Numerical Assessment of Twin-Propeller Performances

A Lungu

“Dunarea de Jos” University of Galati, 47 Domneasca Street, Galati 800008, Romania

adrian.lungu@ugal.ro

Abstract. The paper describes a methodology for assessing the hydrodynamic performances of a four blade fixed-pitch propeller working in open water. Both single propeller and twin-propeller arrangements are studied. The numerical solver, which is a part of the Numeca FineTM-Marine package is based on the finite volume method. Closure to turbulence is based on the detached eddy simulation (DES). A double sliding grid technique is used to treat the contra-rotational flow around the propellers. A grid convergence test is carried out to verify the robustness of the numerical treatment. Comparisons with the experimental data for the single propeller are performed twice to validate the ISIS-CFD viscous flow solver for both starboard and portside propellers. Discussions based on the numerical solutions are provided aimed at clarifying the particular issues of the hydrodynamic interference between the two propellers. Finally, a series of remarks are concluding the findings of the present work.

1. Introduction

Studying the propulsion performances of a ship has become an important issue for the naval architect, especially after one of the main interest subjects had to be related to the fuel consumption of ships to meet the requirement for minimum energy efficiency level measured by an Energy Efficiency Design Index (EEDI) and Energy Efficiency Operational Indicator (EEOI). These technical measures are regulated by the Marine Environment Protection Committee of the International Maritime Organization to indicate the efficient use of energy [1]. The EEDI for new ships is the most important technical measure aimed at promoting the use of more energy efficient equipment and engines. The EEDI requires a minimum energy efficiency level per capacity mile (e.g., tons mile) for different ship type and size segments. On the other hand, the EEOI is the monitoring tool for managing ship and fleet efficiency performance over time. It therefore enables operators to measure the fuel efficiency of a ship in operation.

Under the pressure exerted by the additional requirements imposed by the above-mentioned decisions, the naval architecture community had to find solutions to overcome the change of the design paradigm. As a direct result, quick concrete solutions that may sometimes conflict with the immediate interests of the ship owners had to be prospected, validated and imposed in the common practice. Consequently, extensive experimental and theoretical studies have been recently carried out aimed at describing in every detail not only the propulsion performances, but also the complexity of the unsteady turbulent flow around a propeller working behind a ship hull in various conditions. Doubtlessly, many of the researches on the propeller performances are still based on the experiment performed either in cavitation tunnels, when cavitation or local flow features are of scientific interest, or in towing tanks, when the open water performances are sought [2-4].
Going back in time, the design of a propeller was almost exclusively based on the use of empirical series of diagrams, which served as the only tool for establishing the geometrical characteristics and some global information concerning the expected hydrodynamic performances. In spite of their serious limitations, the diagrams were used for a rather long period of time, as the only available tool in the naval architecture community. Years later methods based on the Prandtl lifting-line theory, developed by Betz [5], which are mathematical models that predict lift distribution over a three-dimensional wing based on its geometry, were accepted in the common practice. Because not only to their drawbacks, but also to the abstract formulation that made them hard to use, these methods were progressively replaced by the more accurate method of lifting surface. The method is based on the Lerb induction factor approach [6], but because this was a design procedure the Lerb method was used in the inverse sense, which proved to be unstable. To overcome this instability to determine the induced velocities and circulation distribution for a given propeller geometry, van Oossanen introduced an additional iteration for the hydrodynamic pitch angle [7].

With the advent of the computational power, the boundary element methods (BEM) for propeller analysis have been developed to overcome the two major shortcomings of the lifting surface methods. The first is the occurrence of local errors near the blade leading edge and the second is the more widespread errors, which occur near the hub where the blades are closely spaced and relatively thick. Hess and Valarezo introduced a method of analysis based on the earlier work of Hess and Smith in [8]. Later, Hoshino proposed a surface panel method for the hydrodynamic analysis of propellers operating in steady flow [9]. In this method the surfaces of the propeller blades and hub are approximated by a number of small hyperbolic quadrilateral panels with constant source and doublet distributions. The main advantage of the BEM technique resides in their efficiency, which makes them usable even on moderate hardware equipment. However, because of the errors introduced by the ideal fluid supposition, their attractiveness is rather limited to simple problems for which fast solutions are expected.

