Numerical and Experimental Investigation on the Free-surface Flow and Total Resistance of the DTMB Surface Combatant

A Bekhit and D Obreja
“Dunarea de Jos” University of Galati, Faculty of Naval Architecture, 47 Domneasca Street, 800008, Galati, Romania
E-mail: adham.bekhit@ugal.ro

Abstract. Starting with the initial design stage, the ship resistance constitutes one of the most important hydrodynamic aspects. In order to generate and optimize the body lines plan, the ship resistance performance must be considered. In this case, both numerical and experimental methods must be applied to obtain the minimum ship resistance controlled by the speed domain. In the last years, numerical studies based on the Computational Fluid Dynamic (CFD) were implemented in the initial ship design stage in order to analyse the free-surface flow and to predict the ship resistance. The numerical optimization process of the hull forms must be validated by means of the experimental model tests in the specific towing tanks. The constant evolution of the numerical and experimental methodologies can be noted. As a consequence, the accuracy level of the ship resistance prediction is increased; though, difficult numerical problems can occur in case of the unconventional hull forms. Taking into account this aspect, a complex numerical and experimental investigation of the ship resistance problem and free-surface topology was performed in the case of the DTMB surface combatant. A parallel numerical simulation was carried out by using the viscous flow solver ISIS–CFD of the software FINETM/Marine provided by NUMECA. The solver is based on the finite volume method to build the spatial discretization of the transport equation to resolve the Reynolds-Averaged Navier Stokes (RANS) equation. Closure to turbulence is achieved by making use of the $k-\omega$ SST model, while the free-surface is captured through an air-water interface based on the Volume of Fluid (VOF) method. For the validation purpose, the tank conditions are reintroduced in the numerical model to account for any banking effect, especially for the high speed cases. Experimental model tests were performed in accordance with the ITTC standard procedures, while the extrapolation process to the full-scale was obtained by using an in-house code, based on the ITTC 57 method. The experimental results are compared through with some internationally recognized towing tank experiments that have been performed for a similar ship model, showing a good agreement. Also, the numerical results are validated against the experimental data and showed to be within a satisfactory congruence.

1. Introduction
Predicting ship resistance and free-surface flow is one of the most essential aspects in the modern ship hydrodynamics. Ship resistance stands as the primary parameter a designer can think about in the initial design stage. The total resistance along with the propulsive efficiency can set the base for the required power, which represents the most important element in the initial design stage that can decide the design quality from an economical and environmental point of view. On the other hand, the free-
surface flow prediction can help reducing the total resistance by means of hull form optimization in order to control mainly the wave making resistance to increase the energy efficiency of the ship. Most recently, the continuous development in the marine industry imposes new challenges through the demands of more complex designs and unconventional ship geometries. Theoretical based methods, such as the methodical series or statistical based, for example, the Holtrop & Mennen's method, can be very useful and straightforward for conventional ship types. Unfortunately, when speaking about modern design requirements, such methods cannot be implemented. Besides, these methods can be useful to predict forces and moments, but the flow configuration and free-surface are completely disregarded. The recent three decades recorded a very promising success for the Computational Fluid Dynamics (CFD) methods in predicting ship hydrodynamics aspects in general, and more specifically the ship resistance and free-surface flows. The availability and the evolution of the computational capacity that grew significantly in the past decades opened the gates for more complex physical modelling and faster turnaround simulation time. This resulted, not only in increasing the popularity of applying CFD in the initial design stage, but also increased the results accuracy and reliability. Still, validation of the numerical results by means of experimental tests remains crucially important to set up the final decision for the proposed design.

Heading from this principle, this study focuses on the combination of the numerical and experimental methods in order to predict the free-surface flow and total resistance of the unconventional and geometrically complex ship model of the David Taylor Model Basin (DTMB), US Navy surface combatant. A consistent series of numerical and experimental investigations are performed for the bare hull ship model sailing in calm water with two degrees of freedom at different speeds. The general purpose, as previously described, is to predict the ship resistance curve and free-surface flow at different velocities in the medium-high speed stage. The detailed is to highlight the reliability of mixing the CFD and experimental methods to predict the hydrodynamic aspects of concern to use it as a reference for similar unconventional geometries new designs. This approach is achieved by means of a series of systematic verification and validation studies to assess the numerical errors and uncertainties. Other validation procedures are performed; all aimed at assessing the accuracy of the experimental data measured in the towing tank in the current study against some world recognized towing tanks whose data are available in the public domain.

