11th International Workshop on Ship and Marine Hydrodynamics
Hamburg, Germany, September 22-25, 2019

A Linearized Free-Surface RANS Method for Self-Propulsion and Maneuvering

Paolo Geremia[1,*], Kevin J. Maki[2], Pavlos Alexias[3]

[1]ENGYS S.R.L., Strada per Vienna, 9, 34151 Trieste, Italy
[2]University of Michigan, Naval Architecture and Marine Engineering, Computational Ship Hydrodynamics Laboratory, USA
[3]ENGYS Ltd., Studio 20 - Royal Victoria Patriotic Building, John Archer Way, London SW18 3SX, United Kingdom

*Corresponding author, p.geremia@engys.com

ABSTRACT

Numerical prediction of the hydrodynamical performance of unsteady ship operations, such as self-propulsion and maneuvering, is an important method to help naval architects design optimal ship hulls. Currently, traditional finite-volume Computational Fluid Dynamics (CFD) methods offer a well-proven simulation platform to realize such predictions with a high degree of accuracy. In this work, a novel transient CFD method based on an unsteady linearized free-surface RANS solver is presented for the objective of simulating ship maneuvering. The specific results presented in the paper are for the self-propulsion and turning circle of the Korean Containership KCS. The results achieved, show that the new linearized free-surface RANS solver provides a viable and more cost-effective alternative than traditional Volume-of-Fluid (VOF) methods when applied to streamlined bodies, such as displacement hulls. The key new developments highlighted in this paper are in the way that self-propulsion is included and a new robust mesh motion method to handle the relative motion of the rudder.

1 INTRODUCTION

CFD solvers employed in the study of ship hull hydrodynamics are usually based on either Volume-Of-Fluid (VOF) [1] or level-set methods [2], both implemented as part of the Reynolds Averaged Navier Stokes (RANS) equations in a finite-volume framework. These methods have proven to deliver highly accurate predictions for hull resistance and other important hydrodynamic performance parameters, but they often require large parallel computers and long simulation times. Furthermore, the robustness of the VOF and level-set solvers requires highly customized grid generation techniques and small time-step sizes to retain a robust and accurate solution. The combination of these shortcomings precludes in most cases the application of these methods at the early stages of the design process.

In order to overcome the high computational costs associated with traditional VOF and level-set methods, a new RANS based Linearized Free-Surface (LFS) solver with viscous effects was successfully implemented and employed to perform fast hull-form optimization using a steady-state formulation [3]. This solver allows for faster predictions of hull resistance and other parameters without compromising the overall accuracy of the results. The underlying formulation of the LFS is based on the slender body theory for the freesurface which shows consistent results for streamlined bodies at low Froude numbers, such as displacement hulls travelling in even keel conditions [3]. The LFS method was extended to unsteady flows in [6,8] where the unsteady RANS (uRANS) equations are solved with unsteady linearized free-surface boundary conditions.

The use of the uRANS approach is fundamental to accurately predict both the pressure and viscous forces acting on the hull, including the interaction with the propeller and rudder components. In this context, the application of the LFS solver allows for a more efficient solution in terms of computational time, hence allowing the designer to investigate the effects of multiple hull motions in shorter times.

The current implementation of the unsteady LFS solver is limited to the horizontal-plane motions only, namely the surge, sway and yaw, which determine the 3 degrees of freedom (DoF). This means for it is best suited for maneuvers when the Froude number, roll and pitch angle are small.

2 DESCRIPTION OF THE FLOW SOLVER

The proposed uRANS LFS flow solver is based on the linearized unsteady Neumann-Kelvin ship wave boundary-value problem. The ship generated wave is assumed to be of small amplitude and steepness, and the nonlinear free-surface boundary condition can be satisfied in a linearized form on the calm-water plane. This allows for a double-body discretization to be used together with a single-phase flow solver.

