The effect of appendages on ship resistance

A Mandru and F Pacuraru
"Dunarea de Jos" University of Galati-Romania, Faculty of Naval Architecture, Department of Naval Architecture, 47 Domneasca Street, 800008, Galati, Romania
E-mail address: florin.pacuraru@ugal.ro

Abstract: The availability of the robust commercial computational fluid dynamics (CFD) software and the increasing power of high-performance computers (HPC) boosts the use of CFD techniques for numerical investigations of ship hydrodynamics performances. Nowadays, CFD tools provide a rather accurate solution for complex physical phenomena, such as wave braking, turbulence, flow detachment and overturning. Consequently, the ability to capture this phenomena allows a detailed insight into the flow mechanism that trigger and sustain them.

The paper focuses on a numerical investigation of flow around hull a fully appended ONRT (Office of Naval Research Tumblehome) hull and to estimate the effect of the appendages on ship hydrodynamics performances. Comparison with experimental towing tank results showed good agreement with a difference of up to 2%.

1. Introduction

Due to the widespread availability of robust commercial Computational Fluid Dynamics (CFD) software and high-performance computers, CFD is increasingly being used to solve fluid engineering problems and ship hydrodynamics is no exception. Assuming that the flow solvers are capable of dealing with practical complex geometries as well as taking into account complex physical phenomena such as turbulence and free surface, CFD-based design tools can provide a fairly accurate solution to different ship hydrodynamic problems. The naval architect uses viscous free-surface flow calculations to improve the hull forms for achieving a homogeneous velocity distribution, determining the optimum inclination angle for shaft brackets, studying the hydrodynamic interactions between various appendages and arrangements, and to investigate flow separation regions. The key issues in appendages drag investigation is to provide details about the resistance, interaction and scaling of appendages.

The paper is focused on a numerical investigation using RANS computation to solve the viscous flow around a fully appended combatant ship hull. To better understand the effects of different configurations of the appendages on the wake structure in the propeller disk, several sets of calculations was performed. The NUMECA/FineMarine commercial code has been employed to compute the flow field structure surrounding the ship hull, as well as the forces acting on the bare hull and appendages. The effect of appendages on ship drag of displacement vessels has been studied by
Iwan et al. [1] which considered the scale effects of appendages drag, [2] introduced an Appendage effective Index to assess the effectiveness of an appendage, Lasky [3] developed a procedure for scaling of appendages drag. Numerically investigation of the flow around the ship with appendages has been reported in [4, 5]. Detailed analysis of the flow around the juncture between a plate and a foil considering different inclination angle of the foil have been investigated by [6, 7].

2. Ship hull and computation conditions

The paper considers the investigation of the ONR Tumblehome (ONRT) which is a preliminary design for a new surface combatant that is available to the public for fundamental research. The 1/49 scaled ship model is used as one of the benchmark cases in Tokyo 2015 CFD workshop in ship hydrodynamics. Extensive experiments were performed at IIHR Hydraulics Wave Basin Facility for this ship model. The ONRT’s bow shapes are dominated by the presence of a prominent sonar at the bottom, and at the ship’s stern there are two shaft lines, supported by brackets, with two propellers and two rudders. The model of the ONRT equipped with appendages is shown in figure 1, and the principle geometric characteristics are listed in table 1.

![Figure 1. ONR Tumblehome.](image)

| Dimensions                              | Symbol | Units | Model scale | Full scale |
|-----------------------------------------|--------|-------|-------------|------------|
| Length of waterline                     | L<sub>WL</sub> | m     | 3.147       | 154        |
| Maximum beam of waterline               | B<sub>WL</sub> | m     | 0.384       | 18.78      |
| Depth                                   | D      | m     | 0.266       | 14.5       |
| Draft                                   | T      | m     | 0.112       | 5.494      |
| Displacement                            | Δ      | kg/t  | 72.6        | 8507       |
| Wetted surface area (fully appended)    | S<sub>0</sub> | m<sup>2</sup> | 1.5 | NA |
| Bloc coefficient                         | C<sub>B</sub> | –     | 0.535       | 0.535      |
| LCB (aft of FP)                          | LCB    | m     | 1.625       | NA         |
| Vertical center of gravity (from keel)  | KG     | m     | 0.156       | NA         |
| Metacentric height                      | GM     | m     | 0.0422      | NA         |
| Speed                                   | v      | m/s   | 1.11        | 7.77       |

Table 1. ONR Tumblehome principle geometric characteristics.