The DTMB ship model analyzed in this study was primarily introduced in the Gothenburg 2000 Workshop on Computational Ship Hydrodynamics along with two other geometries in order to assess the state of the art in CFD for hydrodynamic applications [1]. Series of experiments were performed on different geometrically similar ship models built and tested, such as the model of 5.72 m length which was built and tested in the Istituto Nazionale per Studi ed Esperienze di Architettura Navale (INSEAN) in Italy and reported in [2] and the model of 3.048 m length which was built and tested in Iowa Institute of Hydraulic Research (IIHR).

Series of numerical investigation have been performed worldwide for different hydrodynamic aspects such as resistance, propulsion, seakeeping and maneuvering. Likely, the model has been a scope of research in the “Dunarea de Jos” University of Galati; where, a verification and validation study was performed by the author for the INSEAN model and reported in [3]. The appended hull was investigated by the help of the software SHIPFLOW flow solver to predict the viscous flow around the appended hull as reported in [4], a similar attempt for the appended hull was performed by the author using NUMECA FINE™/Marine viscous flow solver [5].

The current study is applied on the bare hull of the DTMB model of 3.232 m length which was built and tested in the “Dunarea de Jos” University of Galati to assess the total resistance and free-surface flow numerically as well as experimentally.

2. Geometry and analysis conditions

The ship model under investigation in this research study is the DTMB, whose preliminary design was introduced in 1980 as a benchmark hull for both explication of flow physics and CFD validation for a non-conventional ship model with a sonar dome and transom stern. The full scale ship has never been
built; however, several models were built and tested by various internationally well recognized towing tank organizations such as National Maritime Research Institute (NMRI) in Japan, the (INSEAN) in Italy, IIHR and the Naval Surface Warfare Center, Carderock Division (NSWC, formerly known as DTMB). A massive database for the ship model tests can be found in the public domain contains experimental data that were performed for various ship hydrodynamic aspects including ship resistance, vertical motion, free-surface, local flow configurations, seakeeping, roll decay, static drift and captive maneuvering.

This study includes only the resistance and free-surface flow investigation obtained numerically and validated experimentally for a new built ship model with scale 1/44, which corresponds to a length of waterline $L_{WL} = 3.232$ m and a draft of 0.14 m. It is worth mentioning that for this type of ships, the waterline length is usually chosen as the reference length, taking into consideration that the rudder does not exist in bare hull condition. Full details of the model ship characteristics are provided in table 1, compared to the full-scale ship, while the geometry of the ship is depicted in figure 1.

### Table 1. Ship model and full scale characteristics.

| Particulars                          | Unit     | Full Scale | Model – UGAL * |
|--------------------------------------|----------|------------|----------------|
| Scale                                | -        | -          | 1:44           |
| Length of Waterline ($L_{WL}$)       | [m]      | 142.0      | 3.232          |
| Beam ($B$)                           | [m]      | 19.06      | 0.434          |
| Depth ($D$)                          | [m]      | 10.98      | 0.25           |
| Draft ($T$)                          | [m]      | 6.15       | 0.14           |
| Volumetric Displacement ($V$)        | [m$^3$]  | 8424.4     | 0.099          |
| Wetted Surface Area ($S$)            | [m$^2$]  | 2972.6     | 1.54           |
| Block Coefficient ($C_B$)            | -        | 0.507      | 0.507          |
| Longitudinal Position of Centre of Gravity ($x_{CG}$) from F.P | [m] | 71.676 | 1.629 |
| Lateral Position of Centre of Gravity ($y_{CG}$) | [m] | 0 | 0 |
| Vertical Position of Centre of Gravity ($z_{CG}$) | [m] | 7.54 | 0.1718 |
| LCB (%$L_{PP}$), fwd+                | -        | -0.683     | -0.683         |

*aModel – UGAL* = the model built and tested in the University of Galati.

The experimental and numerical analysis conditions for ship model include seven different cases assigned based on the ship velocity. The ship is towed ahead with the corresponding speed to a Froude number starting with 0.2 reaching 0.44 with a step 0.04. Table 2 summarizes the computational cases and their corresponding Froude number $Fr = U/(gL_{WL})^{0.5}$ and Reynolds number $Re = UL_{WL}/\nu$, where $U$ represents the ship speed in m/s and $\nu$ represents the water kinematic viscosity in m$^2$/s. Only two degrees of freedom for the vertical ship motions are included in both experimental and numerical.
studies, i.e. the axial translation in \( z \)-directions (sinkage) and rotation around \( y \)-axis (trim), while all the other motions are locked.