Obviously, the methods based on the numerical integration of the real fluid equations of motions are nowadays the mostly used. They are based on the numerical solutions of the Navier-Stokes and continuity equations, which are solved in different manners, depending on the way the closure to turbulence is done. Reynolds averaging still remains the most used technique to fit the purpose [10, 11]. It is reasonably efficient and the overall accuracy is sufficiently good not only for commercial applications, but also for research purposes. Although several turbulence models were made available, out of which the two-equation ones are the most popular, those based on the $k - \omega$ model of Menter [12] seem to be the most preferred ones.

A special class of numerical solutions is based on the large eddy simulations (LES) [13-15] in spite of its higher associated computational cost. Because of the high resolution of the solution, LES-based methods are very suited to local flow studies. Given the actual hardware development, they remain indicated for research purposes mostly. A way to cope with the high cost problem seems to be the use of the so-called hybrid RANS-LES methods such as the detached eddy simulation (DES) [16-18] and delayed detached eddy simulations (DDES) [19]. Both methods are conceived to combine locally the advantages given by efficiency of the RANS $k - \omega$ SST and the accuracy of the LES approaches. Recently another novel class of more efficient methods is proposed to the community, based on its lower computational costs. That is, a coupled RANS-potential technique [20, 21] is gaining in popularity because of the promising advantages which are offered to the users mostly in terms of the reduced computational costs for an acceptable good accuracy of the numerical solution. Since the communication between the panels of the BEM potential method and the grid cells of the viscous solver is achieved through interpolation, the overall accuracy might be affected and additional validations may be necessary.

The present study embodies some of the achievements of the references above-mentioned. As a continuation of some previous investigations performed by the author on the same propeller geometry working in oblique flow [22], it proposes a study based on the DES-SST method to study the hydrodynamic performances of a twin propeller working in an open water condition. The study is based on a DES SST approach, in which the flow features are computed by using the sliding grid based ISIS.
unsteady finite volume solver implemented in the Numeca Fine\textsuperscript{TM}/Marine package. A series of computations are performed to check out the grid sensitivity, the overall accuracy of the solver as well as its ability to capture the detailed features of the wake field developed behind the propellers.

2. Mathematical model, numerical milestones and computational strategy
Equations which describe the flow are written in the tensor form as follows:

\[ \frac{\partial (u_i)}{\partial t} + \frac{\partial}{\partial x_j}(u_i u_j) = \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial u_i}{\partial x_j} - \overline{u_i' u_j'}\right) + S_j \]

where \( u_i \) and \( u_j \) denote the time-averaged values of velocity components \((i, j = 1, 2, 3)\), \( p \) denotes time-averaged pressure, \( \mu \) denotes the dynamic viscosity coefficient, \( \rho u_i' u_j' \) denotes the Reynolds stress term, and \( S_j \) denotes the generalized source term of the momentum equation. The hybrid DES model corresponds to a mixed numerical model that combines the strengths of both RANS and LES methods. The DES approach is based on an implicit splitting of the computational domain into two zones. In the first 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. In all DES simulations, the \( k-\omega \) SST two-equation turbulence model \([12]\) is employed to resolve turbulence structures in the propeller wake. Given the advantage of the relatively low computational effort, the \( k-\omega \) SST model proved to be sufficiently robust to capture the details of complex flows, \([18, 22]\). In the SST-DES hybrid model the dissipation term in the \( k \) transport equation reads as follows:

\[ \rho \varepsilon = \beta^* \rho k \omega F_{DES} \]

where:

\[ F_{DES} = \max\left(\frac{L_t}{C_{DES}} (1 - F_{SST}), 1\right) \]

and

\[ F_{SST} = 0, F_1, \text{ or } F_2 \]

\( \Delta = \max(\Delta x, \Delta y, \Delta z) \) is the maximum local grid spacing, \( L_t = \sqrt{k}/(\beta^* \omega) \) is the length scale, and \( C_{DES} = 0.78 \) is a constant of the model, whose value was proposed by Menter \([12]\). \( F_{SST} = 0 \) recovers the Strelets model, whereas \( F_1 \) and \( F_2 \) are the two blending functions of the \( k-\omega \) SST model, which were set so that be equal to unity in the near wall region and zero away from the surface. Function \( F_1 \) extends in the wake region of the boundary layer while the function \( F_{12} \) extends further out into the boundary layer than \( F_1 \). The function \( F_1 \) is defined as \( F_1 = \tanh(\arg 1^4) \), with

\[ \arg 1 = \min\left(\max\left(\frac{\sqrt{k}}{0.09 \omega y}, \frac{500 v}{y^2 \omega} \right); \frac{4 \rho \sigma_{\omega_2} k}{C D k \omega y^2}\right) \]

