VISCOUS FREE SURFACE FLOW PAST A SHIP IN STEADY DRIFT MOTION

A. Di Mascio, R. Penna, M. Landrini, E.F. Campana

INSEAN - Italian Ship Model Basin, Via di Vallerano 139, 00128 Roma, Italy

Abstract

The Reynolds averaged Navier–Stokes (RANS) equations with non linear free surface boundary conditions have been solved to simulate the flow field past a ship hull advancing with a drift angle. A Finite Volume technique has been used to discretize the equations, leading to a nonlinear algebraic system solved by a standard multigrid algorithm. Preliminary numerical results obtained for the Series 60 hull have been compared with experimental data.

Introduction

The computation of the unsteady incompressible viscous flow past a ship in maneuvering remains a great challenge. In fact, the general problem is a formidable one. The nonsymmetric flow over the hull is fully three-dimensional, boundary layers are turbulent, flow separation is common and may be followed by reattachment, and large wakes and complicated wave pattern are formed.

In principle, a fully viscous computation allows the prediction of the generation and transport of vorticity in the boundary layer and in the wake, and the coupled free surface and boundary layer interaction. However, the numerical solution of the general problem is in practice still strongly constrained by computer resourches. Reliable simulations of the flow past a manouvering ship are at present only feasible for steady drift motion.

Nevertheless, numerical computations of the RANS equations for the steady problem are particularly important also for the development of improved simplified models. Indeed they may provide useful detailed information on the location of the separation lines and on the evolution of the wake and may be used to calibrate and validate inviscid rotational models. As an example, the influence of an approximate choice of the location of the separation line on the values of the hydrodynamical lateral force and the yaw moment is still to be investigated. In fact, inviscid rotational models give satisfactory results once the separation line is known, as for a flat plate [1] or for a Wigley model. Unfortunately, this can be easily done only when the geometry is such to force the separation (i.e. sharp edges). Furthermore, to define the separation line in the case of 3D ship flows may not be an easy task.

A previous computation of the viscous free surface flow around a yawed Wigley model was attempted by using a domain decomposition approach [2]. In the present paper a large domain solution has been developed.

Mathematical model

We consider the steady flow past a ship hull $\mathcal{B}$ moving in an incompressible viscous fluid. The flow domain is bounded by the free surface $\mathcal{S}$, by the hull surface and extends to infinity. We assume a body-fixed reference frame with the x-axis aligned with the uniform flow and the z-axis positive upwards. The variables have been nondimensionalized by the ship length L and the free stream velocity U.

The velocity field is divergence free

$$\nabla \cdot \mathbf{u} = 0 \qquad \text{in } \mathcal{D} \tag{1}$$

and the momentum equation has to be satisfied

$$\mathbf{u} \cdot \nabla \mathbf{u} + \nabla P = \nabla \cdot \tau \qquad \text{in } \mathcal{D} \tag{2}$$

In the previous equation P is the 'total' pressure, i.e. the sum of the pressure term and the gravity term

$$\nabla P = \frac{\nabla p}{\rho} + \frac{\mathbf{k}}{Fr^2} = \nabla \left[\frac{p}{\rho} + \frac{z}{Fr^2} \right] \tag{3}$$

k being the unit vector aligned with the z-axis. τ is the stress tensor, including the turbulent stresses

$$\tau = (\frac{1}{Re} + \nu_T)\,[(\nabla \mathbf{u}) + (\nabla \mathbf{u})^T] \tag{4}$$

ν_T being the kinematic eddy viscosity. In the present work the Baldwin–Lomax turbulence model has been used.

The boundary conditions to be imposed are the standard ones for Navier-Stokes computations. At the solid wall no slip conditions are enforced, i.e. velocity is set to zero at the boundary (no conditions are required for the pressure). On the free surface $\mathcal{H}(x, y)$, neglecting the effects of surface tension and viscosity, the following kinematic and dynamic (constant pressure on $\mathcal{S}$) boundary conditions are to be satisfied:

$$u\frac{\partial \mathcal{H}}{\partial x} + v\frac{\partial \mathcal{H}}{\partial y} = w \tag{5}$$

$$P = \frac{\mathcal{H}}{Fr^2} \tag{6}$$

Numerical Solution

The Series 60 model (C_b=0.6) has been selected for simulation of this type of flow. In fact, for this model, detailed experimental data are available [3]. Furthermore, we have performed also some comparisons with measured data obtained at INSEAN.