Table 2. Computational cases and corresponding ship speed parameters.

| Case Number | C1 \([\text{m/s}]\) | C2 \([\text{m/s}]\) | C3 \([\text{m/s}]\) | C4 \([\text{m/s}]\) | C5 \([\text{m/s}]\) | C6 \([\text{m/s}]\) | C7 \([\text{m/s}]\) |
|-------------|----------------------|----------------------|----------------------|----------------------|----------------------|----------------------|----------------------|
| \( U \) \([-] \) | 1.126 | 1.351 | 1.577 | 1.802 | 2.027 | 2.252 | 2.477 |
| \( Fr \) \([-] \) | 0.20 | 0.24 | 0.28 | 0.32 | 0.36 | 0.40 | 0.44 |
| \( Re \times 10^6 \) | 3.46 | 4.15 | 4.84 | 5.53 | 6.22 | 6.91 | 7.60 |

3. The CFD study

This section describes the numerical-based simulation of the DTMB-UGAL ship model for the aforementioned seven cases and provides general information for the solver used, the governing equation and turbulence closure model, computational domain and boundary conditions, discretization grid and finally the solution strategy and framework. Numerical results will be covered separately in section 4.

3.1. Numerical solver

The numerical simulations performed in this study are carried out using the ISIS-CFD solver of the software FINE™/Marine provided by NUMECA. The solver is based on the finite volume method to build the spatial discretization of the transport equation to solve the Reynolds-Averaged Navier Stokes Equations (RANSE) [6]. The solver implies a face-based spatial discretization that constructs the fluxes face by face, which make it suitable for using arbitrary shape control volumes with arbitrary number of faces. Hence, it relies on the use of unstructured grids to provide more flexibility for complex hull geometries, ship appendages, etc. The temporal discretization is cell-centred based on a second order three-level scheme. The velocity field is obtained from the momentum equation; while the pressure is extracted from the mass constraint or continuity equation transformed into a pressure equation conform a velocity pressure coupling based on a Rhie and Chaw SIMPLE type algorithm [6]. Closure to turbulence is achieved through the Menter two-equation Shear Stress Transport \( k-\omega \) SST model. Multi-phase flow is used to model the free-surface interface based on the Volume of Fluid (VOF) interface capturing method, where incompressible and non-miscible flow phases are modelled by introducing conservation equations of the volume fraction for each volume phase/fluid [6]. Convection and diffusion terms in the governing equations are discretized using second-order upwind and central differencing scheme, respectively.

3.2. Governing equations

The time averaged continuity and momentum equations for the incompressible flow with external forces, can be written in tensor form, in the Cartesian coordinate system as

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

\[
\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho \bar{u}_i \bar{u}_j + \rho \bar{u}_j \bar{u}_i \right) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}
\]

where \( \bar{u}_i \) is the relative averaged velocity vector of flow between the fluid and the control volume, \( \bar{u}_i \bar{u}_j \) is the Reynolds stresses, \( \bar{p} \) 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 expressed as

\[
\bar{\tau}_{ij} = \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \tag{3}
\]
3.3. Computational domain and boundary conditions
The computational domain is having a rectangular prism configuration that represents only the half body of the ship in order to reduce the computational cost and effort, taking into consideration the symmetry of the hull and the fact that the ship is moving only in the forward direction. The dimensions of the computational domain in the Cartesian coordinate system is chosen based on two approaches; a general domain and specific domain dimensions, as illustrated in figure 2. The first has its dimensions in (x–y–z) directions set as $(5.0L_{WL}–2.0L_{WL}–2.0L_{WL})$, which are chosen according to the ITTC recommended procedures for ship CFD applications [7]; while the latter is selected specifically to regenerate the towing tank dimensions in order to simulate the effect of the tank walls on the results. The corresponding dimensions for the second approach in (x–y–z) directions are $(5.0L_{WL}–0.67L_{WL}–2.0L_{WL})$. A global earth-fixed reference frame is set for the numerical analysis located at the point $(0, 0, 0)$ of the forward perpendicular (F.P.) which results from the intersection of the undisturbed free-surface with the stem line of the ship.