\( \sigma_{\omega_2} = 0.856, y \) is the distance to the next surface and \( CD_{k\omega} \) is the cross-diffusion of the \( \omega \) equation defined as:

\[ CD_{k\omega} = \max\left(2 \rho \sigma_{\omega_2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}; 10^{-20}\right) \]

The \( F_2 \) function is defined as \( F_2 = \tanh(\arg 2^2) \), where

\[ \arg 2 = \max\left(2 \frac{\sqrt{k}}{0.09 \omega y}, \frac{500 v}{y^2 \omega} \right) \]
The ISIS-CFD finite volume solver, which is a part of Numeca FineTM-Marine suite is employed to solve the 3D unsteady flow problem. The forces integration is done on the solid-surface 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. Three sets of simulations are performed in the present study. Two sets are devoted to the investigation of the propeller open water (POW hereafter) performances, one for each propeller. Numerical solutions in this care are computed by using the classic rotating frame technique. Since the approach is widely known, no more details will be provided in the followings. On the opposite, when the twin counter-rotating propellers are studied, the sliding grid method is employed.

The two four-bladed propellers with an inward rotation direction, of a diameter $D=0.25$ m, are those mounted on the ONR tumblehome ship used as a benchmark case in the 2015 Workshop on CFD in Ship Hydrodynamics [23]. Propeller geometry as well as the experimental data for the propellers working in open water (POW) were provided in [23]. The rectangular computational domain for the twin case shown in Figure 1 extends three propeller diameters at the upstream, five diameters on lateral and vertical directions and $7$ diameters downstream of the propeller. The clearance between the tips of the two propellers is $1.6$ $D$. When the single propeller is computed, the rotating computational domain is of a cylindrical geometry with a $5$ diameter radius, which extends $3$ diameters at the upstream and $7$ diameters at the downstream.

The boundary conditions for the twin-propeller case are indicated in Figure 1. When the single propeller is computed, the far field condition is used for the inlet and external boundaries, whereas the frozen pressure is imposed at the outlet. In both cases the no-slip condition is imposed on the solid surfaces of the propellers. The time step $\Delta t = 10^{-3}$ seconds is chosen such as the Courant number be finally less than unit. The flow is accelerated from the rest condition to the rotational speed on a half sinusoidal ramp within a couple of seconds regardless of the advance coefficient value. All computations are performed until the thrust and torque stabilize. A number of ten iterations per time steps are enforced for this first run. Once the forces stabilize, the computation is stopped, the time step is decreased by an order, the turbulence model is switched to the DES SST, the number of iterations per time step is doubled and the computation is restarted.

![Computational domain and boundary conditions](image)

**Figure 1.** Computational domain and boundary conditions

### 3. Computational grids

A non-structured body and boundary fitted sliding grid with hexahedral elements generated in Hexexpress, which is another component of the Numeca FineTM-Marine suite, is used for the discretization. Since the computations deal with a counter-rotating propeller case, the sliding grid technique is employed in the study reported herein. The generation process is conducted such that the grid cells be clustered around the blades edges and tips, as depicted in Figure 2. Moreover, another rather heavy clustering area is placed in the propeller wake, aimed at a good resolving of the vortices released by the blades.
Figure 2. Computational mesh. (a) grid around a single propeller; (b): sliding grids; (c): isometric view of the twin propellers grids.

Viscous boundary layers are placed in near the proximity of each solid surface in a sufficient number such that the flow can be properly resolved. Four different grids were generated for the grid sensitivity computations. Let those meshes be denoted by G1…G4, G1 being the coarsest and G4 the finest. The cells numbers vary from $4.6 \times 10^6$ to $18.4 \times 10^6$, with the recommended augmentation ratio of $\sqrt{2}$. During the grid convergence test, the adaptive mesh refinement is not activated. On the contrary, when the solution is computed for the flow analysis, mesh adaption based on the hessian of the pressure is used. The criterion is tensorial, being based on the Hessian tensor of second spatial derivatives of the pressure, to the power 0.5. During the mesh adaption buffer layers are not necessary but a minimum cell size, but a threshold was necessary to be set so that a continuous close control over the maximum number of cells can be kept.

