International Journal of Instruction, Technology, and Social Sciences
Vol. 5, 106-115, 2026
ISSN: 2716-6546
www.ijitsc.net

SIMPLIFIED METHODOLOGY FOR CFD ANALYSIS IN NAVAL ARCHITECTURE. APPLICATIONS TO SHIP RESISTANCE AND HYDRODYNAMIC PERFORMANCE

Constantin Petre DIȚU
Mircea cel Batran Naval Academy of Constanta
e-mail: constantinpetre.ditu@gmail.com

Anca Florentina ANDRON
Mircea cel Batran Naval Academy of Constanta
email: andron_anca@yahoo.com

Ioan ȚUȚUIANU
Mircea cel Batran Naval Academy of Constanta
email: kawaitutu2@gmail.com

Ionuț Cristian SCURTU
Mircea cel Batran Naval Academy of Constanta & University POLITEHNICA of Bucharest, Bucharest, Romania
e-mail: scurtucristian@yahoo.com

ABSTRACT: Computational Fluid Dynamics (CFD) is one of the most widely used numerical methods for analyzing and optimizing the hydrodynamic performance of ships as early as the preliminary design stages. Computational Fluid Dynamics (CFD) simulations represent an effective tool for investigating complex flow phenomena occurring around ship hulls, propulsion systems, and additional hydrodynamic components. By applying numerical methods, CFD analysis can significantly reduce the time and financial resources required for experimental investigations performed on physical ship models. The present study proposes a simplified computational fluid dynamics framework intended for applications in naval engineering. The methodology is designed to investigate ship resistance behavior and to assess the performance of a marine propeller during open-water operation. The numerical workflow involves several essential stages, starting with the preparation of the 3D model and the establishment of the simulation environment, followed by the definition of the mesh strategy, selection of the governing equations and numerical solvers, and final evaluation of the computed results using dedicated post-processing methods.

KEYWORDS
CFD, RANSE, marine propeller, open water, ship resistance, trim, sinkage, turbulence model, unstructured mesh.

1.INTRODUCTION

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and computational algorithms to solve the equations describing fluid flow. By discretizing the computational domain and solving the fundamental equations of mass, momentum and energy conservation, CFD enables the simulation of hydrodynamic and aerodynamic phenomena with a high degree of accuracy, providing information on pressure distribution, velocity fields, turbulence and heat transfer.

In naval architecture, CFD is an essential tool for analyzing and optimizing the hydrodynamic performance of ships. Computational simulations play a key role in the hydrodynamic assessment of marine vessels by enabling the prediction of resistance, examination of free-surface flow characteristics, evaluation of propeller performance, investigation of cavitation effects, optimization of hull geometry, and analysis of the complex interactions between the vessel and the surrounding fluid.

The experimental determination of these quantities, through scale tests in the towing tank, remains the reference method, but it involves high costs and a lengthy preparation time, which is why CFD type numerical methods (Computational Fluid Dynamics) have become, over the last two decades, a standard tool in the preliminary design stage. Compared with experimental methods, these simulations allow many design variants to be evaluated in a shorter time and at significantly lower cost, making them an efficient tool in the preliminary design and optimization stages.

Modern CFD methods rely on solving the Navier-Stokes equations through numerical methods such as the Finite Volume Method (FVM), the Finite Difference Method (FDM) or the Finite Element Method (FEM). In naval applications, the most widespread approach is based on the Reynolds-Averaged Navier-Stokes (RANS) equations, used together with suitable turbulence models to describe the complex flows around the ship’s hull and propulsor.

Within the design process, the results obtained from CFD simulations contribute to evaluating the ship’s hydrodynamic behavior and to substantiating design decisions. Analysis of the pressure distribution, velocity fields, streamlines and the propulsor’s operating characteristics makes it possible to identify areas of energy loss and to optimize the geometric configuration. As a result, the application of CFD contributes to enhanced vessel performance by improving hydrodynamic efficiency, lowering fuel consumption, and increasing overall energy efficiency. In addition, it minimizes the reliance on extensive experimental testing during the early phases of the ship design process, leading to significant reductions in development time and cost.

