Overall performances of a propeller operating near the free surface

A Lungu
"Dunarea de Jos” University of Galati, Faculty of Naval Architecture, Department of Naval Architecture, Domneasca Street No. 47, Galati 800008, ROMANIA
E-mail: adrian.lungu@ugal.ro

Abstract. The paper describes a numerical study on the effects of the interaction of the free surface with a marine propeller operating close to it, including the effects of the immersion on the cross flow. Extensive simulations using the ISIS-CFD naval hydrodynamics computational fluid dynamics solver are conducted to study the open water characteristics (POW hereafter), transient blade loads, and flow behavior around the DTMB 4119 three-bladed propeller, with comparisons to the fully submerged experimental data at two different immersions in respect to the undisturbed free-surface. Computations and experiments for the deeply submerged case show that the cross flow results in an increase of the energy spent and lower efficiency relative to the uniform inflow. Near the water surface, numerical solutions show that effects on thrust and torque increase significantly as the propeller load increases. Furthermore, the presence of the free surface breaks the symmetry resulting in highest blade force losses when the blade is near top dead center. As the propeller approaches the surface, not only the amplitude of the blade higher harmonics increases, but also the efficiency drops down.

1. Introduction
In certain working conditions, it may sometimes happens that a propeller approaches significantly the water surface, a fact that may easily lead to unwanted associated effects out of which the most important one is the ventilation which is accompanied by the decrease of the overall efficiency. This can be induced by a number of different causes, such as the insufficient draught for navigation in ballast condition, excessively high trim angles, stepped hulls, or even hull appendages. As the propeller acts in an aerated water environment, the water flow may easily detach from the blades. Since the water streamlines detach, the propeller loading reduces and the necessary torque applied to the propeller can more easily spin the propeller to a much higher rate of revolution even at the same given power input. This eventually reduces the thrust provided by the propeller, slowing the ship speed and/or acceleration, as well as reducing the propeller control over the hull. Summing up, ventilation typically occurs when the propeller loading is high and the propeller submergence is limited, which is most likely to happen in heavy seas when the relative motions between the free surface and the propeller are large.

Propeller ventilation inception depends on various initial conditions, such as propeller loading, forward speed and the clearance from the propeller to the free surface [1-3]. In such cases, the numerical treatment has to be based on special provisions at it will be described in the followings. On the contrary, when the propeller is sufficiently far of the free-surface, the working conditions are not influenced at all, therefore the open water performances can be taken into account and no special
attention has to be paid to the numerical simulation, as previously described by the author in [4-6]. Once the unwanted ventilation phenomenon takes place, the hydrodynamic parameters that describe the flow at the ship stern changes rapidly, vibrations are likely to occur so the local and global hull structure strength state may therefore be affected.

When conceiving a simulation setup for studying a possible ventilation one may cover all the regimes and submergences at which the propeller works. Two different ventilation inception mechanisms depending on the level of submergence of the propeller were described in [7], showing that either ventilation can start by forming an air-filled vortex from the free surface, and the free surface can be sucked down to the propeller, or it becomes surface piercing, such that air can enter the suction side of the blade directly from the atmosphere. Doubtlessly, the free surface vortex ventilation mechanism is characterized by significant thrust losses which occur when a vortex that develops on the blade surface funnels a considerable quantity of air from the free surface down to the suction side of the blade. It has been proven that vortex ventilation happens only for high thrust loadings at low forward speeds. Surface piercing ventilation is characterized by uniform thrust losses over a complete revolution of the propeller.

As mentioned above, depending on the working conditions, a propeller might be working normally [8-11], or may be partially or fully-ventilated depending on several factors, where submergence and advance coefficient are meeting the critical criteria of the occurrence. Obviously, the thrust losses are depending not only on the ventilation. A larger thrust loss is caused by the propeller coming partly close to or even out of the water surface so that the effective propeller disk area may get significantly reduced. When the propeller is partly out of the water, thrust loss can be computed from the fraction of the propeller disc area that is above the water. This is typically complemented by adding the loss caused by the so-called Wagner effect [12], which accounts for the dynamic lift effect of the immersed propeller blades.