4. Results and discussions

4.1. Single propeller computations

4.1.1. Grid convergence test. The first set of computations in performed only for the starboard propeller for all four meshes and four advance ratio coefficients for verification and validation purposes. The numerical scheme is that described above and the experimental data used for validation are those provided in [23]. The absolute errors of the numerical solutions are tabulated in Tables 1 and 2 for the four grids considered in the convergence test. The figures contained in the two tables reveal a maximum level of error of 3.18% for thrust and of 4.38% for torque which may be considered as acceptable. They all prove the monotonic dependence of the solution on the mesh refinement, thus confirming the stability of the numerical scheme chosen to solve the problem. Obviously, the best accuracy corresponds to the finest grid, therefore it will be further considered for the next numerical computations.

4.1.2. POW performances. The next set of POW computations will be performed for the portside propeller. The numerical solutions are again compared to the experimental data provided in [23] and the conclusions are similar to the starboard propeller case. Figure 3 shows the comparisons with the experiments for both propeller. Figure 3(a) depicts the portside propeller, whereas Figure 3(b) the
The numerical solutions are drawn by lines, while the experimental data are figured by symbols. The computations had to be performed for a series of 15 advance coefficients in each case since the maximum efficiency is found around $J = 1.3$, as Figure 3 bears out. An important issue of any POW simulation is the accurate description of the wake field behind the rotating propeller. Its importance is due to the periodicity of velocity and pressure fields, which manifests through the vortices released by the propeller, which may eventually affect the performances of the appendages placed in the propeller stream [11], [24]. In these circumstances, a detailed analysis of the velocity, vorticity and turbulent kinetic energy is further proposed in Figure 4.

Table 1. Grid sensitivity computations.

| $J$ | $|\varepsilon| K_r$ [%] | $|\varepsilon| K_Q$ [%] | $|\varepsilon| \eta$ [%] |
|-----|------------------------|------------------------|------------------------|
| 0.1 | 3.06                   | 0.11                   | 2.16                   |
| 0.2 | 3.18                   | 0.93                   | 2.65                   |
| 0.3 | 1.66                   | 2.47                   | 2.74                   |
| 0.4 | 1.05                   | 4.38                   | 2.79                   |
| 0.5 | 0.95                   | 3.76                   | 2.37                   |
| 0.6 | 0.80                   | 3.28                   | 2.55                   |
| 0.7 | 0.73                   | 3.18                   | 2.64                   |
| 0.8 | 0.64                   | 2.60                   | 2.98                   |
| 0.9 | 0.51                   | 2.12                   | 2.34                   |
| 1.0 | 0.48                   | 1.99                   | 2.11                   |
| 1.1 | 0.26                   | 1.815                  | 2.06                   |

Table 2. Grid convergence average absolute errors.

| Grid | $|\varepsilon| K_r$ [%] | $|\varepsilon| K_Q$ [%] | $|\varepsilon| \eta$ [%] |
|------|------------------------|------------------------|------------------------|
| G1   | 1.21                   | 2.42                   | 2.49                   |
| G2   | 1.16                   | 2.36                   | 2.23                   |
| G3   | 0.94                   | 2.17                   | 2.12                   |
| G4   | 0.65                   | 1.91                   | 1.01                   |
| Average | 0.99                   | 2.215                  | 1.985                  |

Figure 3. Propellers diagrams of performance. (a): portside propeller; (b): starboard propeller.

The figure depicts the numerical solution computed on the starboard propeller at an advance ratio of 1.2, which corresponds to the highest efficiency. Figs. 4(a) and 4(b) depict the velocity and vorticity distribution on the vertical plane of symmetry, while Figure 4(c) shows the turbulent kinetic energy in the same location. They all bear out the computed wake pattern for the considered advance ratio. Analyzing the figures one may see that three systems of vortices are cohabitating behind the propeller. The most significant one is generated by the tips of the blades. This is a rather stable vortex, of a constant
strength and periodicity, which propagates far in the downstream till the viscous dissipation will favor its pairing.