Since the study is devoted to the simulation of the rectilinear motion of a ship advancing with a drift angle, in the numerical solution we cannot exploit the symmetry of the problem about the (x, z)-plane and therefore port and starboard sides are discretized. Hence the computational domain is decomposed into a port and a starboard block, the topology of each block being of H-O type. RANS equations have been written in a pseudo–transient formulation and a Finite Volume technique has been used to discretize the problem. Time integration has been carried out by a Runge–Kutta algorithm, second order accurate in time. The convergence has been accelerated by a FMG–FAS (Full Multigrid–Full Approximation Storage) multigrid technique.

As first test case, we simulate the flow past a Series 60 advancing in a oblique course for a drift angle $\alpha = 5°$ and for $Fr = 0.316$, $Re = 1.5 \times 10^7$. In this computation we have used $128 \times 64 \times 32$ cells in each block (port, starboard) of the fluid domain (streamwise, normal, and girthwise directions respectively).

The wave profile along the hull for both port and starboard side is shown in fig. 1 in comparison with some experimental data obtained by Longo [3]. The wave profile at the bow is dramatically modified with respect to the case $\alpha = 0$, since the different pressure values in that area, between pressure and suction side, imply respectively an increase and a lowering of the wave height. The numerical simulation was able to catch the main features of the flow. The agreement is satisfactory from $x = 0.2$ to $x = 1.2$ (the hull is located between 0 and 1). The flow at the bow ($x < 0.2$) is qualitatively predicted but the maximun free surface elevation in this region are underestimated. The unsufficient longitudinal grid resolution is this area is obviously a major factor in the loss of accuracy of the numerical prediction.

Forces and moments acting on the hull have been also predicted and compared with eperimental data obtained at INSEAN for the fixed hull case. Measurements were made on a model 6.096 m long, following the ASME guidelines [4] for the uncertainty analysis. As a preliminary check, numerical results for the force and moment coefficients for a drift angle $\alpha = 5°$ and for $Fr = 0.316$, $Re = 1.5 \times 10^7$, are compared with the experimental data in Table 1. The computed normal force coefficient C_x shows a satisfactory agreement with the measured data, while the moment coefficient C_{Mz} and the lateral force coefficient C_y are overpredicted.

In fig. 2 the history of the convergence for C_y has been reported as a funtion of the work, defined as the cost of one iteration on the finest grid. The best performance, from the point of view of the CPU time requirements, has been found with a five level computation. C_y values obtained on each level can be easily followed from the coarsest to the finest grid and compared with the reported measured data.

C_x		C_y		C_{Mz}	
Exp.	*Num.*	*Exp.*	*Num.*	*Exp.*	*Num.*
-0.0155	-0.0141	0.0222	0.0314	0.0106	0.0133

Table 1: Computed and measured forces and moment coefficients for $\alpha = 5°$, Fr = 0.316, Re = 1.5×10^7

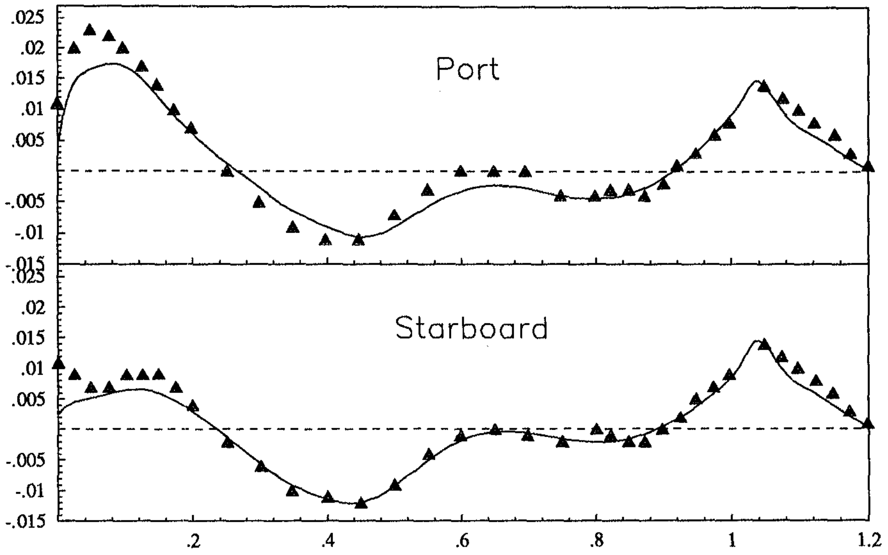

Figure 1: Wave profile along the hull: $Fr = 0.316$, $Re = 5 \times 10^6$, $\alpha = 5°$. Solid lines, numerical simultation ; $\triangle$, experimental data by [3]