The purpose of this paper is to present a simplified CFD methodology, applied to two case studies: the open-water analysis of a four-blade propeller and the ship resistance analysis of a container ship hull. The two applications address the main components of a ship’s hydrodynamic performance, propulsion and ship resistance, and form the basis for future self-propulsion studies, in which the propeller operates in the non-uniform wake generated by the hull.

2.SIMPLIFIED CFD ANALYSIS METHODOLOGY

The first stage of the CFD analysis consists of defining the geometric model to be numerically investigated. Two distinct configurations were used in the studies analyzed: a ship model for determining ship resistance, and a marine propeller analyzed under open-water conditions.

International Journal of Instruction, Technology, and Social Sciences
Vol. 5, 106-115, 2026
ISSN: 2716-6546
www.ijitsc.net

The geometry of each model was imported into the FineMarine™, simulation environment, where the solid surfaces and the fluid domain needed for the numerical calculations were defined. For the ship resistance analysis, the ship's characteristics were used across several flow domains corresponding to different Froude number values, while for the propeller study, different advance coefficient values were analyzed to determine the hydrodynamic characteristics.

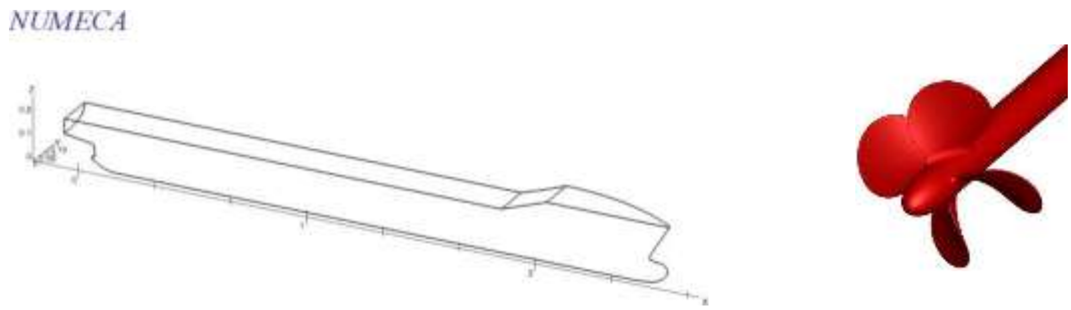

Figure 1. Ship and propeller models

2.1. COMPUTATIONAL DOMAIN

After defining the geometry, the computational domain in which the fluid flow develops was constructed. It was sized large enough that its boundaries would not influence the solution in the vicinity of the body under analysis.

For a propeller, the domain is typically a cylinder surrounding the propeller, hub and shaft, with a diameter of about 5 propeller diameters and a length of about 10 diameters, the propeller being positioned so that a flow development space remains upstream, and a space sufficient for the convection of blade tip vortices remains downstream.

For a hull, the domain extends 3 lengths between perpendiculars (L_{pp}) upstream and downstream, 2 L_{pp} laterally and 1.5 L_{pp} vertically for the air domain, with half of the ship modeled through longitudinal symmetry.

The numerical domain is defined to encompass both the water volume and the free-surface region, enabling the accurate representation of the fluid–structure interaction around the vessel. This configuration also allows the prediction of wave generation and propagation associated with ship resistance calculations.