The present study is focused on the open water characteristics of the DTMB 4119 propeller computed by solving numerically the unsteady viscous equations of the three-dimensional flow. Closure to turbulence is achieved through the detached eddy simulation (DES hereafter based on shear stress transport (SST) model. The paper contains a grid convergence tests as well as a verification and validation estimate of the ISIS-CFD solver of the Numeca Fine™/Marine software package. Comparisons with the available experimental data are performed for proving the accuracy of the employed theoretical method.

2. Propeller geometry, computational domain and grid generation
The three bladed propeller model is of a diameter of 305 mm. The propeller hub of a purely cylindrical geometry is of a relative radius of 0.1967. Its blades of a modified NACA66 profile, having neither skew nor rake, are embedded into the shaft without any fillet at their roots, as shown in figure 1, which depicts the propeller. The geometry of DTMB 4119 propeller is given in table 1, which tabulates for each relative radius \( r/R \), the relative chord \( c/D \), the relative pitch \( P/D \), maximum thickness \( t_{\text{max}}/c \) and maximum camber distribution \( f_{\text{max}}/c \), where \( R \) and \( D \) are the propeller radius and diameter.

| \( r/R \) | \( c/D \) | \( P/D \) | \( t_{\text{max}}/c \) | \( f_{\text{max}}/c \) |
|---|---|---|---|---|
| 0.2 | 0.3200 | 1.1050 | 0.2055 | 0.0143 |
| 0.3 | 0.3635 | 1.1022 | 0.1553 | 0.0232 |
| 0.4 | 0.4048 | 1.0983 | 0.1180 | 0.0230 |
| 0.5 | 0.4392 | 1.0932 | 0.0916 | 0.0218 |
| 0.6 | 0.4610 | 1.0879 | 0.0696 | 0.0207 |
| 0.7 | 0.4622 | 1.0839 | 0.0542 | 0.0200 |
| 0.8 | 0.4347 | 1.0811 | 0.0421 | 0.0197 |
| 0.9 | 0.3613 | 1.0785 | 0.0332 | 0.0182 |
| 0.95 | 0.2775 | 1.0770 | 0.0323 | 0.0163 |
| 1.0 | 0.0800 | 1.0750 | 0.0316 | 0.0118 |

![Figure 1. DTMB 4119 propeller.](image)
Figure 2 shows not only the computational domain on which the main boundaries are marked, figure 2(a), but also the discretization grids drawn on the propeller, figure 2(b), and in its immediate surrounding area, figure 2(c). Simulations described in the present paper are rather different from those for classical POW computations where the computational domain is of a cylindrical geometry, as described in [13]. Since the main interest is related to the free-surface presence, the computational domain is rectangular here, similar to that built for a classical ship resistance simulation. Under such circumstances, a special attention concerning the cells clustering close to the free-surface water as well as to the tip, leading and trailing edges of the blades and their roots is paid since those areas are expected to host large gradients of the physical parameters of the flow. Moreover, to simulate the flow around the propeller, a sliding grid shown in figure 2(c) is built around the rotating propeller. Whenever dealing with a free-surface flow problem, a key parameter of the computation is the Froude number whose value is calculated based on the characteristic length. In the atypical case of the computation reported in here, Froude number is based on the total length of the propeller and shaft set at 1m and on an axial incoming velocity of 1 m/s.

![Figure 2](image)

**Figure 2.** Computational domain and grid. (a) computational domain; (b) computational grid around the propeller; (c) sliding grid.

3. Numerical approach

The ISIS-CFD flow solver of the FINE\textsuperscript{TM}/Marine is employed in the study to investigate the flow field structure around the hull, based on a VOF approach. The numerical simulation of the free-surface is performed by using a free-surface capturing strategy. The solver uses algorithms providing a strong pressure-velocity coupling for the RANSE, whereas an automatic grid adaptation by a posteriori error estimation is employed to resolve the local issues of the flow. The simulation is accomplished in a global approach in which RANS equations written in respect to a Cartesian system of coordinates are solved numerically.