The second vortex is released by the propeller hub. It is a strong swirl vortex, which does not show any sign of oscillation. Seemingly this is due to the high value of the streamwise velocity, which may be responsible for its fast transport towards the downstream. Systematic computations have proven that this vortex extends far in the stream till its energy dissipates and eventually get paired with the tip vortices. The third vortical system is represented by the blade passage generated vortices, which manifest almost like a cloud. These vortices of a lower intensity are more prominent in the near wake, then they die away because of the viscous dissipation, as Figs. 3(b) and 3(c) clearly show. The mechanism of vortex formation, evolution and destruction can be easier suggested if the vertical component of vorticity is considered.

Vortices shown in Figure 4(b) are computed as iso-surfaces of $Q = 10000$. Here the $Q$ invariant is defined as $Q = 0.5(\|\Omega\|^2 - \|S\|^2)$, where $\|\Omega\| = \text{tr}[\Omega\Omega^T]^{1/2}$ and $\|S\| = \text{tr}[SS^T]^{1/2}$. Here $S_{ij} = 0.5(u_{ij} + u_{ji})$ is the strain-rate tensor, $\Omega_{ij} = 0.5(u_{ij} - u_{ji})$ is the spin tensor, $u_{ij} = \partial u_i / \partial x_j$ and $u_{ij}$ are the three components of a vector field. When the $Q$ Invariant of a region is positive and at the same time the local pressure has a lower value than the ambient, it can be viewed as a vortex core.

![Image](a)

(a)

![Image](b)

(b)

![Image](c)

(c)

**Figure 4.** Starboard propeller wake. (a): non-dimensional axial velocity; (b): vorticity; (c): turbulent kinetic energy.

Figure 5 proposes a diagram which depicts the evolution in the horizontal plane of symmetry of the vertical component of vorticity. For the sake of clarity the figure is drawn at an instant. Two distinctive lines, almost parallel to the flow directions, signifying the loci of the cores are drawn on the diagram. Besides, there is the intermediate region above-mentioned, where the blade passage vortex cloud is acting. Previous extensive studies proved that the tip vortex breaks down at a certain distance from the propeller, manifesting itself as a spiral with an irregular shape [25, 26]. The location of transition to instability strongly depends on spiral-to-spiral distance as the number of blades affects the onset of tip vortex instability location through influencing the spiral-to-spiral distance. The perturbation resulting from the destabilization of tip vortices can cause the hub vortex to become unstable as well. The breakdown of tip vortices occurs prior to that of hub vortices and thus the hub vortices proved to be independent of the blade number and the advance coefficient. This fact may also explain why the swirl vortex does not show any periodicity. Next, an analysis of the longitudinal distribution of the streamwise velocity and of the vorticity magnitudes is proposed in Figure 6. The longitudinal distribution of the
velocity and velocities profiles is considered for a distance of $0.8D$ measured from the propeller origin towards the downstream. At a first glance, it is worth mentioning that both axial velocity and vorticity profiles exhibit a fully symmetric shape in respect to the axis of symmetry, as it was also shown in Figure 4. Aside of that, several other particular flow features are revealed. Firstly, it is obvious that within less than a diameter, both axial velocity and vorticity show a significant decay, which is caused by the inherent viscous dissipation.

![Image](image.jpg)

**Figure 5.** Spatial evolution of the vorticity in the wake.

In spite of this decay, the symmetry is not lost, a fact which may prove the robustness of the numerical method chosen to solve the flow problem. Secondly, Figure 6(a) shows that regardless of the distance from the propeller, streamwise velocity drops are located on the hub vortex line and slightly below the tips vortex line, i.e. at about $0.45D$. Seemingly, this is due to the flow contraction shown in Figure 5. Lately, the figure emphasizes that the velocity and vorticity become almost constant around $z/C = 1$, which suggests that the boundary condition imposed on the outer cylinder worked properly. Exactly the same conclusions may be withdrawn from Figure 6(b) which bears out the vorticity profiles drawn at the same seven cross sections as for the streamwise velocity, except for the fact that the velocity drops shown in Figure 6(a), correspond here to peaks of vorticity. This behaviour is somehow conflicting to that reported in [27], where the authors concluded that velocity profiles become flat as the advance coefficient increases showing a better propulsion performance for higher $J$ cases. The reason for that contradiction may be given by the manner in which the turbulence is treated in the two studies. The authors of [27] used a RANSE-based method employing the $k-\omega$ SST turbulence model, whose capacity in resolving the small scale turbulence is limited, whereas the hybrid DES SST model is used in here.