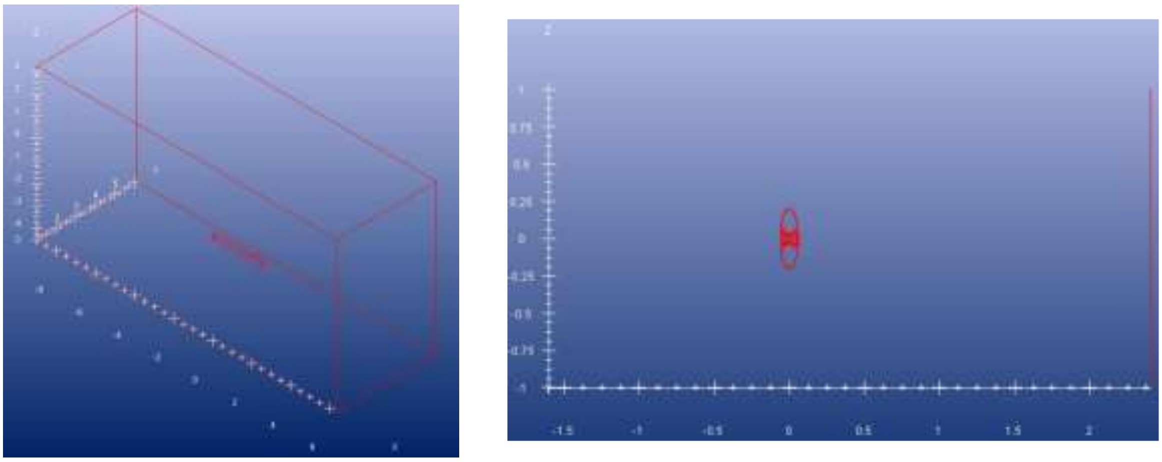

Figure 2. Positioning of the hull and propeller in the computational domain and the boundary conditions

2.2.MESH GENERATION

The mesh is generated using an unstructured hexahedral discretization program, which allows direct import of the CAD geometry and local refinement in the critical areas: the boundary layer on the wetted surface, the propeller disk, and the bow and stern wave regions.

The resolution of the computational mesh has a significant influence on the accuracy of the numerical results. To quantify discretization errors, it is generally recommended to perform a grid convergence study using at least two or three progressively refined meshes.

However, the achievable mesh resolution is often constrained by the available computational resources, which limit the total number of cells that can be employed in the simulation.

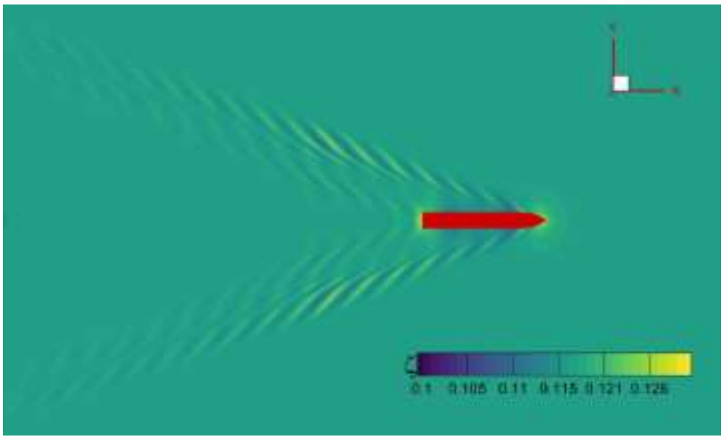

Figure 3. Example of unstructured hexahedral discretization of the domain around the propeller