The turbulence is treated by making use of the SST DES model $k - \omega$, according to which when the turbulent length scale exceeds the grid dimension, the model switches to a subgrid scale formulation and the flow is solved using the LES model. The DES approach is based on an implicit splitting of the computational domain into two zones. In the region near solid walls, the conventional RANS equations are solved. Within the second region, the governing equations are the filtered Navier Stokes equations of the LES approach. Since the hybrid nature of DES is not linked to any specific turbulence model, the one employed in here is a variant based on the $k - \omega$ SST model of Menter [14].

The forces integration is performed on the solid-surface cell based on the quaternions formulation. The full tensor is considered for the moments of inertia. The integration in time is done in an Euler explicit way, whereas an upwind discretization scheme is used for the convective terms with a second order for the acceleration. Conservation applies to the mass and momentum and a Piccard model applies for the linearization. The pressure-correction is imposed and the Krylov technique is used for the iteration of the solution. A non-structured body and boundary fitted grid with hexahedral elements generated in Hexpress is used for the domain discretization. The advanced smoothing capability of the
grid generator provides not only a high-quality boundary layers insertion, but also a close control over the cell clustering in the free-surface vicinity.

An unsteady approach is used to advance the solution in time. The initial conditions refer to the incoming flow velocity, pressure and turbulent viscosity only. The computational domain is limited at ~1.5 propeller shaft lengths at the upstream where the velocity components, the pressure and the turbulent viscosity are imposed, whereas a Neumann condition is imposed for the pressure. At the downstream, which is located at 3 shaft lengths, the velocity components and the turbulent viscosity are zero-extrapolated, whereas the pressure has the static value. The no-slip condition is imposed on the propeller surfaces. For the upper and bottom boundaries the pressure is prescribed at the static value and kept unchanged over the entire simulation. Laterally, on the boundaries located at 1.5 shaft lengths on each side, all the hydrodynamic parameters are extrapolated with zero-gradients.

The flow starts from rest and it is accelerated within 2 seconds up to the given oncoming velocity. Firstly the flow is computed based on the RANS method in which closure to turbulence is achieved through the $k − \omega$ SST model till forces and moments stabilize in time. Usually this step requires about 15 seconds. After that initial step the turbulence model is switched to the DES, the time step is decreased so that the $y^+$ be less than 0.3, the number of the iteration per time step is increased up to 20 and the solution is restarted for a couple of more seconds.

4. Results and discussions

Three different sets of computations are carried out in the followings. The first one regards the POW performances. It is preceded by a grid convergence test aimed at establishing the limits of the cumulated numerical errors of the simulation. For that purpose, comparisons with the experimental data [15, 16] are provided. The second set regards the assessment of the propeller performances when working below the free surface at a depth equal to one diameter. Finally, the third set refers to the propeller working at half of a diameter beneath the otherwise undisturbed water surface.

4.1. Grid convergence test

In the present study the gridding is done by using the Hexpress module of the FINE\textsuperscript{TM}/Marine, a generator which has not only direct CAD import capabilities, but it also allows the manipulation or the decomposition of the geometry. The automatic full hexahedral grids generation is possible as the buffer cell and boundary layer insertion for high quality cells in boundary layer regions. An automatic refinement procedure based on defined sensors placed either next to solid walls or inside specified area in the domain is also available. Three different grids varying from 4.59 to 19.27 million cells were generated for the sake of performing the grid convergence test for the solver. Let these grids be denoted by G\textsubscript{1}, G\textsubscript{2} and G\textsubscript{3} in table 2, which tabulates a comparison between the computed solutions (CFD) and the corresponding POW experimental data (EFD), [16], performed for the grid convergence test. For the sake of conformity is should be mentioned that the intermediate grid G\textsubscript{2} comprises 9.13 million cells.