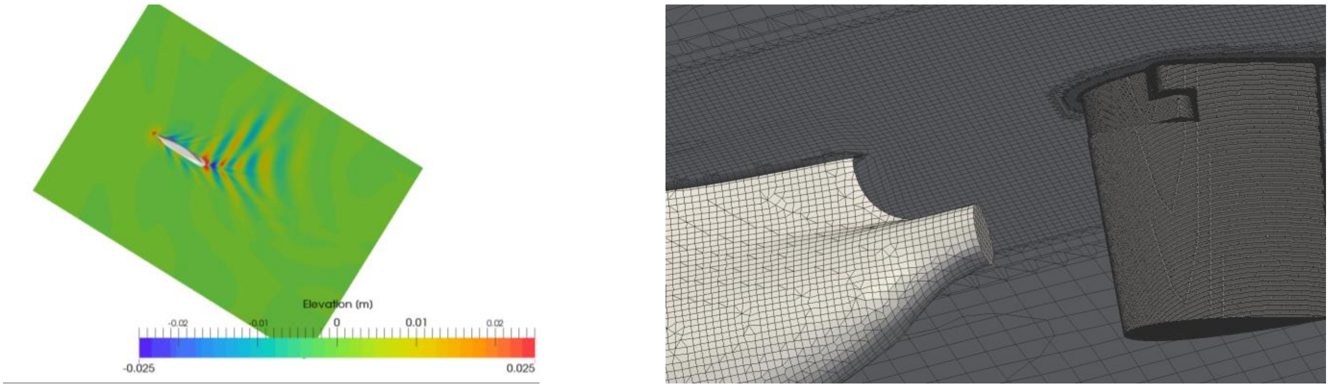
In the present work, the unsteady kinematic condition is coupled to the unsteady uRANS equations via the dynamic free-surface condition that is applied to the free-surface boundary of the domain. The mathematical details and extensive validation of the formulation can be found in [6, 8].

A novel improvement to the unsteady LFS solver is the way in which the dynamic mesh motion (which includes the 3-DoF of the in-plane motion) and the Multiple Reference Frame (MRF) are combined.

The 3-DoF motion capabilities also make this model applicable to a wide range of problems, including the free-fall of a body that impacts the air-water interface, the seakeeping response of a surface vessel or submarine, or the maneuvering response of a ship.

For maneuvers, it is imperative to be able to model the motions of both the rudder and propeller explicitly. In order to avoid the use of complex and expensive overset grids methods, a simpler option would be to use a sliding mesh approach to cope with the relative movements between component. However, this is impossible in most cases due to their close proximity (Figure 1(b)).

The alternative method proposed herein relies on mesh motion to model the rudder movement with a sliding mesh to model the propeller rotation. To perform the mesh motion around the rudder in an effective way, it is necessary to use a fast and reliable mesh adaptation algorithm to follow the boundary movement. The mesh deformation algorithm employed in this work is based on the solution of a partial differential equation coupled with a sophisticated mesh optimization technique that guarantees high mesh quality, as described in [9].



(a) Free-surface elevation. (b) Computational mesh.

Figure 1: Sample free-running maneuvering test.

3 GRID CONVERGENCE AND ACCURACY FOR STATIC DRIFT

The first application considered in this paper is a series of captive tests performed on the KRISO Container Ship (KCS) model [5]. The results had been previously published in [10] and are repeated here to emphasize the convergence properties of the LFS method for this hull form. The characteristics of the KCS hull are summarized in Table 1.

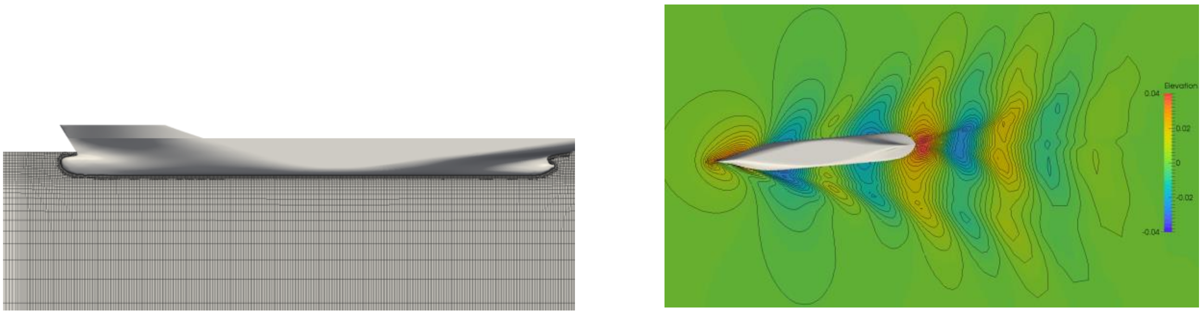
Table 1: KCS Geometry and conditions for PMM tests

