THE ANNALS OF “DUNAREA DE JOS” UNIVERSITY OF GALATI
FASCICLE XI – SHIPBUILDING. ISSN 1221-4620, e-ISSN 2668-3156
2021

PROPULSION SIMULATION ON FULLY APPENDED SHIP MODEL

Andreea Mandru
“Dunarea de Jos” University of Galati,
Faculty of Naval Architecture, Galati, Domneasca Street, No. 47, 800008, Romania,
E-mail:andreea.mandru@ugal.ro

Liliana Rusu
“Dunarea de Jos” University of Galati,
Faculty of Engineering, Galati, Domneasca Street, No. 47, 800008, Romania,
E-mail:liliana.rusu@ugal.ro

Florin Pacuraru
“Dunarea de Jos” University of Galati,
Faculty of Naval Architecture, Galati, Domneasca Street, No. 47, 800008, Romania,
E-mail:florin.pacuraru@ugal.ro

ABSTRACT

This study presents the numerical investigation for the flow around the propeller of the ONR Tumblehome combatant in open water and for the flow around the same ship in the case of self-propulsion with actuator disk method. Computational Fluid Dynamics based on RANS-VOF solver have been used in order to analyse the flow. The free surface treatment is multi-phase flow approach, incompressible and nonmiscible flow phases are modelled through the use of conservation equations for each volume fraction of phase. Accuracy involves close attention to the physical modelling, particularly the effects of turbulence, as well as the numerical discretization.

Keywords: CFD, RANSE, propeller open water, self-propulsion, actuator disk.

1. INTRODUCTION

Marine propellers account for the vast majority of modern propulsion systems. Understanding their performance is critical for improving ship powering performance, reducing fuel consumption, and avoiding unwanted phenomena such as cavitation, noise, and vibration.

With the vast range of commercial CFD software available and the developments in computing capabilities that have recently resulted in High Performance Computations, CFD has become one of the most robust and dependable solutions for solving flow issues in all industrial domains. CFD in ship hydrodynamics has made significant progress in the last two decades, particularly when the problem is ship propulsion.

Today, CFD offers the possibility to simulate a propeller in open water in a free stream flow on a geometrically comparable propeller model, thus determining the propulsion performance. Also, self-propulsion simulation can be successfully performed using either the actuator disk method or the actual propeller. The current study presents open water simulations for a four-bladed propeller and self-propulsion simulations for a fully appended ship model using the actuator disc method.

The ONR Tumblehome (ONRT), a preliminary design for a surface combatant, is the subject of research for this paper. This ship model is one of the benchmark cases used in Tokyo 2015 CFD workshop in ship hydrodynamics [1]. The model of the ONRT equipped with appendages and the ONRT’s propeller is shown in Figure 1 respectivley

Figure 2, and the principle geometric characteristics are listed in Table 1.

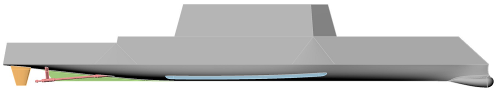


Fig. 1. ONR Thumblehom geometry

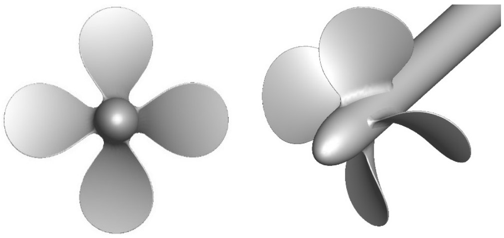


Fig. 2. ONR propeller geometry

Table 1. Main particulars of the ONRT model and ONRT's propeller

Main Particulars	Value
Length of waterline - L_{WL} [m]	3.147
Beam of waterline - B_{WL} [m]	0.384
Draft - T [m]	0.112
Displacement - Δ [m^3]	72.6
Wetted surface area - S_0 [m^2]	1.5
Block Coefficient - C_B	0.535
Propeller diameter – D [m]	0.106

