 Numerical simulation of viscous flows with free surface around realistic hull forms with transom. *Int. J. Num. Meth. Fluids.* **37**, 601.

Muzaferija, S. & Peric, M. 1998 A two fluid Navier-Stokes solver to simulate water entry. In *Proceedings of 22nd Symposium on Naval Hydrodynamics*, Washington DC.

Olivieri, A., Pistani, F., Avanzini, A., Stern, F. & Penna, R. 2001 Towing tank experiments of resistance, sinkage and trim, boundary layer, and free surface flow around a naval combatant INSEAN 2340 model. *IIHR Technical Report No. 421*, The University of Iowa.

Osher, S. & Sethian, J. 1988 Fronts propagating with curvature dependent speed: Algorithms based on Hamilton Jacobi formulations. *J. Comp. Phys.* **79**, 12.

Patankar, S. V. & Spalding, D. B. 1972 A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *Int. J. Heat Mass Trans.* **15**, 1787.

Rhee, S. H. & Stern, F. 2001 Unsteady RANS method for surface ship boundary layer and wake and wake field. *Int. J. Num. Meth. Fluids.* **37**, 445.