Scale	52.667
Lpp [m]	4.367
Bwl [m]	0.611
T [m]	0.205
U [m/s]	1.701
Fn [-]	0.26

The maneuvering simulation tests are performed in even keel conditions with dynamic sinkage and trim suppressed to mimic the results obtained in the towing tank experiments when using the PMM system. For all captive test simulations completed using the uRANS LFS method, the two equations k-ω SST turbulence model is used due to its proven ability to predict accurately pressure forces when applied to ship hydrodynamics.

The first test performed as part of this study is a static drift maneuver of the KCS model with a yaw amplitude ξ_6= 5 deg, travelling at a constant speed equivalent to $Fn = 0.26$ in calm-water conditions. The ship is started from rest and accelerated over a time of 2.5 s until it reached the target speed.

Three grids labelled coarse, medium and fine grid were used with 321K, 843K and 2.8M cells, respectively. The mesh layout and the resulting free-surface elevation for the fine grid case are shown in Figure 2.



(a) Fine mesh overview. (b) Free-surface elevation.

Figure 2: Static drift computational model and results.

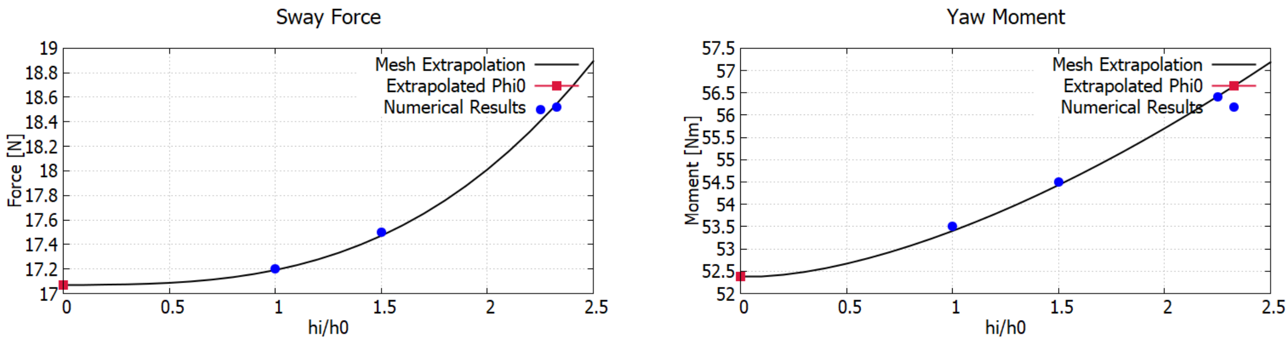
To check the convergence of the solver against the grid size, the sway force and yaw moment output responses are considered. Assuming h_i as the reference size of the i-th grid, where h_1 is the fine mesh reference cell size, the Richardson extrapolated exact solution ϕ_0 is calculated according to the following formula:

$$\phi_0 = \phi_1 + \frac{\phi_1 - \phi_2}{r^p - 1} \qquad (1)$$

where p is the order of the interpolation defined as follows:

$$p = \frac{ln[(\phi_3 - \phi_2)/(\phi_2 - \phi_1)]}{ln(r)} \qquad (2)$$

and $r = h_{i+1}/h_i$ is the constant ratio of refinement between the grids. The extrapolated solution ϕ_0 as well the mesh extrapolation curves are plotted in Figure 3 for both the sway force and the yaw moment.



(a) Sway force. (b) Yaw moment.

Figure 3: Force and moment convergence with mesh refinement.