2. NUMERICAL APPROACH

The flow solution around ONR Tumblehome has been computed using Fine^{TM}/Marine commercial code. The implicit solver uses the finite volume method to build the spatial discretization for the governing equation in order to solve the incompressible steady RANSE in a global approach, Guilmineau et al. [2]. For turbulence closure, the k-ω SST turbulence model with wall function formulation is used. Utilizing a face-based approach, pressure equation formulation (SIMPLE) is used to manage velocity-pressure coupling. The velocity field is obtained from the momentum conservation equations, whereas the pressure field is obtained from the mass conservation constraint, or continuity equation, which is then transformed into a pressure-equation. When dealing with turbulent flows, additional transport equations for modelled variables are discretized and solved using the same approaches as in Duvigneau et al. [3]. Convection and diffusion terms in RANSE are discretized using a second-order upwind and a central difference scheme, respectively. The free-surface capture strategy is based on the multi-phase flow approach with high-resolution interface schemes using the Volume of Fluid method. For each volume fraction of phase/fluid, conservation equations are used to simulate incompressible and nonmiscible flow phases, Queutey and Visonneau [4].

In the case of self-propulsion simulation, the actuator disk method has been used to model the effects of a propeller without using the real propeller. The approach integrates a body forces method with a RANS-based finite volume solver, and it treats propeller thrust and torque as a field of forces that may be added to the body force components in the RANS equations. The radial distribution of forces is based on non-iterative calculation of Stern et al [5], the Hough and Ordway [6].

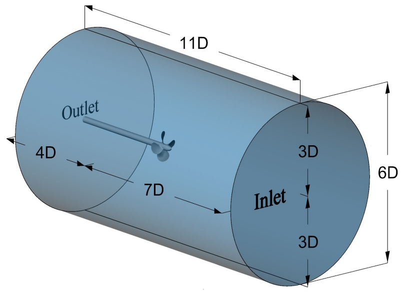


Fig. 2. Computational domain in the case of open water simulations

The computational domain for propeller open water simulations is a cylindrical geometry, whose diameter and length are 6 and 11 times the propeller diameter, respectively, of which 4 diameters are at the downstream of the propeller and 7 diameters are at the upstream of the propeller (Figure 3). The coordinate system is located at the geometric center of the propeller. The x-axis is oriented towards the upstream and the y-axes and z-axes are oriented in the lateral and vertical directions. Regarding the boundary conditions, prescribed pressure condition was used for the inlet, far field condition was applied for the outlet and outer boundary. Wall-function condition has been adopted on the cap and blades of the propeller and slip condition for the shaft of the propeller.

In the particular case of the ONR Tumblehome, the longitudinal half of the hall has been used in resistance simulation in order to reduce the computational costs. The dimensions of the computational domain (Figure 4) have been generated as $-3 \leq x/L_{WL} \leq 1$, $0 \leq x/L_{WL} \leq 1.5$, $-1.5 \leq x/L_{WL} \leq 0.5$, where the L_{WL} is the length of the waterline. The negative x-axes have been defined in the direction of flow, along with positive y-axes, starboard and z-axes in upward directions. The centreline plane has been defined as the boundary condition of symmetry. The inlet, outlet, and side boundaries have been defined as far field. The top and bottom of computational domain has been defined as prescribed pressure. On the hull surface, it has been applied the wall-function condition, while on deck has been adopted the slip condition.

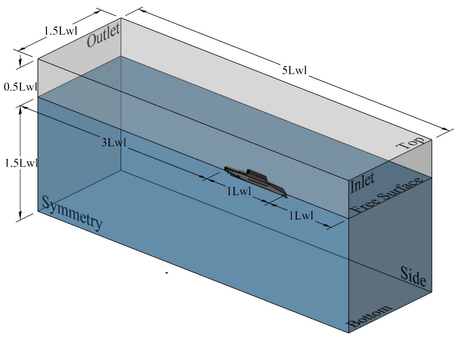


Fig. 4. Computational domain in the case of actuator disk propulsion

For open water simulation, an unstructured hexahedral mesh has been generated using the HEXPRESS automatic grid generator included in the Fine™/Marine software. The grid topology is an H-H type. Details of the grid around the propeller blade are shown in Figure 5. A sector refinement in the shape of a tour which covers the blade tips and another one next to the hub has been used to define a finer mesh in the wake area of the propeller. A mono-block unstructured grid has been generated to cover the entire computational domain along the propeller.