(a)

(b)

Figure 2. Computational domain, dimensions and boundary conditions showing: (a) general domain configuration and (b) the regenerated towing tank-like domain.
The boundary conditions are set on the domain boundaries and solid surfaces as follows:
- The inflow boundary is located upstream at distance 1.0LWL from the F.P. with a velocity inlet boundary condition;
- The outflow boundary is located downstream at distance 4.0LWL from the F.P. with a pressure outlet boundary condition;
- Bottom and top boundaries are set at distances 1.5LWL and 0.5LWL, respectively, from the undisturbed free-surface, which is located at z=0. The boundary condition for top and bottom is set to the prescribed pressure, updating the hydrostatic pressure during simulation.
- Symmetry boundary condition is set at y=0;
- The side boundary is chosen to be at distances 2.0LWL and 0.67LWL for the general and towing tank approaches, respectively. In general approach the boundary condition for the side is set to a far field, while for the towing tank approach is set to a solid wall with no slip condition having a wall-modelled treatment imposing y’=35.
- Ship hull is set to a solid body with no slip condition having a wall-modelled treatment imposing y’=35.
- The ship deck is set to a slip condition, bearing in mind the fact that it remains in air during all the simulation time, where the relative viscosity effect in air could be neglected compared to that in water.

3.4. Computational grids
The computational grids generated for the numerical simulations performed in this study are created by making use of the unstructured hexahedral grid generator HEXPRESS™ that is included as a semi-automatic grid generator module in the FINE™/Marine package. In the beginning, three computational grid are generated with respect to the simulation approach; i.e. one for the general approach and two for the towing tank simulator approach; the first is having the side wall active by imposing a no slip condition, whereas the second is having non-active sides imposing a far field boundary condition on the domain side boundary. These two grids are generated identically to highlight the influence of the tank walls on the total resistance of the ship and the wave reflections on the side boundary. Then, in the scope of assessing the numerical errors and uncertainties, the general grid is systematically coarsened to regenerate a set of four geometrically similar grids to be used in the grid convergence study. For the consistency of the numerical results, the first three grids are generated with identical grid parameters such as, initial cell size, number of refinements for curves, surfaces, free-surface refinement criteria, refinement diffusion and finally, the viscous layer insertion parameters. For the grid convergence study, the initial cell size and refinement diffusion parameters were systematically varied to insure geometrically similar grids with a relative refinement ratio \( r_G \approx 1.7 \). The computational grids are summarized in table 3; where, M1 refers to the finest grid and M4 refers to the coarsest, whereas the grid configuration and the different refinement zones are highlighted in figure 3.

| Computational approach | Towing tank simulator | General case |
|------------------------|-----------------------|--------------|
|                        | active walls (AW)     | disabled walls (DW) | Grid convergence |
|                        | 24,084                | 15,141       | M1    | 17.755 | 10.41  | 6.181  | 3.626  |

Two levels local refinements is applied on the free-surface by imposing a global surface refinement at the undisturbed water level for the entire domain with relatively coarse grid, while a very dens grid is applied on the Kelvin pattern zone with a wider opening angle of 30° to capture more details of the generated wave by the ship, as shown in figure 3(b). The refinement for the Kelvin pattern is chosen to
provide at least 60 grid cells per wave length in \(x\)- and \(y\)-directions, chosen based on the lowest speed such that \(\Delta x=\Delta y=2\pi F r^2 L_{WL}/60\); while in \(z\)-direction, the thickness of the refinement zone is chosen to cover the stagnation wave amplitude defined by \(\zeta_{\text{max}}=0.5 U^2/g\), selected based on the maximum ship speed. The cell size in \(z\)-direction for the free-surface refinement sector is chosen as \(\Delta z = L_{WL}/1000\).

The refinement levels for the ship extremities; i.e. the forepeak, sonar dome and transom stern, are imposed higher than those for the other hull parts to provide sufficient grid resolution for the zones with higher gradients located mainly fore and aft.

![Computational grids showing: (a) Fine and coarse grids, (b) free-surface refinement and (c) a forward section.](image)

**Figure 3.** Computational grids showing: (a) Fine and coarse grids, (b) free-surface refinement and (c) a forward section.