| Table 2. Grid convergence test computations. |
|--------------------------------------------|
| J   | 0.5 | 0.7 | 0.833 | 0.9 | 1.1 | 0.5 | 0.7 | 0.833 | 0.9 | 1.1 |
| EFD | 0.285 | 0.2 | 0.146 | 0.12 | 0.034 | 0.477 | 0.36 | 0.28 | 0.239 | 0.106 |
| G\textsubscript{1} | 0.2756 | 0.1939 | 0.1418 | 0.1169 | 0.0332 | 0.4606 | 0.3478 | 0.2709 | 0.2318 | 0.1029 |
| $|e|\%$ | 3.31 | 3.02 | 2.89 | 2.61 | 2.36 | 3.44 | 3.41 | 3.25 | 3.01 | 2.96 |
| CFD | 0.2767 | 0.1948 | 0.1428 | 0.1176 | 0.0333 | 0.4618 | 0.3490 | 0.2716 | 0.2321 | 0.1034 |
| $|e|\%$ | 2.93 | 2.61 | 2.21 | 2.04 | 1.99 | 3.18 | 3.06 | 2.99 | 2.86 | 2.47 |
| G\textsubscript{2} | 0.2773 | 0.1953 | 0.1429 | 0.1177 | 0.0334 | 0.4630 | 0.3502 | 0.2730 | 0.2334 | 0.1038 |
| $|e|\%$ | 2.71 | 2.37 | 2.11 | 1.93 | 1.71 | 2.93 | 2.72 | 2.51 | 2.33 | 2.12 |
Based on the figures in table 2, it may be worth to underline that the absolute error varies from 3.44\% for the torque coefficient computed on the coarsest grid to 1.71\% for the thrust coefficient computed on the finest grid, a fact which may suggest not only the accuracy of the numerical solution, but also the robustness of the solver. For obvious reasons, only the solutions computed on the finest grid will be presented and discussed in the followings.

4.2. Open water propeller performances

The performance coefficients computed on the finest grid for the propeller working in open water conditions are compared with the experimental data at five advance coefficients in figure 3, in which the experimental data are represented by solid lines whereas the numerical solutions by symbols. The figure also contains the error bars for each computation performed in the present study. As shown in table 2, the largest level of errors corresponds to the heavier working regimes, i.e. to those related to lower streamwise velocity, a fact which agrees with previous findings of the author, reported in [6, 13].

![Figure 3. Comparison between the computed and measured [15] thrust and torque coefficients.](image)

A key issue for any propeller study is represented by the axial velocity distribution in the propeller plane and downstream of it since this velocity component has an overwhelming contribution to the effective wake. Previously reported comparisons with the LDV experimental data have shown that for a highly loaded propeller significant errors still exist in the axial velocity in the disk, especially near the tip. Moreover, despite the uniform load, the axial velocity in the propeller plane varies in the radial direction. Instead, the velocity magnitude remains almost uniform only for thrust coefficients below unity. Under such circumstances, in the followings a comparison of the normalized streamwise velocity fields computed in the propeller plane and downstream of it, at distances equal to 10\% and 20\% of the propeller diameter $D$, is proposed in figure 4.

Analysing figure 4, drawn for a computation performed at $f = 0.9$, it may be seen that all the existing LDV observations are confirmed by the numerical solution. That is, the axial velocity having its highest value in the hub region varies significantly in the radial direction towards the tips, figure 4 (a) as the maximum magnitude of the normalized velocity remains constant within the range between 0 on the blades and 1.2, figures 4 (a), (b) and (c). Aside of that, one may see that going down the stream the velocity decay becomes important in the hub area because of the rope vortex that develops there, as figure 5 bears out. Figure 5 depicts the longitudinal field of the streamwise component of velocity normalized by the oncoming flow velocity, together with the vortices released by the tips and washed in the downstream. The three vortical structures are coloured by helicity to suggest the intensity and the propagation direction of the helical structures since the helicity density measures how an instantaneous streamline (an integral curve of the velocity) is close to a right-hand screw, favourable in positive helicity. Figure 5 reveals that the vortices trajectories are related to the lower
values of the axial velocity which appears in the areas of the tips, a fact which is confirmed by the physics behind.

![Diagram](image)

**Figure 4.** Computed axial velocity field at $J=0.9$: (a) - in the propeller plane; (b) - at 0.1$D$ downstream the propeller; (c) - at 0.2$D$ downstream the propeller.