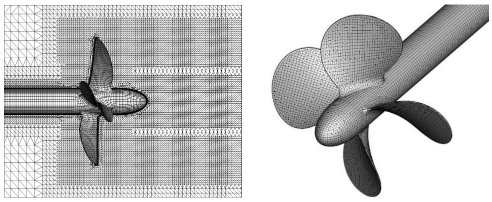


Fig. 5. Computational mesh generated around the propeller

An unstructured hexahedral mesh with an H-H type grid topology has also been used in the case of ONRT propelled with an actuator disk. Details of the near-hull grids are presented in Figure 6. A box refinement has been used to define a finer mesh in the sonar and actuator disk area. In the case of the transom stern, extra refinement has been applied to the transom curves. In the area where ship waves system develops, box refinement has been used to define a finer mesh. As long as both, trim and sinkage, are solved during the computations, an analytical weighted mesh deformation technique is used. For each appendage, extra refinement has been applied.

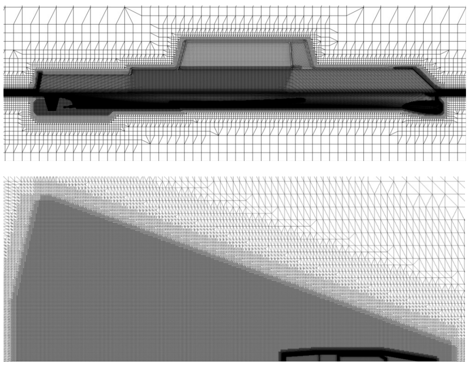


Fig. 5. Computational mesh generated around the ONRT hull

3. RESULTS AND DISCUSSION

In this paper, two studies were conducted: propeller in open water simulation and self-propulsion simulation with the actuator disk method. The obtained results will be detailed further below.

3.1. PROPELLER IN OPEN WATER

Propeller in open water simulations were performed for the ONRT's propeller, which is a 4-blade propeller with a diameter of 0.1066 m. Open water simulations were carried out for 14 sets of advanced ratios at a speed of 1.11 m/s. The calculated open water results were compared with the experimental results performed at IHR and available at the Tokyo 2015 CFD Workshop [1].

In Table 3, there are the numerical simulation results and the experimental results for the thrust coefficient and torque coefficient for all 14 advanced ratios, while in Figure 6, it is represented the performance diagram of the propeller in comparison with the experimental data.

Table 3. Comparison between the open water propeller performances coefficients measured and computed

J	k_T			$10k_Q$		
	EFD	CFD	|ε|%	EFD	CFD	|ε|%
0.1	0.60	0.62	2.5	1.46	1.51	3.0
0.2	0.59	0.58	-2.7	1.45	1.44	-0.6
0.3	0.55	0.52	-7.0	1.35	1.40	3.6
0.4	0.54	0.51	-5.1	1.32	0.34	1.2
0.5	0.49	0.48	-2.1	1.23	0.26	2.9
0.6	0.45	0.44	-1.9	1.14	1.19	3.9
0.7	0.40	0.41	2.4	1.01	1.11	8.6
0.8	0.35	0.37	4.1	0.91	1.03	11.6
0.9	0.32	032	-0.4	0.86	0.94	8.8
1.0	0.26	0.28	5.9	0.73	0.86	14.3
1.1	0.22	0.24	9.6	0.64	0.76	16.5
1.2	0.17	0.19	9.4	0.55	0.67	18.6
1.3	0.12	0.14	13.2	0.43	0.57	24.0
1.4	0.08	0.09	14.3	0.33	0.46	28.8