A typical wave pattern is depicted in fig. 3 for $Fr = 0.316, Re = 1.5 \times 10^7, \alpha = 5°$ As espected, the free surface elevation in the starboard side is less pronunced, expecially near the bow. The wave pattern is stretched in the port side and spread in the starboard side.

Finally, for the same case as before, the visualization of the computed wake shed from the keel and the stern lines is reported in fig. 4.

Acknowledgment

This research activity was supported by *Ministero dei Trasporti e della Navigazione* through INSEAN Research Program 1994-1996.

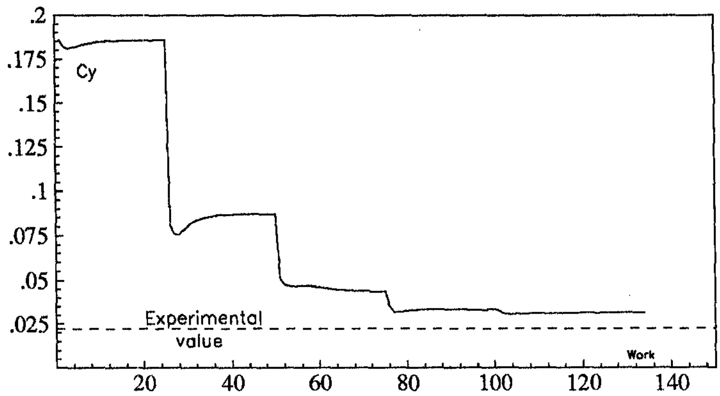

Figure 2: Convergence history for C_y as a function of the work. Five grid levels have been used in the multigrid algorithm. Correspondingly the value of C_y varies from the coarsest grid to the finest one. The experimental value is reported with the dashed line.

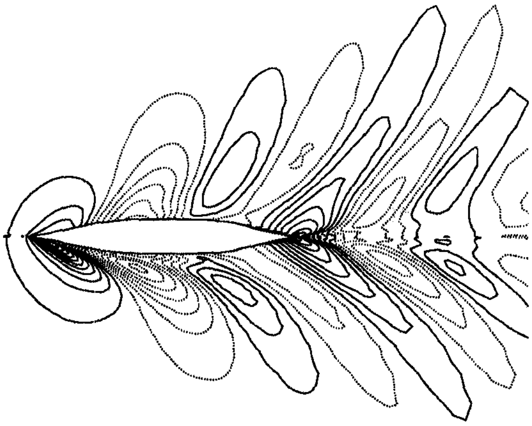

Figure 3: Computed wave pattern for $Fr = 0.316$, $Re = 1.5 \times 10^7$, $\alpha = 5°$

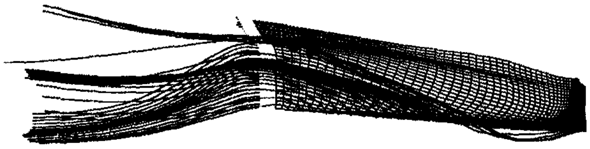

Figure 4: The wake shed from the keel and the stern lines of a Series 60 advancing with a drift angle $\alpha = 5°$

References

[1] Landrini M., Campana E.F., "Steady Waves and Forces About a Yawing Flat Plate", J. Ship Res., Vol. 40, No.3, pp.179–192 (1996).

[2] Campana E.F., Esposito P.G., Penna R., "Numerical Simulation of the Drift Motion of a Ship", Proc. of the 20-th O.N.R. Symposium on Naval Hydrodynamics, Santa Barbara, (1994).

[3] Longo J.,Ph.D. Thesis, Dept. of Mech. Eng., Univ. of Iowa (1996).

[4] ANSI/ASME PTC 19.1-1985, Part 1 "Measurement Uncertainty " (1986).