It can be noticed that a monotone convergence is achieved for both the forces considered in Figure 3. The same can be also seen if the Grid Convergence Index (GCI) is defined as follows:

$$GCI = 1.25 \frac{|\phi_{i+1} - \phi_i|}{\phi_i} \frac{1}{r^p - 1} \tag{3}$$

In fact, an asymptotic range of convergence is obtained for both sway force and yaw moment, as follows:

$$\begin{aligned} \left(\frac{GCI_2}{r^p GCI_1}\right)_{Surge\ Force} &= 0.984 \approx 1 \\ \left(\frac{GCI_2}{r^p GCI_1}\right)_{Yaw\ Moment} &= 0.980 \approx 1 \end{aligned} \tag{4}$$

4 FREE-RUNNING TESTS

The newly developed uRANS LFS solver is also employed on a series of free-running tests, including self-propulsion, turning circle and zig-zag maneuvers. In order to test the performance of the new mesh motion libraries described in this paper, the coupling between discretized rotating propeller motion and rudder motion is analyzed. First, a self-propulsion test at the ship point in calm-water conditions with an applied thrust force is performed to measure the resulting target velocity and the hull resistance force. Secondly, the same self-propulsion test is repeated using a discretized propeller instead of a thrust model. Finally, to check the robustness of the new rudder motion library, a simplified turning circle maneuver is completed.

The test case considered for these validations is the KCS hull form in deep water conditions according to the SIMMAN 2014 workshop specifications. The aim of this test was to evaluate the ability of the uRANS LFS to predict accurately the forces acting on the hull body and the propeller system in preparation for subsequent free-running tests, including zig-zag maneuvers, to be performed as part of the workshop tests. The KCS model test geometry and conditions are outlined in Table 2.

Table 2: KCS Geometry and conditions for the self-propulsion preliminary test

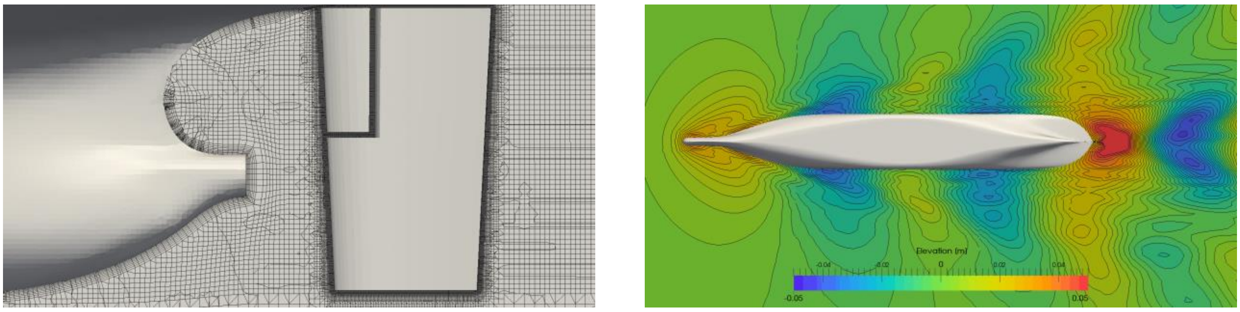
Scale	37.890
Lpp [m]	6.070
Bwl [m]	0.850
T [m]	0.285
U [m/s]	2.005
Fn [-]	0.26

4.1 Self-propulsion Model Tests

The first self-propulsion test is carried out using a simplified thrust model with the KCS hull in even keel conditions and 1 DoF enabled. A constant thrust force is applied on the ship body without modelling the propeller geometry to simulate the hull travelling at ship-point at a target speed equivalent to $Fn=0.26$, corresponding to 24 knots at full scale.

The k-ω SST turbulence model is employed with an automatic wall function (all y+ values). For the solver settings, a first-order Euler scheme for time and a second-order scheme for advection are applied. Two PIMPLE outer solver iterations are used to improved solution convergence with a time-step tuned to reach a maximum Courant number of 10. Typically, for traditional VOF methods the Courant number should be below 1 to achieve a bounded solution for the phase fraction. The absence of a phase fraction equation in the LFS solver allows for larger time steps and faster solution times.