**4.2. Twin propeller computations**

**4.2.1. Twin propellers acting independently.** Next set of computations is performed for the twin propellers working in tandem, but independently. This means that in the longitudinal plane of symmetry is placed a solid plate of a zero-thickness, which is somehow similar with the skeg mounted on the ship hull to separate the two working propellers. The solid plate extends three propeller diameters in the downstream. Except for the sliding grids used now, the computational conditions are exactly the same as those for the $J = 1.2$ case of the single propeller discussed above. In terms of the boundary conditions formulation, the wall function condition is imposed on the central plate the far field is imposed on all the other boundaries excepting the outlet where the frozen pressure is imposed. The numerical strategy is the same as for the single propeller, i.e. the flow is accelerated for 2 seconds and then the computation is continued for 28 more seconds so that the forces stabilize. Afterwards, the turbulence model is switched to the DES SST and the computation is continued for five seconds, which means 50 more complete rotations.

Figure 7 depicts the two pairs of four counter-rotating vortices released by the tips of the blades. They are coloured by helicity, so the inwards rotation can be easily remarked. The existence of the
separating diaphragm in the vertical plane of symmetry prevent their mutual interaction. The lateral components of velocity are suppressed by the separation wall, therefore no contamination of the vortices is detected. However, immediately behind the separator a lack of periodicity can be seen, a fact which is attributable to the sudden interaction which takes place there.

![Figure 6. Spatial evolution of the axial velocity and vorticity magnitude in the wake. (a): axial velocity; (b): vorticity.](image)

4.2.2. Twin propellers working in tandem. The next computation considers the two propeller working in conjunction with each other, which means that the zero-thickness separator previously used is simply removed now. All the other flow conditions described at §4.2.1 are kept unchanged. The scope of the present computation is to clarify how the mutual interaction between the two propellers affect not only the propulsive performance, but also the flow kinematics in the wake. Figure 8 bears out the vortices released by the propellers. For the sake of similarity with Figure 7, the scale of representation is the same. The vortical iso-surfaces are colored in terms of axial velocity.

![Figure 7. Vortical structures released by the propellers working independently coloured by helicity.](image)

Comparing the two figures, one may figure out that when the propeller can interact, a secondary vortical system is separating from the primary one at about a diameter behind the propeller center, immediately after the contraction zone. The secondary vortex of a lower strength propagates in the downstream with the same periodicity, but it vanishes more rapidly than the primary one. Its diameter is almost constant, as it is not the case with the primary one, which has a diverging pattern due to the
influence exerted by the other propeller. Pilot computations have proven that responsible for that is the lateral component of velocity, which is not suppressed any longer. Figure 8 proves also that the streamwise velocity does not seem to be affected, which is important for the overall propulsive efficiency.

Figs. 9(a) and 9(b), which show the spatial evolution of the non-dimensional axial velocity sustain this statement. The velocity profiles are drawn at the same locations as those used in Figure 6 for the single propeller. Comparing Figure 9 with Figure 6(a) it may be seen again that the tandem regime does not affect the axial velocity.

![Figure 9. Spatial evolution of the non-dimensional axial velocity in the wake. (a): starboard propeller; (b): portside propeller.](image)

5. Concluding remarks

The present paper describes a numerical investigation of the hydrodynamic performances of a propeller working either alone or in a twin arrangement. The numerical solution is obtained by using the ISIS-CFD solvers, which is based on the finite volume method. Closure to turbulence is achieved through the hybrid SST detached eddy simulation. After the grid convergence test, a first set of complete computations was performed for validation purposes. Both starboard and portside propellers were considered. The numerical solutions of the flow around the propellers were compared with the corresponding experimental data provided in [23]. The good agreement with the open water data confirmed the accuracy of the numerical method based on the rotating frame technique.
Next set of computations referred to the twin propeller case for which the sliding grid technique is employed. Both cases of propellers acting independently and in tandem were considered. Analyses concerning the flow kinematics were performed aimed at clarifying the details of the mutual interactions between the two propellers with an inward direction of rotation. With this bird’s eye view at hand, the following conclusions may be withdrawn:

- The rotating frame method proved to be well suited for the single propeller computations;
- A proper grid clustering in the annular region behind the propeller, doubled by an automatic mesh refinement, could successfully allow the capturing of the tip vortices even far in the downstream;
- When the twin propellers are working independently, the wake flow field remains unaffected;
- When the twin propellers are working in tandem, the vortices released by the tips split, thus contributing to the generation of a secondary vortex system of a weaker intensity. Additionally, the vorticity field loses the symmetry shown by the velocity field.