### 3.5. Solution strategy and available resources

All the computations are performed on a High Performance Computing (HPC) machine with available 152 cores at 2.5 up to 3.3GHz, distributed between jobs as 32 cores for coarser grids and 120 for finer grids. The simulation is performed for 30 seconds to ensure adequate numerical convergence for all ship speeds. The flow is accelerated based on a steady quasi-static approach for a selected period based on the ship speed to satisfy the condition \(T_{\text{acc}} = 2L_{WL}/U\) in order to avoid any numerical instabilities in the beginning of the simulation. All cases are executed with 10 non-linear iterations and the time step \(\Delta t\) is chosen based on the ITTC formula for RANS simulations with \(k-\omega\) SST turbulence models as \(\Delta t = 0.005U/L_{WL}\). A pilot computation was performed in the beginning to ensure the solution stability and to estimate the entire convergence time. The total simulation time for each case is within 22–39 physical hours, which are definitely influenced by the grid resolution and number of cores used in the simulation.

### 4. Numerical results

Following the main scope presented in the introduction of this research, a blind approach is chosen to present first the numerical results which are summarized in the following sections divided in three levels: ship resistance, verification of the numerical results and the free-surface and wave profile. Ship resistance results are presented based on the solution approach regarding the general and towing tank approaches. The verification study is performed based on the generalized Richardson Extrapolation method summarized in the ITTC recommended procedures and guidelines for verification and validation of the ship CFD applications [8]. Finally, the free-surface is presented for wave pattern and
profile highlighting the reflection effect at the tank walls by means of wave profile and free-surface pattern. The validation of the numerical results will be covered in the final section of the results, showing a comparison between the CFD results and experimental data from the performed experiment in the UGAL tank test and the other EFD data provided in the public domain.

4.1. Resistance results

The total ship resistance predicted for the seven simulation velocities are summarized in table 4 showing a comparison between the general simulation, the active tank walls, and the disabled tank walls approaches.

| Velocity [m/s] | 1.126 | 1.351 | 1.577 | 1.802 | 2.027 | 2.252 | 2.477 |
|---------------|-------|-------|-------|-------|-------|-------|-------|
| Total Resistance [N] | General domain (GD) | 4.910 | 6.85 | 9.696 | 12.973 | 17.194 | 25.846 | 36.308 |
| | Disabled walls (DW) | 4.947 | 6.881 | 9.814 | 13.176 | 17.458 | 26.492 | 37.192 |
| | Active walls (AW) | 4.950 | 6.941 | 9.830 | 13.164 | 17.468 | 26.528 | 37.422 |
| Difference | % AW/GD | 0.808 | 1.311 | 1.363 | 1.451 | 1.569 | 2.571 | 2.977 |
| | % AW/DW | 0.061 | 0.864 | 0.163 | -0.091 | 0.057 | 0.136 | 0.347 |

The relative difference between each approach is computed based on a relative error-like relation; e.g., % \( AW/GD = 100 \times (AW-GD)/AW \), which is calculated to predict the influence of the tank walls on the numerical results, as previously described. The relative difference between a general simulation and active tank walls simulation is within 1~3%, with an ascending trend compared to the model speed. The increase in the total resistance due to the bank effect is reasonable and theoretically justified especially for high speed cases; this imposes an initial relative confidence in the obtained results. The same domain size with disabled tank walls shows a similar behavior with less than 1% difference compared to the active walls model; it also highlights the influence of the domain size on the numerical results. For \( U=1.802 \) m/s, the disabled wall represents a slightly higher resistance value than the one for the active walls approach; thought the difference is very small, it remains unphysical, which might be a result of associated numerical errors.

The computed total resistance against ship speed diagram is plotted in figure 4, while both components of the ship resistance represented in viscous and pressure resistance diagrams obtained in the numerical simulation are plotted in figure 5, showing that the pressure resistance \( R_P \) increases as the ship speed increases. This is normal for slender type ships with small block coefficient performing in high speed as the wave making resistance might exceed the frictional resistance effect.

![Figure 4. Numerically computed total ship resistance \( R_T \).](image-url)
4.2. Results verification

A grid convergence study is performed on a systematically varied four grids in order to assess the numerical errors and uncertainties based on Richardson Extrapolation method. A monotonic convergence is achieved for the total ship resistance with a descendent behavior, having the highest values for ship resistance obtained for the finest grid M1 and the lowest for the coarsest grid M4. The results computed for total ship resistance are plotted in figure 6, while the grid convergence parameters are tabulated in table 5, where the numerical errors and uncertainties are related to the finest grid solution $S_1$, assuming by default that it should produce the most accurate solution.