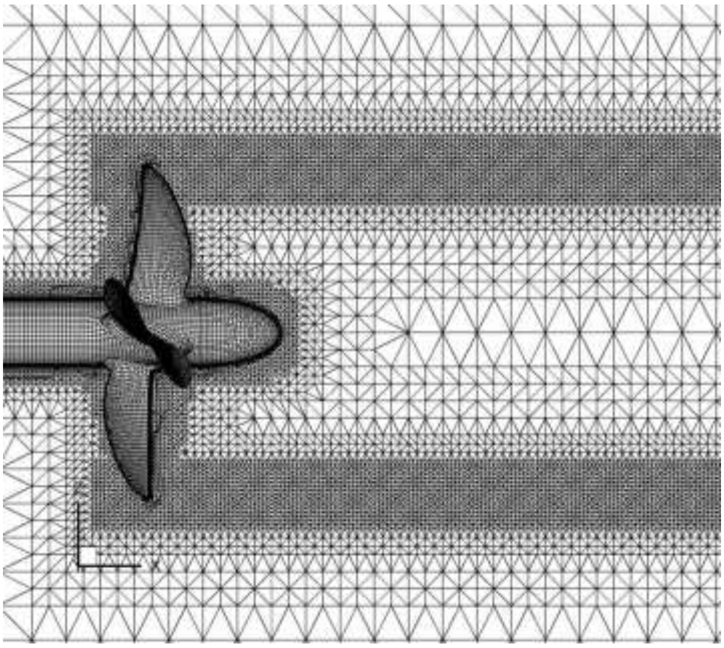

Figure 4. Example of unstructured hexahedral discretization of the domain around the hull

2.3.NUMERICAL MODEL AND TURBULENCE CLOSURE

The equations describing fluid flow are solved numerically using the finite volume method. The SIMPLE algorithm is used for velocity pressure coupling, and turbulence is modeled according to the type of flow analyzed. For flows around the hull, the **k-ω SST** model is used, owing to its accuracy near walls and in regions where the flow may be separated. For analyzing the flow around the propeller, where more complex rotational and turbulence effects occur, the **EASM** model can be used, which offers a more faithful description of these phenomena.

In simulations that include the free water surface, the water-air interface is tracked using the **VOF (Volume of Fluid)** method, which allows simulation of the waves generated by the ship's motion.

2.4.POST-PROCESSING AND INTERPRETATION OF RESULTS

The raw simulation results velocity, pressure and vorticity fields are exported and analyzed with a dedicated visualization program, from which the global quantities of interest are extracted: the forces and moments on the propeller or hull, the dynamic equilibrium position (trim, sinkage), and the profile of the generated waves.

These parameters are expressed in dimensionless form using appropriate coefficients, enabling the comparison of CFD predictions with experimental measurements as well as with results obtained for different geometric configurations.

International Journal of Instruction, Technology, and Social Sciences
Vol. 5, 106-115, 2026
ISSN: 2716-6546
www.ijitsc.net

3.APPLICATIONS

3.1. HYDRODYNAMIC PERFORMANCE OF A PROPELLER IN OPEN WATER

The first application concerns a four-blade marine propeller, with fixed pitch and a cylindrical hub, with a diameter of 0.4 m, operating at a rotational speed of 450 rpm. The simulation was carried out for three values of the advance coefficient J (1.0; 1.1; 1.2), corresponding to advance speeds of 3.0; 3.3 and 3.6 m/s respectively, in fresh water, at a temperature of 15 °C.

The computational domain was discretized with a hexahedral mesh of approximately 2.37 million cells, and turbulence closure was achieved using the EASM model.

The propeller performance was characterized by the thrust coefficient K_T, the torque coefficient K_Q and the open-water efficiency η_0, defined with respect to the rotational speed n, diameter D and fluid density ρ.

The numerical results, compared with experimental data obtained on the same propeller model, are summarized in Table 1.

J	K_T (CFD)	K_T (exp.)	Error K_T (%)	$10 \cdot K_Q$ (CFD)	$10 \cdot K_Q$ (exp.)	Error K_Q (%)	η_0 (CFD)	η_0 (exp.)
1,0	0,2837	0,253	12,1	0,839	0,719	16,6	0,538	0,560
1,1	0,2406	0,210	14,6	0,746	0,633	17,8	0,565	0,581
1,2	0,1976	0,165	19,6	0,652	0,539	20,9	0,579	0,585

Table 1. Comparison of numerical results with experimental data, for the three advance coefficient values

It can be observed that, in both the numerical and the experimental data, K_T and K_Q decrease as J increases, while the efficiency η_0 increases a physically expected trend: as the advance speed increases relative to the peripheral blade speed, the effective angle of attack decreases, and the propeller produces less thrust, but more efficiently.