6. Acknowledgements
The computations were performed on the HPC at the “Dunarea de Jos” University of Galati. A special thank goes to A. Istrate for providing optimal access to the computational resources.

7. References
[1] MARPOL, 2012, MEPC 62/24/Add.1, Annex 19
[2] Felli, M., Di Felice, F., Guj G. and R. Camussi, R., 2006, *Exp. Fluids*, 41, pp. 441-451
[3] Paik, B.G., Kim, J., Park, Y. H., Kim, K.S. and Yu, K.K., 2007, *Ocean Eng.*, 34, pp. 594-604
[4] A.M. Kozlowska A. M. and Steen, S., 2017, *Appl. Ocean Res.*, 67, pp. 201-212
[5] Betz, A., 1927, Vier Abhandlungen zur Hydrodynamik und Aerodynamik, pp. 68–92
[6] Lerbs, H. W., 1952, *SNAME Trans.*, 60, pp. 73–123
[7] van Oossanen, P., 1974, Publication 457 Netherlands Ship Model Basin Wageningen
[8] Hess, J. L. and Valarezo, W. O., 1985, *J. Propulsion and Power*, 1(6), pp. 470-476
[9] Hoshino, T., 1989, *J. of Soc. of Naval Arch. of Japan*, 166, pp. 79-92
[10] Gaggero, S., Dubbioso, G., Villa, D., Muscari, R. and Viviani, M., 2019, *Ocean Eng.*, 178, pp. 283-305
[11] Lungu, A., 2020, *J. Offshore Mech. Arct. Eng.*, 142(2): 021905, doi: https://doi.org/10.1115/1.4045332
[12] Menter, F. R., 1994, *AIAA J.*, 32, pp. 1598-1605
[13] Jang, H. and Mahesh, K., 2013, *J. Fluid Mech.*, 729, pp. 151-179
[14] Kumar, P. and Mahesh, K., 2007, *J. Fluid Mech.*, 814, pp. 361-396
[15] Posa, A., Broglio, R., Felli, M., Falchi, M., Balaras, E., 2019, *Comput. Fluids*, 184, pp. 138-152
[16] Spalart, P. R., 2009, *Ann. Rev. Fluid Mech.*, 41, pp. 181-202
[17] Lungu, A., 2019, *J. Mar. Sci. Eng.*, 7(11), 404, https://doi.org/10.3390/jmse7110404
[18] Lungu, A., 2020, *J. Mar. Sci. Eng.*, 8(4), 297, (2020), https://doi.org/10.3390/jmse8042097
[19] Chase, N. and Carrica, P.M., 2013, *Ocean Eng.*, 60, pp. 68-80
[20] Martin, J. E., Michael T. and Carrica P.M., 2015, *J. Ship Res.*, 59, pp. 31-48
[21] Gaggero, S., Villa, D. and Viviani, M., 2017, *Appl. Ocean Res.*, 66, pp. 55-78
[22] Lungu, A., 2020, IOP Conference Series: Materials Science and Engineering, 916, 012055, https://doi.org/10.1088/1757-899X/916/1/012055
[23] ***, 2015, Workshop on CFD in Ship Hydrodynamics, https://t2015.nmri.go.jp/onrt_gc.html
[24] Pacuraruru, F., Lungu, A. and Marcu O., 2011, AIP Conference Proceedings 1389, 191, https://doi.org/10.1063/1.3636699
[25] Felli, M., Camussi, R. and Di Felice, F., 2011, *J. Fluid Mech.*, 682, 5-53
[26] Pecoraro, A., Di Felice, F., Felli, M., Salvatore F. and Viviani, M., 2015, *Ocean Eng.*, 108, 181-190
[27] Heydari M. and Sadat-Hosseini, H., 2020, *Ocean Eng.*, 204, 107247, (2020)