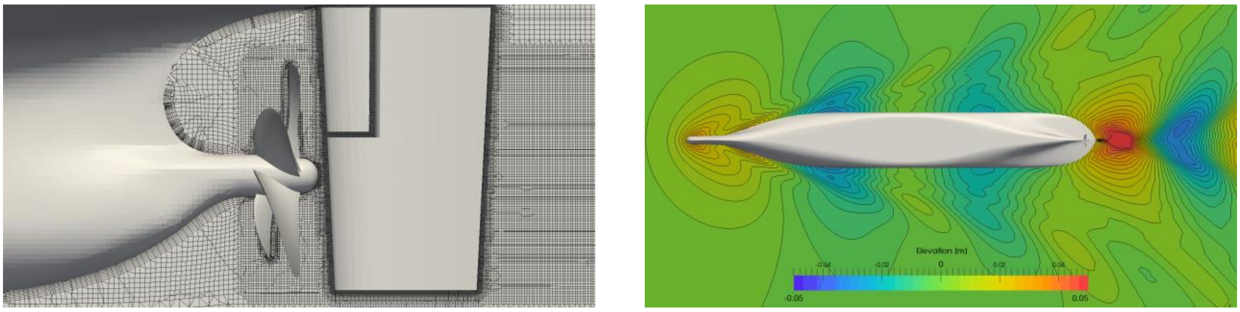
The computational mesh (7.8M cells) and the corresponding free-surface elevation field for the initial self-propulsion test are shown in Figure 4. The resulting resistance force measured using the thrust model on the hull is $R_t=52.186$ *N*.



(a) Mesh aft section. (b) Free-surface elevation.

Figure 4: Self-propulsion case with applied thrust force.

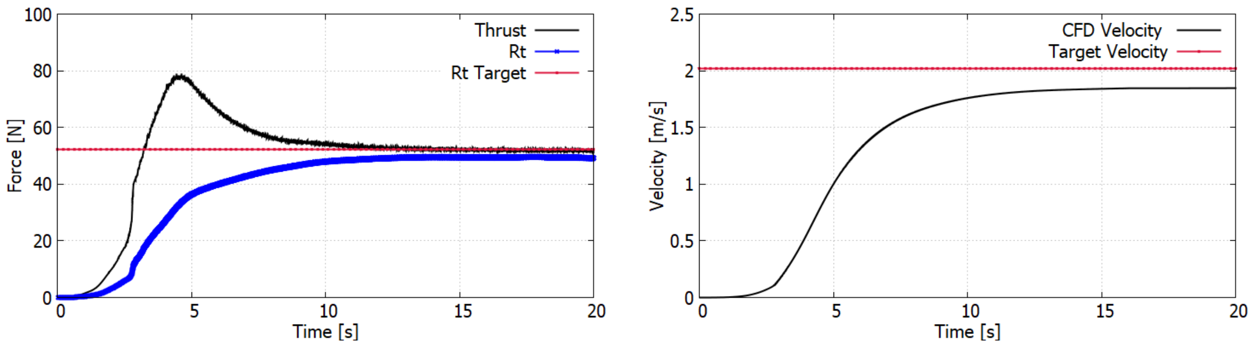
The second self-propulsion test model is completed using a 9.1M cells computational grid with a fully discretized propeller geometry made to rotate using a sliding mesh technique. The propeller rotational speed is set to 11.5 rps based on the experimental results provided by MARIN for the SIMMAN 2014 workshop. The same turbulence model, discretization schemes and solver settings applied in the first test described in Figure 4 are employed here to deliver an average propeller angular resolution of 4.2 deg per timestep. The resulting grid and free-surface elevation for the second test are shown in Figure 5.



(a) Mesh aft section. (b) Free-surface elevation.

Figure 5: Self-propulsion case with discretized propeller.

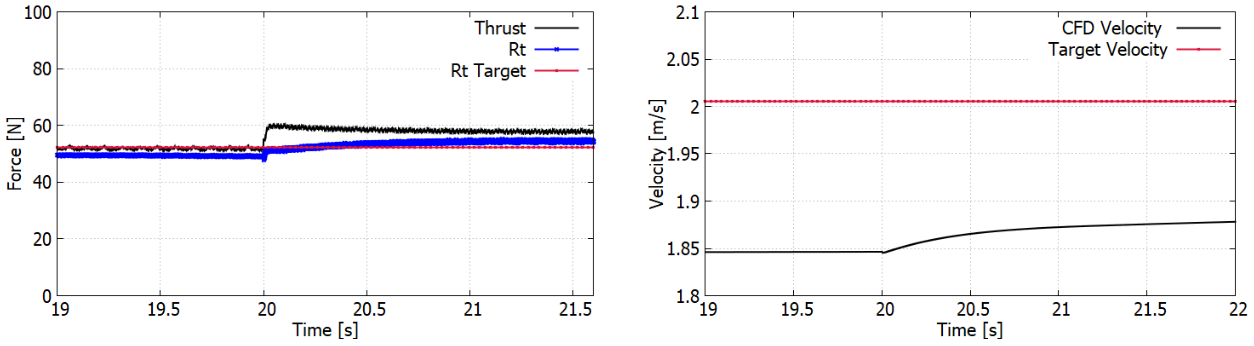
The total force acting on the hull and propeller for the second self-propulsion test model are shown in Figure 6 as a function of time. The target velocity calculated with the uRANS LFS solver is slightly lower than the experiments. The lower velocity in the second case model also explains the slight differences observed in free-surface elevation between the first and second cases, as seen in Figures 4 and 5.