The numerical values are systematically overestimated compared to the experimental ones, and the error increases with J, reaching nearly 20% for thrust and torque at J=1.2. By contrast, the error in efficiency remains below 4% in all cases, since the efficiency depends on the ratio K_T/K_Q, and the overestimation errors of the two coefficients partly cancel out. This observation is methodologically important: a simplified CFD methodology, even if it does not reproduce the absolute force values with high accuracy, can still provide a reliable estimate of efficiency, a quantity essential for the preliminary sizing of the propulsion system.

International Journal of Instruction, Technology, and Social Sciences
Vol. 5, 106-115, 2026
ISSN: 2716-6546
www.ijitsc.net

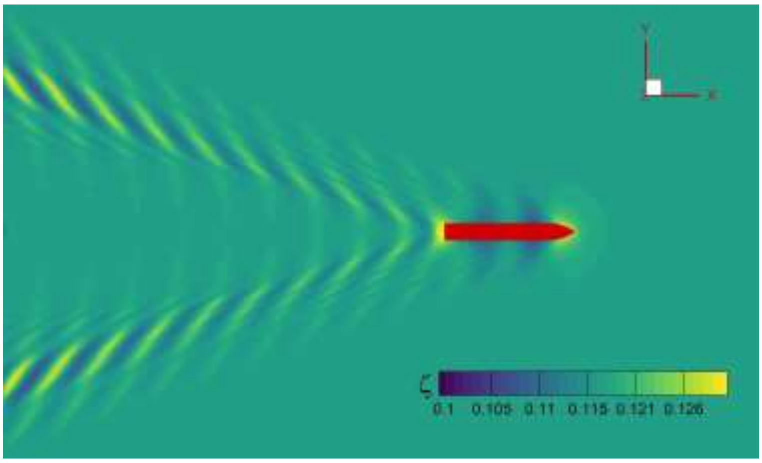

Figure 5. Coefficients K_T, $10 \cdot K_Q$ and the efficiency η_0 obtained from the simulation, as a function of the advance coefficient J

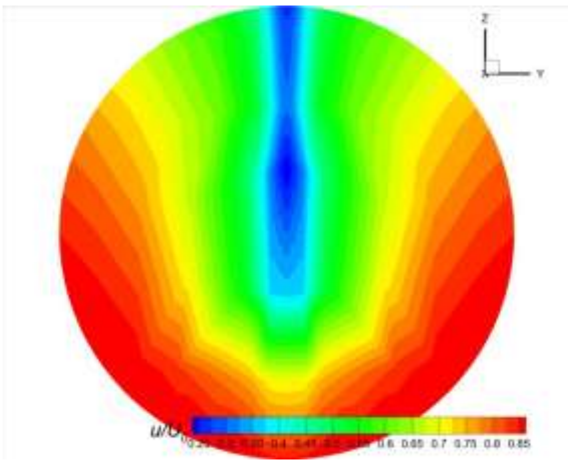

Figure 6. Comparison of numerical (CFD) results with experimental data (EFD)

Flow-field visualization based on iso-surfaces of the second invariant of the velocity gradient highlighted the characteristic helical development of the blade tip vortices. As the advance coefficient (J) increased, these vortical structures became larger and more pronounced, whereas the vorticity generated in the vicinity of the propeller blades and hub exhibited a noticeable reduction in intensity.

The pressure distribution on the blades confirms the stagnation effect at the leading edge, on the pressure side, and an extended low-pressure region on the suction side, both intensifying as the propeller loading increases (lower J).

3.2. SHIP RESISTANCE, TRIM AND SINKAGE OF A HULL

The second application concerns a container ship hull (length between perpendiculars 2.5 m, draft 0.117 m, beam 0.35 m, block coefficient 0.6505), analyzed for three Froude numbers (0.20; 0.22; 0.24), corresponding to speeds of 0.990; 1.089 and 1.188 m/s respectively. The computational mesh, also generated through unstructured hexahedral discretization, had approximately 2.12 million cells, and the free surface was treated using the water-air interface capturing method.