Simulating propeller-induced jet velocities that extend from near to far fields represents a key step in elucidating the mean and turbulent flow fields downstream of a rotating propeller. Several previous studies [6, 13, 15] have proven that the propeller-induced swirling effect is directly proportional to the propeller rotational speed, but decreases as the distance from the efflux plane increases. As said before, this effect has also a significant effect on the radial distribution of the axial velocity and on the decay of the maximum velocity.

![Diagram](image)

**Figure 5.** Axial velocity distribution in the longitudinal plane of symmetry computed at $J=0.9$.

Since the velocity field is always closely related to the vorticity distribution and ultimately to the turbulent kinetic energy (TKE hereafter) for a given flow condition, an analysis of the distribution of these parameters in the propeller wake is further proposed in the followings. Figure 6(a) depicts the vorticity distribution in the wake of the propeller, computed for the same advance coefficient of 0.9. Areas corresponding to the propeller tips in which the axial velocity registered minimal values are characterized by maximal values of the vorticity. Similarly, high values of the vorticity are seen immediately at downstream of the cap where the rope vortex develops within a certain distance from the propeller. Pilot computations proved that the extension and intensity of this vortex are consistently dependent on the advance coefficient, being stronger at higher advance ratios, therefore at weaker loading conditions of the propeller.

As mentioned before directly linked to the flow dynamics is the TKE, which manifests the same periodic distribution as the velocity, a fact which was expected indeed, see figure 6(b). It has been already proven that if the vortices are advected in the wake of the propeller far from any boundary layer, turbulence in the core of the vortex should be quickly dissipated, due to the regularizing effects
of the swirl. In that case the DES-SST model may predict better than any eddy-viscosity based turbulence closures since it is able to predict the re-laminarization of the core of the tip vortex in the wake of the propeller, as figure 6(b) shows. As a conclusion, although the DES-SST does not always behave better on the surface of the propeller, thanks to the quick attenuation of the turbulent kinetic energy in the core of the vortex once it leaves the blade surface, the model does not contaminate the vortex during its evolution in the wake.

![Computed vorticity and turbulent kinetic energy fields at J=0.9.](image)

**Figure 6.** Computed vorticity and turbulent kinetic energy fields at $J=0.9$.

### 4.3. Propeller working beneath the free-surface

When a ship operates in waves, complicated motions are expected to occur, therefore the propeller attached to the stern may undergo various inflow conditions and a change of submergence depth. A decrease of the submergence depth leads to an interaction between the propeller and free-surface so risks related to the air ventilation and surface piercing running may occur. In order to evaluate the probability for such a phenomenon to take place, it is desirable find a critical condition for its appearance. Under these circumstances, the hydrodynamic characteristics, such as the thrust and torque of propeller, the wave pattern of free surface, and the evolution of propeller wake will be investigated in the followings for three advance coefficients and two submergence depths of the propeller. The free-surface location is determined by making use of a volume of fluid approach based on a free-surface capturing strategy. The solver uses algorithms providing a strong pressure-velocity coupling for the equations of motion and continuity, whereas an automatic grid adaptation based on the pressure Hessian activated by an error estimation at the free surface is employed.

All the computations are performed for the same DTMB 4119 propeller shown in figure 1, at the same advance coefficients as for the POW computations, just to evaluate which are the losses induced by the free-surface proximity. The computational domain is that shown in figure 2(a). The propeller and its shaft are advancing at a speed of 1 m/s, therefore the Froude and Reynolds numbers based on the shaft-propeller common length are $Fr=0.319$ and $Re=4.733 \cdot 10^6$, respectively. The two propeller submergences are $h_1 = 0.305$ m and $h_2 = 0.22875$ m, which means that firstly $h = R$, then $h = R/2$ measured from the tip of the vertical blade placed at 12 o’clock.
4.3.1. Propeller working at $R$ beneath the free surface. All the computations are performed for the above-mentioned constant values of the $Fr$ and $Re$ at the same five advance coefficients tabulated in table2. For any typical free-surface flow computation three are the parameters which are of a special interest. The first one regards the forces and moments developed, the second is concerning the free-surface topology and the generated wave geometries, whereas the third is related to the distribution of the main parameters of the flow in the immediate vicinities of the water surface and the solid body. In the following only the aspects related to the second and third issues will be discussed now. Figure 7(a) depicts a top view of the free-surface, while figure 7(b) shows the longitudinal pressure field downstream of the propeller. As far as the pressure distribution in the propeller wake shown in figure 7(b) is concerned, one may notice that the negative levels of pressure do not exhibit a direct link between the propeller blade and the free surface, a fact that suggests that the ventilation is not expected to occur at this submergence. $\zeta$ in figure 7(a) is the wave elevation, while $p_0$ and $P$ in figure 7(b) are the pressure and the atmospheric pressure, respectively. Worth to mention that the change of the pressure field beneath the water surface leads to an upwards inclination of the generated train of vortices of about $2.8^\circ$.