(a) Thrust and resistance forces. (b) Hull velocity.

Figure 6: Forces and velocity measured on the KCS hull for fixed propeller rpm.

The under-prediction of the target hull velocity is primarily due to the lower resistance force. There are several factors which contribute to the slightly under-predicted resistance, including the wave linearization, near wall modelling accuracy, and the time and spatial discretization error. To quantify the role of time discretization error, the time step size was reduced by a factor of ten at time of 20 secs. The results are shown in Figure 7, where one can see that the thrust increases more than the resistance, and, the ship velocity increases and is closer to the target velocity. This underscores the importance of the time-step size to accurately resolve propulsive forces.



(a) Thrust and resistance forces. (b) Hull velocity.

Figure 7: Forces and velocity measured on the KCS hull for fixed propeller rpm with fine time step.

To quantify the speed-up in terms of computational time of the proposed uRANS LFS solver compared to a traditional VOF solver, an additional run is carried out on a different grid using a two-phase VOF solver with the same surface and volume discretization settings as before. The computational times required by the two solvers to simulate 20 s of physical time are presented in Table 3. As expected, the LFS solver needs around 38% of the time required by the VOF solver for completion.

Table 3: Self-propulsion tests: Computational times for LFS and VOF solvers

LFS Solver (CPU hours)	VOF Solver (CPU hours)
1959	5291

4.2 Rudder Motion

For the final test case, a 3 DoF model of the KCS hull is setup to model the initial part of the trajectory of a 20 deg turning-circle maneuver with free heave, sway and yaw. The aim is to verify that the new mesh motion capabilities implemented as part of the uRANS LFS solver work properly when the rudder and propeller are moving.

The same computational grid and solver settings employed in the previous self-propulsion cases were used to complete the turning circle maneuver. For this purpose, a 20 deg rudder rotation was imposed with a 14.28 deg/s turning rate.

At the time of submission of the paper 10 s of physical time after the rudder is actuated has been completed. The corresponding trajectory is shown in Figure 8.

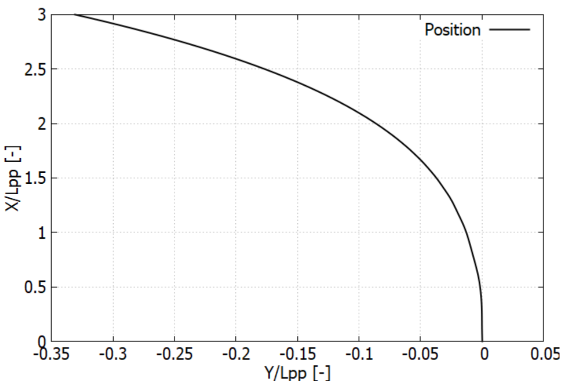
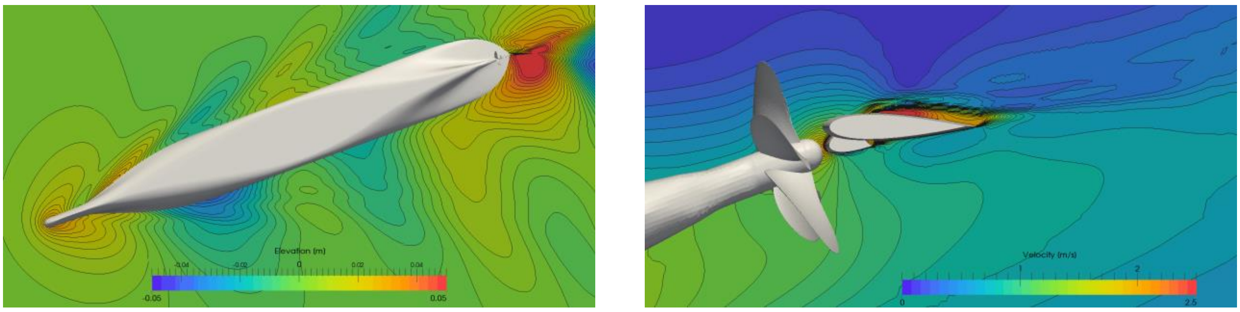