The results obtained show a monotonic increase in total resistance, dynamic sinkage and trim angle with increasing Froude number, as can be seen in Table 2.

Froude	Speed [m/s]	$C_T \times 10^3$	Sinkage $\sigma \times 10^2$ [m]	Trim τ [°]
0,20	0,990	4,425	0,104	0,104
0,22	1,089	4,371	0,128	0,128
0,24	1,188	4,386	0,155	0,155

Table 2. Total resistance, dynamic sinkage and trim, for the three Froude numbers

The time evolution of total resistance shows a rapid increase during the first approximately 7 seconds of simulation, followed by stabilization around a mean value, with low amplitude oscillations, indicating that the quasi-steady flow regime has been reached.

The profile of the waves generated by the ship, extracted at 10% and 20% of the length between perpendiculars respectively, was compared with the theoretical deep water wavelength, calculated directly from the Froude number; the differences obtained, ranging between 1% and 7%, fall within the expected range for this type of simplified comparison, since the theoretical formula does not account for the three dimensional effects of the wave system generated by a real hull.

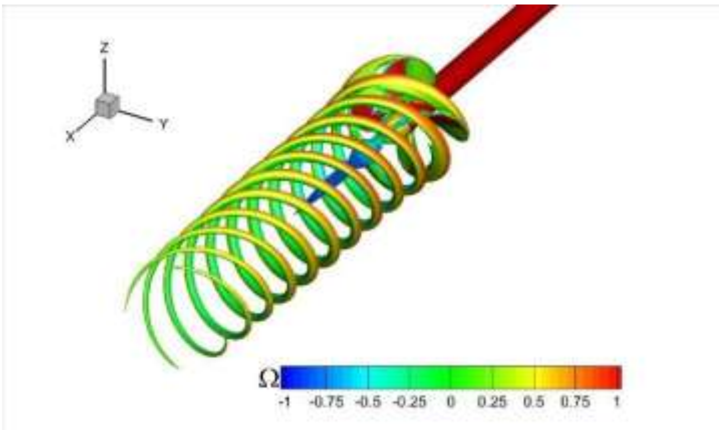

Figure 7. Flow field around the hull, highlighting the generated wave system (free surface)

International Journal of Instruction, Technology, and Social Sciences
Vol. 5, 106-115, 2026
ISSN: 2716-6546
www.ijitsc.net

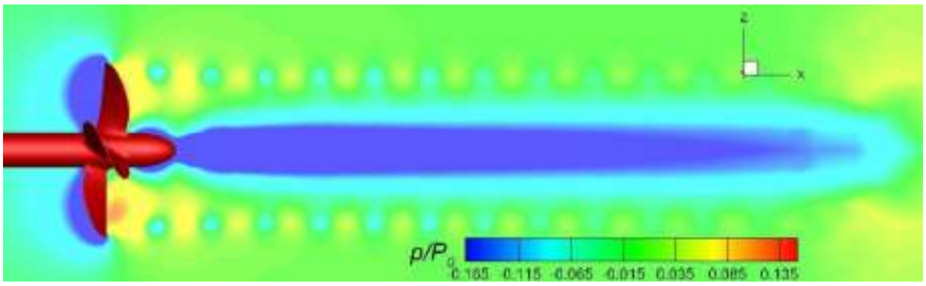

Figure 8. Pressure distribution along the hull

The pressure contours obtained on the hull surface indicate a gradual increase in hydrostatic pressure with depth below the free surface. At the same time, the relative velocity distributions evaluated in the three coordinate directions reveal the highest flow velocities over the main deck, where the structure is fully exposed to the incoming air stream, whereas the lowest velocities occur along the side shell near the waterline, corresponding to the region where the boundary layer reaches its greatest thickness.

4.CONCLUSIONS

The two applications presented illustrate, in different forms, the same simplified CFD methodology: defining a computational domain adapted to the geometry, generating an unstructured hexahedral mesh, selecting a turbulence model appropriate to the nature of the flow, and extracting non-dimensional coefficients comparable with reference data.