In table 2 are presented the test cases for calm water resistance analysis. Six different sets of simulations were carried out for 5 speeds corresponding to Froude numbers between 0.20 and 0.40. The first series of computations were performed in the case of the ONR Tumblehome without appendages in order to investigate the flow characteristics around the bare hull. The purpose of the next four configurations is to determine the influence of each appendage on the ONRT total resistance.
The last set of simulations considers the investigation of the flow characteristics around the ONR Tumblehome fully appended.

Table 2. Calm water resistance computed test cases.

| Configurations            | Description                      | Wetted surface area [m²] | List of speeds [m/s] | Corresponding Froude number |
|---------------------------|----------------------------------|--------------------------|----------------------|-----------------------------|
| ONRT                      | ONR Tumblehome bare hull         | 1.310                    | 1.11                 | 0.20                        |
| ONRT_bilge_keel           | ONR Tumblehome with bilge keel   | 1.402                    | 1.39                 | 0.25                        |
| ONRT_skeg                 | ONR Tumblehome with skeg         | 1.343                    | 1.67                 | 0.30                        |
| ONRT_rudder               | ONR Tumblehome with rudder       | 1.363                    | 1.94                 | 0.35                        |
| ONRT_shaft                | ONR Tumblehome with shaft        | 1.357                    | 2.22                 | 0.40                        |
| ONRT_appended             | ONR Tumblehome fully appended    | 1.537                    |                      |                             |

3. Numerical Approach

3.1. Governing equations

The averaged continuity and momentum equations for incompressible flows involving external forces (1 and 2) can be written in tensor form in the Cartesian coordinate system as:

\[
\frac{\partial (\rho \overline{u}_i)}{\partial x_i} = 0
\] (1)

\[
\frac{\partial (\rho \overline{u}_i \overline{u}_j)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho \overline{u}_i \overline{u}_j + \rho u_i u_j \right) = - \frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}
\] (2)

where \( \overline{u}_i \) is the relative averaged velocity vector of flow between the fluid and the control volume, \( \overline{u}_i u_j \) is the Reynolds stresses, \( P \) is the mean pressure and \( \tau_{ij} \) is the mean viscous stress tensor components for Newtonian fluid under the incompressible flow assumption, and it can be written as in equation (3):

\[
\tau_{ij} = \mu \left( \frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right)
\] (3)

The NUMECA/FineMarine commercial code has been employed to compute the flow solution around the ONR Tumblehome. 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 [8]. The k-ω SST turbulence model with wall function formulation is used for turbulence closure in this paper. Velocity-pressure coupling is handled with pressure equation formulation (SIMPLE) using a face-based approach. The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure-equation. In the case of turbulent flows, additional transport equations for modelled variables are discretized and solved using the same principles Duvigneau et al [9]. A second-order upwind scheme and a central difference scheme, respectively, are used to discretize the RANSE’s convection and diffusion terms.
For resistance computations, the flow is accelerated using a steady quasi-static approach over a time-span that corresponds to the relationship between the ship’s length and the inflow velocity such that $T_{acc} = 2.0L_{pp}/U_w$. The computations time step is calculated with respect to the equation $\Delta t = 0.005L_{pp}/U_w$ for $k-\omega$ SST turbulence model as it is suggested by the ITTC recommended procedures for CFD [10]. Free-surface capturing strategy is based on multi-phase flow approach using Volume of Fluid method with high-resolution interface schemes. Incompressible and nonmiscible flow phases are modelled using conservation equations for each volume fraction of phase/fluid, Queutey and Visonneau [11].

