International Journal of Offshore and Polar Engineering (ISSN 1053-5381)

Vol. 17, No. 2, June 2007, pp. 89–96

Numerical Study of Confined Water Effects on Self-propelled Submarine in Steady Manoeuvres

Riccardo Broglia, Andrea Di Mascio and Roberto Muscari
The Italian Ship Model Basin, INSEAN, Rome, Italy

This paper deals with the analysis of confined water effects on the manoeuvring capabilities of a submarine. The analysis is carried out by using numerical simulations based on the Reynolds Averaged Navier Stokes equations of a submarine moving with constant speed on a straight path at zero and nonzero drift/pitch angles, in open water, close to the bottom, and close to the free-surface conditions. The features of the flow around the submarine are described in terms of velocity and pressure fields; the computed force and moment coefficients are presented and compared with experimental data collected at INSEAN.

INTRODUCTION

This work is part of a European research project supported by the WEAO, whose aims are:

- to improve numerical prediction methods for submarines manoeuvring in confined waters;
- to provide an experimental database of submarine manoeuvres in confined water; and,
- to validate the computational fluid dynamic prediction of forces and moments on a body close to a boundary.

To this aim, we considered straight-course, steady-pitch and steady-drift manoeuvres of a submarine operating in open waters, close to a solid flat bottom, and close to the free-surface conditions. In the latter case, otherwise calm water conditions are assumed. The submarine is appended with aft planes, 2 vertical rudders and forward horizontal planes; the operating propeller is accounted for by means of a suitable actuator disk model. The analysis is carried out by numerical simulations with a general purpose 2nd-order finite volume solver for incompressible turbulent free-surface flows; the algorithm was developed at INSEAN (Di Mascio et al., 2001, 2006). The theoretical assumptions and numerical features of the INSEAN solver will be briefly described below.

When analyzing the results, the local flow characteristics are considered first; particularly investigated are the velocity and pressure fields, the streamline patterns and the boundary layer thickness. Also analyzed are the influence of the body parameters (arrangement and configuration) and of the solid and deformable boundaries on the local flow. Then reported and discussed are the global loads (forces and moments) acting on the submarine; the effects of the body parameters (the configuration) and the water conditions (open or confined waters) on the loads are also analyzed. Presented for one selected case are the investigation of the properties of the provided numerical solutions, in terms of convergence and uncertainties of the numerical data, as well as the comparison with experimental data collected at INSEAN. At paper's end, conclusions are drawn.

MATHEMATICAL AND NUMERICAL MODELS

The mathematical model employed in the simulations is described by the Reynolds Averaged Navier Stokes (RANS) equations, with the turbulent viscosity calculated by means of the 1-equation model developed by Spalart and Allmaras (1994). The problem is closed by enforcing appropriate conditions at the physical and the computational boundaries. On solid walls, velocity is set at zero (while no condition on the pressure is required). At the inflow boundary, velocity is set at the undisturbed flow value, and the pressure is extrapolated from inside. At the outflow, the pressure is set at zero for flow without the free surface, while zero pressure gradient is enforced for free-surface flows, and the velocity extrapolated from inner points. At the free surface, the dynamic boundary condition requires continuity of stresses across the surface, while its location is determined by the kinematic condition. On the bottom, the free-stream velocity and zero pressure gradient conditions are enforced.

Here, only the main aspects of the numerical method are briefly recalled; the reader interested in more details is referred to Di Mascio et al. (2001, 2006). When only the average steady state has to be computed, the RANS equations can be conveniently replaced by a pseudo-compressible formulation (Chorin, 1967). In the numerical scheme, these equations are approximated by a finite volume technique with pressure and velocity co-located at cell centre. Discretization of the physical domain is achieved by means of a multiblock structured grid with partial overlapping (Muscari and Di Mascio, 2005) applied to simplify the discretization of appendages and to include blocks for local refinement. Viscous terms are computed by means of a standard 2nd-order centred finite volume approximation, while for the inviscid part, a 2nd-order Essentially Non Oscillatory (ENO) scheme has been adopted (Harten et al., 1987). Time integration of the discrete model is achieved by means of an implicit Euler scheme; the resulting discrete system of algebraic equations is solved in delta form (Beam and Warming, 1978). Convergence toward steady state is accelerated by local time stepping and a multigrid algorithm (Favini et al., 1996). The free surface is handled by a single-phase level set algorithm discretized by an ENO technique (similar to the one used for the bulk flow).

The presence of the propeller is taken into account by a model based on the actuator disk concept, according to which both axial and tangential forces are distributed in the flow field within a disk of finite thickness, in order to simulate both the acceleration and the increase in swirl that the flow undergoes when passing through

---

Received April 20, 2006; revised manuscript received by the editors December 25, 2006. The original version (prior to the final revised manuscript) was presented at the 16th International Offshore and Polar Engineering Conference (ISOPE-2006), San Francisco, May 28–June 2, 2006.

KEY WORDS: RANSE-based simulations, manoeuvring, turbulent flows, submarine, confined water.