![Figure 7](image_url)

**Figure 7.** Propeller working at a submergence equal to its radius, $J = 0.9$. (a) wave pattern; (b) longitudinal pressure distribution.

4.3.2. Propeller working at $R/2$ beneath the free surface. Again, all the computations are performed for the same five advance coefficients as in 4.1 and 4.3.2. Basically the discussions made in the subsection above remain valid here as well. However, the decrease of the submergence determines two
different effects. The first one regards the free surface pattern shown in figure 8(a), whose elevation in the wake increases up to the threshold limit of the sub-breaking occurrence. Since the mechanism of the early stage breaking was completely described in [17], no special attention will be paid herein. The second effect regards the probability of the ventilation occurrence. Figure 8(b) shows that the pressure field behind the propeller and around the tip of the uppermost blade shows local drops below the atmospheric value, a fact which may suggest the air entrapment. Because of that, the train of vortices washed in the stream is inclined about 6° upwards of the “0-0” axis because of the pressure defect. As a result of the interference between the free-surface presence and the flow field the intensity of the vortices decreases as well, see also the figure 7(b), as described in detail in [18, 19].

A comparison between the three vortical structures released by the propeller in the present study and depicted in figure 9 shows clearly the influence manifested by the free-surface presence on the intensity of the vortices. Obviously, the water surface influence does not break the periodicity of the vortices, a fact that may suggest that the ventilation does not take place despite the favourable conditions for its occurrence discussed above. Figures 9(b) and (c) show free surface shots taken below the propeller. Even though the mechanism for air entrapping does not reached the critical value, the thrust coefficient drops by about 7% for $h = R$ and by about 16% for $h = R/2$, as figure 10 bears out. The decrease of the torque coefficient is even more significant, going up to 9% for $h = R$ and to about 21% for $h = R/2$.

**Figure 8.** Propeller working at a submergence equal to $R/2, J = 0.9$. (a) wave pattern; (b) longitudinal pressure distribution.
Next, a short analysis of the wave profiles is proposed in figures 11 and 12. Figure 11 depicts a longitudinal cut on the free-surface for the two submergences computation. In both cases the propeller and its shaft extend from $x = -1$ to $x = 0.05$. The proximity of the propeller to the free-surface generates a wave which does not show a Kelvin-like profile. Even though the ventilation did not occur, clear signs of incipient wave breaking are present. This can be clearly seen in figure 12, which depicts two transversal cuts on the free-surface and where the computed waves show steep crests suggesting the breaking imminence. Aside of that, one may notice the lack of the longitudinal symmetry because of the propeller sense of rotation.
5. Conclusions
The hydrodynamic characteristics of a propeller operating beneath a free surface were investigated using numerical simulations. A commercial code based on the unsteady incompressible equations of the flow was applied for these simulations. The numerical method applied to this study was verified through a comparison with the experimental data of POW test performed in DTMB with a large model propeller of a diameter of 0.305 m. The main findings of the present research may be summarized as follows:

- The propeller generated a significant wave, particularly in the first crest area;
- When the submergence depth decreased, the reduction ratios of thrust and torque increased. In addition, the wave amplitude of the free surface decreased at the deeper submergence depth, and the wave pattern like a Kelvin wave could not observed;
- The small submergence computational case clearly reveals the fulfilment of the critical condition for the sub-breaking occurrence;
The reduction ratios of thrust and torque increased with increasing advance coefficient, except for the largest value of $J = 1.1$.