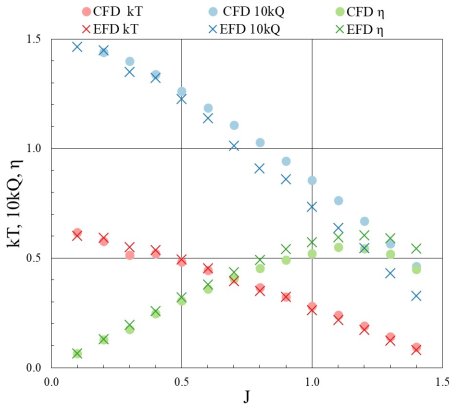
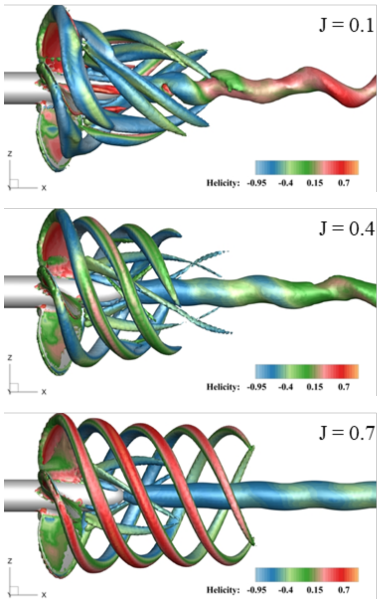


Fig. 6. Propeller performances diagram. Comparison between numerical solution and experimental data

The vortical structures developed in the wake of the propeller can be seen in Figure 7, which shows the iso-surface of the second invariant of the velocity gradient computed for five advance ratios (0.1, 0.4, 0.7, 1.0, and 1.3). The iso-surface (Q = 500) for the second invariant is colored by helicity.



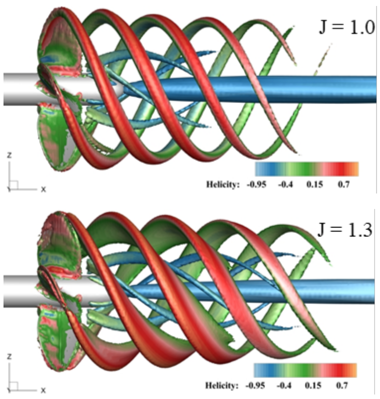


Fig. 7. Vortical structures released by the propeller

In Figure 8, there are presented the axial velocities computed in the longitudinal plane of symmetry for the same advanced ratio considered for Figure 7.

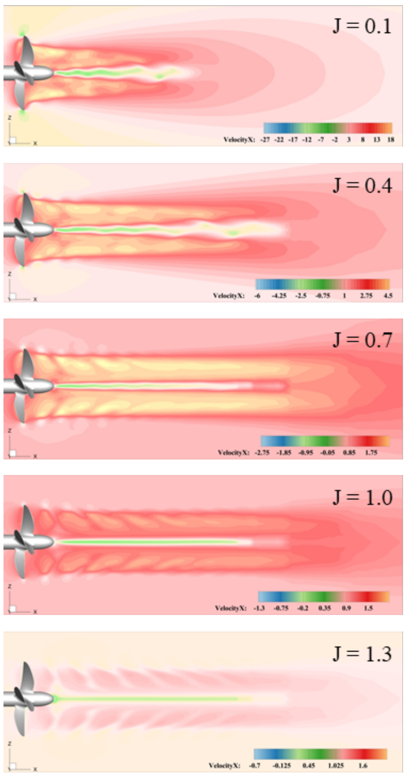


Fig. 8. Axial velocity computed in the longitudinal plane of symmetry

3.2. SELF PROPULSION WITH ACTUATOR DISK

Self-propulsion simulations with the actuator disk method were performed for the ONR Tumblehome hull. For this study, three configurations were considered: AD_ONRT - bare hull ONRT, AD_ONRT_shaft – ONRT hull with shaft, and AD_ONRT_appended – ONRT hull fully appended. The simulations were performed for five Froude numbers between 0.2 and 0.4. "AD" is an abbreviation fort "Actuator Disk".

In Table 5 the thrust values are represented, and Figure 9 shows the thrust curve for each configuration.