3.2. Computational domain
Dimension of the computational domain (figure 2) have been generated as $-3 \leq x/L_{WL} \leq 1$, $0 \leq y/L_{WL} \leq 1.5$, $-1.5 \leq z/L_{WL} \leq 0.5$, where the $L_{WL}$ is the length of waterline. The origin of the Cartesian coordinate axis has been selected as the point where fore perpendicular of model intersects the waterline. The negative x-axes have been defined in the direction of flow, positive y-axes, starboard beam and z-axes upward directions. Only half of hall has been used in resistance simulation in order to reduce the computational costs. 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. Wall-function condition has been adopted on the hull surface and slip condition for the deck and superstructure.

![Figure 2. Computational domain dimensions and boundary condition.](image)

3.3. Grid generation
An unstructured hexahedral mesh has been generated to cover the entire computational domain along the ONR Tumblehome model. The grid topology is a H-H type. Surface grid refinement at the bow and stern area has been considered. A box refinement has been used to define a finer mesh in the area where ship waves system develops. An analytical weighting mesh deformation approach is employed as long as both trim and sinkage were solved during the computations.

4. Results and discussions
In experimental ship hydrodynamics, there is a common practice to measure the drag of the appendages as the difference in drag conducting resistance tests with a ship-model in the bare hull and the appended condition. In the present study, several pilot calculations were performed for the
flow around the bare hull to investigate the flow characteristics, but also determine the bare hull resistance which is considered the reference value for the comparison of different appendages configuration. For the bare hull, the most important task is to capture the vortex structures that develop in the boundary layer of the ship. Unlike full-shaped ships, such as tanks and bulk carriers, where the flow is strongly influenced by bilge vortices, fast military ships do not experience flows with such vortex structures. In the case of the ONRT combatant ship model, the main flow characteristic is the vortex generated by the sonar dome located at the bow of the ship, as it is depicted in figure 3.

Analysing the variation of the total resistance coefficient against the Froude number, one can see from the table 3 and figure 4, that large increase in the slope of both curves appear at Froude 0.35, which is caused mainly by the significant increase of the trim as presented in figure 5. The trim is related to the position of wave trough/crest at the ship extremities, which is demonstrated by representation of wave pattern generated by the ship for Froude numbers between 0.2 and 0.4 plotted in figure 6.

### Table 3. ONRT bare hull computed results.

| Fn  | v [m/s] | R_T [N] | C_T   | z [m] | θ [deg] |
|-----|---------|---------|-------|-------|---------|
| 0.20| 1.11    | 3.48    | 4.30E-03 | 0.083 | 0.067   |
| 0.25| 1.39    | 5.49    | 4.35E-03 | 0.082 | 0.096   |
| 0.30| 1.67    | 8.57    | 4.72E-03 | 0.079 | 0.088   |
| 0.35| 1.94    | 11.76   | 4.75E-03 | 0.077 | 0.162   |
| 0.40| 2.22    | 19.04   | 5.89E-03 | 0.073 | -0.105  |

![](image1.png)

Figure 3. Vortex structure generated by the sonar dome.

![](image2.png)

Figure 4. ONRT bare hull total resistance coefficient.

![](image3.png)

Figure 5. ONRT bare hull trim.
The results provided by Delen and Bal [11] were used as a reference for the validation of the calculations performed. In the following, it is proposed the quantitative validation of the numerical solution obtained for the hull with skeg configuration. Table 4 and figure 7 presents a comparison between the calculated forward resistance coefficients and those obtained experimentally. The percentage difference between the coefficients of the total resistance to advance ($C_T$), determined numerically and experimentally, is less than 2%.

| Fn | $R_T [N]$ | $C_T$ | $\varepsilon [%]$ |
|----|----------|-------|-----------------|
| 0.20 | 3.64 | 3.57 | 4.39E-03 | 4.30E-03 | 1.96 |
| 0.25 | 5.82 | 5.71 | 4.49E-03 | 4.41E-03 | 1.82 |
| 0.30 | 9.04 | 8.86 | 4.84E-03 | 4.75E-03 | 1.85 |