6. References

[1] Califano A. 2010 Dynamics loads on marine propellers due to intermittent ventilation. Ph.D. Thesis, Norwegian University of Science and Technology, Trondheim, Norway.

[2] Koushan K. 2006 Dynamics of ventilated propeller blade loading on thrusters. Proc. of the World Maritime Technology Conference (WMTC), Rome, Italy.

[3] Kozlowska A. M., Dalheim O. O., Savio L. and Steen S. 2020 Time domain modeling of propeller forces due to ventilation in static and dynamic conditions. J. Mar. Sci. and Eng. 8 (31).

[4] Lungu A. 2007 Numerical simulation of the free-surface turbulent flow around a VLCC ship hull. AIP Conference Proceedings 936, pp. 647-650.

[5] Lungu A. 2018 Numerical simulation of the resistance and self-propulsion model tests. Proc. of the ASME 37th Int. Conf. on Ocean, Offshore and Arctic Engineering OMAE2018, Madrid, Spain, Paper OMAE2018-77767.

[6] Lungu A. 2020 Numerical simulation of the resistance and self-propulsion model tests. J. Offshore Mech. Arctic Eng., 142(2): 021905.

[7] Kozlowska A., Savio L. and Steen S. 2017 Predicting thrust loss of ship propellers due to ventilation and out of water effect. J. Ship Res, 61, 198–213.

[8] Lungu, A. and Pacurarui F. 2009 Numerical study of the hull-propeller-rudder interaction. AIP Conference Proceedings, 1168, pp. 693-696.

[9] Pacurarui F. and Lungu, A. 2010 Numerical investigation on ship podded propulsion, Numerical Analysis and Applied Mathematics. 0 AIP Conference Proceedings, 1281, pp. 111-114.

[10] Pacurarui F., Lungu A., Ungureanu C., and Marcu O. 2010 Numerical simulation of the flow around a steerable propulsion unit. Proceedings of the 25th IAHR Symposium on Hydraulic Machinery and Systems, Timisoara, 12, pp.246-252.

[11] Pacurarui F., Lungu A. and Marcu O. 2011 Self-propulsion simulation of a tanker hull. AIP Conference Proceedings, 1389, pp. 191-194.

[12] Wagner H. 1932 Über stoß-und gleitvorgänge an der oberfläche von flüssigkeiten. Z. Angew. Math. Mech. 1932 (12), pp. 192–235.

[13] Lungu A. 2019 Scale effects on a tip rake propeller working in open water. J. Mar. Sci. and Eng. 7 (11), 404.

[14] Menter F.R. 1993 Zonal two-equation k-omega turbulence models for aerodynamic flows. AIAA 24th Fluid Dynamics Conf., AIAA Paper 93-2906.

[15] Jessup S.D. 1998 Experimental data for RANS calculations and comparisons (DTMB P4119). 22nd ITTC propulsion committee propeller RANS/panel method workshop, Grenoble, France.

[16] Jessup S.D. 1999 An experimental investigation of viscous aspects of propeller blade flow, Ph.D. Thesis, The Catholic University of America, Washington DC.

[17] Lungu A., Raad P. E. and Mori K-h. 1997 Turbulent early-stage breaking wave simulation. Proc. of the 1997 ASME Fluids Eng. Div. Summer Meeting, FEDSM97, Vancouver, Canada. Paper FEDSM97-3404.

[18] Lungu A. and Ungureanu C. 2008 Numerical study of a 3-D juncture flow. AIP Conference Proceedings, 1048, pp. 839-842.

[19] Ungureanu C. and Lungu A. 2009 Numerical simulation of the turbulent flow around a strut mounted on a plate. AIP Conference Proceedings, 1168, pp. 689-692.

Acknowledgments

The work reported in here has been entirely conducted by using the research computing facilities HPC-UGAL offered by the “Dunarea de Jos” University of Galați - www.hpc.ugal.ro. A special thanks goes to Adrian Istrate whose unfailing top-level professional effort in providing all the needed technical conditions is greatly acknowledged.