In both cases, the main sources of error remain the same: the finite resolution of the mesh, the approximate modeling of turbulence, and, in the case of free surface flow, the imprecise capturing of the water-air interface in regions with large wave-curvature gradients.

The findings show that numerical simulations can capture the overall hydrodynamic behavior of the propeller, although the computed force and moment values deviate from experimental data by roughly 15–20%. Even with this level of disagreement, key performance metrics, such as propeller efficiency and total resistance coefficient, stay in line with the experimental findings. This indicates that simplified CFD approaches offer a dependable means of comparing different design options and pinpointing those with better hydrodynamic behavior. That said, these models are best treated as tools for relative design comparison rather than as precise predictors of absolute hydrodynamic loads.

Looking ahead, the two case studies can be integrated into a single self-propulsion simulation, in which the propeller operates in the non-uniform wake generated by the hull, a natural next step of the CFD methodology applied to ship design.

This paper presented a simplified CFD methodology, structured in four stages, applicable both to the open-water analysis of a marine propeller and to the ship resistance analysis of a hull. Applying this methodology to the two case studies led to physically consistent results, comparable, within reasonable limits, with reference experimental data.

The results confirm that a simplified CFD methodology, even with limited computational resources, can support design decisions in the early stages, provided that the errors specific to each computed quantity are interpreted correctly.

5.REFERENCES

1. Lungu, A. Hydrodynamic loads and wake dynamics of a propeller working in oblique flow, 2000.

2. Lungu, A. Numerical simulation of the cavitating KP505 propeller working in open water conditions. IOP Conf. Ser.: Mater. Sci. Eng., 400, 042035. https://doi.org/10.1088/1757-899X/400/4/042035, 2018.

3. Bekhit, A. S., Lungu, A. Simulation of the POW performance of the JBC propeller, 2019.

4. Grlj, C. G., Degiuli, N., Farkas, A., Martić, I. Numerical Study of Scale Effects on Open Water Propeller Performance, 2022.

5. Nguyen, T. V. A Numerical Study on Open-Water Propeller Characteristics Using CFD, 2015.

6. Bekhit, A., Obreja, D. Numerical and Experimental Investigation on the Free-surface Flow and Total Resistance of the DTMB Surface Combatant. IOP Conf. Ser.: Mater. Sci. Eng., 916, 012008. https://doi.org/10.1088/1757-899X/916/1/012008, 2020.

7. Bekhit, A. S. Numerical simulation of the ship self-propulsion prediction using body force method and fully discretized propeller model. IOP Conf. Ser.: Mater. Sci. Eng., 400, 042004, 2018.

8. Deng, G. B., Queutey, P., Visonneau, M. RANS prediction of the KCS container ship flow including self-propulsion with simplified and geometrically exact propeller models. Proc. Gothenburg 2010 Workshop on CFD in Ship Hydrodynamics, 2010.

9.Nita, C. M., et all (2022). Experimental methods for determining the characteristic quantities of unconventional naval propellers. *Technium: Romanian Journal of Applied Sciences and Technology*, *4*(8), 56–63. https://doi.org/10.47577/technium.v4i8.7265

10. Queutey, P., Visonneau, M.An interface capturing method for free-surface hydrodynamic flows. Computers & Fluids, 36(9), 1481-1510. https://doi.org/10.1016/j.compfluid.2006.11.007, 2007.

11. S Susilowati, Silvana Dwi Nurherdiana, Sukirmiyadi, Erwan Adi Saputro, Mohammad Tauviqirrahman, Dani Arfiandani, & Paryanto. (2023). Multiphase Flow Analysis of Three-Dimensional Journal Bearing with Roughness Using Computational Fluid Dynamics (CFD). *Technium: Romanian Journal of Applied Sciences and Technology*, *16*(1), 310–316. https://doi.org/10.47577/technium.v16i.10003