Five different sets simulations were carried out for five speeds corresponding to Froude numbers between 0.2 and 0.40 to compute the flow around five different configurations of the hull and appendages: hull with skeg (ONRT_skeg), hull with bilge keel (ONRT_bilge_keel), hull with shaft (ONRT_shaft), hull with rudder (ONRT_rudder) and full appended hull (ONRT_appended), in order to account the effect of each appendage on ship resistance. Bare hull results have been considered as a reference case for all the comparisons. Ship resistance curves for each configuration investigated are plotted in figure 8. Table 5 includes the relative variation of ship resistance for each appendage configuration. Analysis of computational results revealed that the main contribution in drag, of about 8 percent, is added by the rudder and the shaft. Another conclusion that can be drawn is that the influence of the interference between the appendages is not significant. Moreover, for each
configuration the relative difference in resistance drops at Froude number 0.4, fact which is related to the significant increase in ship resistance at that speed, which is mainly caused by the wave resistance. A comparison between bare hull and full appended in terms of variation of total ship resistance coefficient (figure 9) and trim angle (figure 10) showed the same behaviour. The maximum effect of the appendages on the ship drag is 28.7% and it appear for the Froude number 0.35.

Another area of interest in terms of flow mechanism is the stern of the ship, because in this area are located the propulsion and steering systems. To gain more insight on appendages hydrodynamics, the iso-surface second invariant $Q^*=25$ colored is depicted in figure 11 in order to capture the interferes of the vortex structures generated appendages and their junction with the hull.
5. Conclusions
A systematic study of appendages influence on ship resistance at model scale has been performed based on RANS-VOF simulation. Six appendages configurations at five different Froude number have been investigated.

The predicted total resistance coefficient computed for hull with skeg were compared with the experiment of Delen and Bal [12], showed a good agreement with a difference of less than 2%. Analysis of computational results revealed that the main contribution in drag, of about 8 percent, is added by the rudder and the shaft. Another conclusion that can be drawn is that the influence of the interference between the appendages is not significant. The maximum effect of the appendages on the ship drag is 28.7 % and it appears for the Froude number 0.35.

To gain more insight on appendages hydrodynamics the vortical structures generated by the appendages have been investigated. Further studies will focus on propulsion and seakeeping simulation, but also on full scale investigation.

6. References
[1] Iwan MK, Muhammad SR, Yaseen AA 2020 Scaling of RV Athena’s Appendage Drag Using CFD, Advancement in Emerging Technologies and Engineering Applications, Springer, Editor Chun Lin Saw.
[2] Shiju J, Kareem K, Praveen PC, Korulla M, Panigrahi PK Ship Hull Appendages: A Case Study International Journal of Innovative Research and Development 1(10) 74-89.
[3] Lasky MP 1980 An investigation of appendage drag, David Taylor Naval Ship Research and Development Center Bethesda MD, report.
[4] Oliva RA, Pérez RL, Pérez FL 2013 A contribution to appendage drag extrapolation using computational tools Proc. Developments in Maritime Transportation and Exploitation of Sea Resources.
[5] Ozdemir CO and Barlas B 2018 Free surface flow simulation around an appended ship hull Brodogradnja 69(3) pp 25-41.
[6] Lungu A and Ungureanu C 2008 Numerical Study of a 3-D Juncture Flow AIP Conference Proceedings 1048(1) pp 839-842.
[7] Lungu A and Ungureanu C 2009 Numerical simulation of the turbulent flow around a strut mounted on a plate AIP Conference Proceedings 1168(1) 647-650.
[8] Guilmineau E, Deng GB, Leroyer A, Queutey P, Visonneau M, Wackers J 2015 Influence of the turbulence closures for the wake prediction of a marine propeller. Proc. of the 4th International Symposium on Marine Propulsors Austin, Texas, USA pp.177-183.
[9] Duvigneau R, Visonneau M, Deng G.B 2003 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.
[10] International Towing Tank Conference (ITTC) 2014 ITTC – Recommended Procedures and Guidelines Practical guidelines for ship CFD applications 7.5-03-02-03, Revision 01.
[11] Queutey P, Visonneau M 2007 An interface capturing method for free-surface hydrodynamic flows Computers & Fluids 36(9) pp 1481–1510.
[12] Delen C and Bal S 2019 Uncertainty analysis of numerical and experimental resistance test for ONR Tumblehome Sustainable Development and Innovations in Marine Technologies CRC Press.

Acknowledgement
The research study presented in this article was developed in the frame of “Dunarea de Jos” University of Galati, Naval Architecture Research Centre.