Figure 8: Simplified turning circle maneuver trajectory.

The free-surface elevation and the velocity fields are shown in Figure 9 after 10 s the rudder completed the turning motion.



(a) Free-surface elevation. (b) Velocity at $x/L_{pp} = -0.01$.

Figure 9: Solution fields for the simplified turning circle.

5 CONCLUSIONS

In this paper an efficient unsteady CFD solver based on a Linearized Free-Surface (LFS) approach coupled with an accurate rigid-body motion framework to simulate ship hull maneuvering is presented. The proposed LFS solver is initially validated on a series of PMM tests based on experimental measurements publicly available in order to develop best-practices and recommended settings for both the creation of the computational grid and the setup of the CFD solver. In addition, a robust mesh deformation algorithm for rudder motion coupled with a propeller motion library is developed and a series of preliminary tests to carry out more advanced zig-zag and turning circle maneuvers are performed. For this purpose, self-propulsion tests and simplified turning circle maneuver are shown and a robust behavior of the LFS solver coupled to the mesh motion library is demonstrated. Further work is required to fully evaluate the existing CFD models in comparison to the experiments.

The results achieved with the new solver show that it provides a viable and more cost-effective alternative than traditional VOF methods, since the LFS solver can run with high CFL numbers as opposite to the traditional VOF methods to deliver faster solutions for the correct prediction of ship behavior in the early stages of the design process. Further work is being carried out at present to validate the linearized free-surface solver framework for more complex free-running maneuvers, such as course keeping via propeller and rudder active control, turning circle, and zig-zag tests.

REFERENCES

[1] Hirt, W. C. and B. D. Nichols, "Volume of fluid (VOF) method for the dynamics of free boundaries", In: *Journal of Computational Physics*, **39**(1981), 201-225

[2] Yang, J. et al.,"Two Phase Level-Set/Immersed-Boundary Cartesian Grid Method for Ship Hydrodynamics", *9th International Conference on Numerical Ship Hydrodynamics*, Ann Arbor, Michigan, August 5-8, 2007

[3] Rosemurgy, W. J., D. O. Edmund, K. J. Maki and R. F.Beck, "A Method for Resistance Prediction in the Design Environment", *11th International Conference on Fast Sea Transportation*, 2011.

[4] Maki, K. and D. Piro, "Whipping response of a box barge in oblique seas". *28th International Workshop on Water Waves and Floating Bodies,* 2013.

[5] Kim, W.J., S.H. Van and D.H. Kim, "Measurements of Flows around modern commercial ship models". In: *Experiments in Fluids*, **31**(2001), 567-578.

[6] Woolliscroft, M. O., "A Linearized Free-Surface Method for Prediction of Unsteady Ship Maneuvering", *PhD Thesis*, University of Michigan, USA, 2015.

[7] Otzen, J.F., and C.D. Simonsen editors, *Proceedings SIMMAN 2014 Workshop*, Copenhagen, Denmark, 2014.

[8] Woolliscroft, M. O., and K. J. Maki. "A fast-running CFD formulation for unsteady ship maneuvering performance prediction." *Ocean Engineering* **117**(2016), 154-162.

[9] P. Alexias, E. De Villiers, "Sphericity: Mesh Optimization for Arbitrary Element Topology", In: *Proceedings Eccomas Congress 2016*, Greece, 2016.

[10] P. Geremia, K. J Maki and P. Alexias, "A Linearized Free-Surface RANS Method for Ship Maneuvering", In: *Proceedings VIII International Conference on Computational Methods in Marine Engineering*, Gothenburgh, Sweden 2019.