Table 5. Thrust values for each configuration

Fn	T [N]		
	AD_ONRT	AD_ONRT shaft	AD_ONRT appended
0.20	3.39	3.87	4.15
0.25	6.05	6.05	6.45
0.30	9.17	9.17	9.79
0.35	12.51	12.51	13.53
0.40	19.95	19.95	21.58

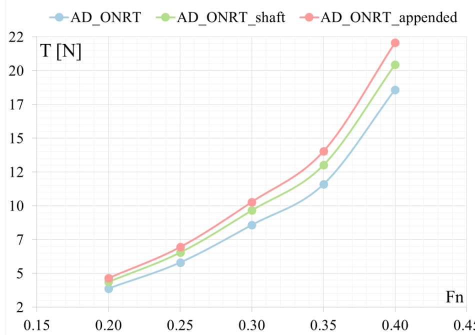


Fig. 9. Thrust curve computed for each configuration

Figure 10 illustrates the comparison between the axial velocities plotted in the propeller plane for each case at a speed of 1.11 m/s (corresponding to Froude number 0.2) for the resistance simulations [7] and self-propulsion simulation.

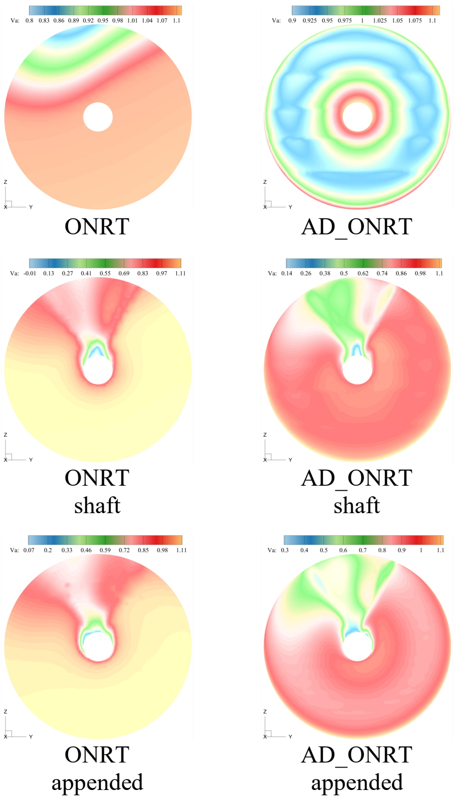


Fig. 10. Axial velocity computed in the propeller plane for Fn = 0.2. Comparison between resistance simulations and self-propulsion with actuator disk simulations for ONRT hull

4. CONCLUDING REMARKS

Numerical simulation of flow around ONRT propeller in free stream have been studied in order to analyses the propeller hydrodynamic performances. Beside the validation of the results for propeller open water test, hub vortex, tip and root propeller blade vortices have been captured.

Self-propulsion tests were performed to determine the thrust for different operating conditions.

The interaction of different appendage configurations on the wake flow during the self-propulsion test have been investigated.

REFERENCES

[1]. **National Maritime Research Institute (NMRI),** *"A Workshop on CFD in Ship Hydrodynamics",* Tokyo, 2015, http://www.t2015.nmri.go.jp.

[2]. **Guilmineau, E., Deng, G., B., Leroyer, A., Queutey, P., Visonneau, M., Wackers, J.,** *"Influence of the turbulence closures for the wake prediction of a marine propeller"*, Proc. of the 4th International Symposium on Marine Propulsors, 2015.

[3]. **Duvigneau, R., Visonneau, M., Deng, G., B.,** *"On the role played by turbulence closures in hull shape optimization at model and full-scale"*, J. Marine Science and Technology, 8 (1), 1–25, 2003.

[4]. **Queutey, P., Visonneau, M.,** *"An interface capturing method for free-surface hydrodynamic flows"*, Computers & Fluids, 36(9) pp 1481–1510, 2017.

[5]. **Stern, F., Kim, H., T., Patel, V., C., Chen, H., C.,** *"A viscous flow approach to the computation of propeller-hull interaction"*, Journal of Ship Research, vol. 32, no. 4, pp. 246–262, 1988.

[6]. **Hough, G., Ordway, D.,** "*The generalized actuator disk."* Technical Report TAR-TR 6401, Therm Advanced Research Inc., 1964.

[7]. **Mandru, A., Pacuraru, F.,** *"The effect of appendages on ship resistance"*, IOP Conf. Ser.: Mater. Sci. Eng. 1182 012041, 2021.

Paper received on November 17th, 2021