![Figure 6. Grid convergence results for total ship resistance.](image)

**Table 5.** Grid convergence parameters for total ship resistance.

| Convergence parameters | $r_G$ | $R_T$ | $p_G$ | $\delta_G$ | $U_\%S_1$ | $U_G\%S_1$ | $U_{SN}\%S_1$ |
|------------------------|-------|-------|-------|------------|------------|------------|----------------|
| $R_T$                  | 1.7   | 0.205 | 4.19  | 0.95       | 0.85       | 2.351      | 2.499         |

where $r_G$ represents the grid refinement ratio, $p_{th}$ and $p_G$ represent the theoretical and grid convergence order of accuracy, respectively, $R_G$ represents the grid convergence index, $\delta_G$ is the grid
error, $U_I$ is the iterative uncertainty, $U_G$ is the grid uncertainty and finally, $U_{SN}$ is the numerical uncertainties. More details can be found in [8].

4.3. Free-surface results
The free-surface and mass fraction prediction for the DTMB UGAL model are presented in figures 7 at different ship speed. The ship model in general can be classified as a medium-high speed type, which definitely affects the generated waves as can be observed in figure 7. The generated waves have a strong bow waves with slightly elongated crests and troughs through the edges of the Kelvin pattern. The stern waves and the mass fraction shows that the transom zone remains almost dry during all the simulation time which results in steeper waves generated behind the stern following the same Kelvin pattern angle and producing a rooster-tail-like pattern in the stern that can be observed clearly as the ship speed increases.
Similarly, the wave pattern for the active tank walls simulation is plotted in figure 8, showing the reflection at the boundary.
\( U = 1.351 \text{ m/s} \)

\( U = 1.802 \text{ m/s} \)

\( U = 2.027 \text{ m/s} \)

\( U = 2.252 \text{ m/s} \)

\( U = 2.477 \text{ m/s} \)

**Figure 8.** Wave pattern for active tank walls domain showing wave reflection at the wall.

The wave elevation diagram is also presented in figure 9 for all ship speeds, showing a comparison in wave elevation between the general domain and the active walls approach, where second wave can be clearly observed showing the wave elevation at the wall boundary.
5. Experimental results

In the scope of validating the numerical results, an experimental test was performed on January, 16th 2019 in the towing tank of the “Dunarea de Jos” University of Galati that has principal dimensions in length, width and depth of 45, 4 and 3 meters, respectively. The DTMB-UGAL model with the main dimensions described in table 1 was created and tested based on the Froude similarity approach according to the ITTC procedures and with respect to the possible dimensions that could be tested in the towing tank. Prior to the experiment, general procedures were followed for the test preparation including marking the draft lines, measuring the total weight of the model, preparing and mounting a specially designed wooden outfit to connect the model to the carriage and measuring water temperature. All the procedures are followed to reduce the errors as much as possible. The measured model weight had a difference of 2.5% less than the estimated weight, which was later treated by distributing extra weights to adjust the ship draft correctly after connected to the carriage. The water temperature was recorded in the beginning of the test as 15 ºC and tracked during the different stages of the test. The ship model and the carriage configuration before the experiment takes place can be visualized in figure 10.

\[
R_T = R_F + R_R;
\]

\[
C_T = \frac{R_T}{0.5 \rho U^2 S};
\]

\[
C_T = C_F + C_R \text{ & } C_R = C_T - C_F
\]

\[
C_F = \frac{0.075}{(\log Rn-2)^2}
\]

5.1. Resistance

Seven experimental conditions depending on the model speed as previously described in the numerical solution are performed. The experimental data measured in the tank test are summarized in table 6 for total ship resistance and for the ship resistance coefficients computed based on the ITTC 57 method, such that:
where $R_T$ represents the total ship resistance, $R_F$ represents the frictional resistance and $R_R$ represents the residuary resistance; similarly for the total, friction and residuary resistance coefficients, $C_T$, $C_F$ and $C_R$, respectively.

### Table 6. Total resistance measured and corresponding resistance coefficients.

| Test | Time  | $U$ [m/s] | $R_T$ [N] | $Fr$  | $C_T$ | $C_F$ | $C_R$ |
|------|-------|-----------|-----------|-------|-------|-------|-------|
| 1    | 10:40 | 1.126     | 5.177     | 0.200 | 5.308 | 3.640 | 1.667 |
| 2    | 11:30 | 1.351     | 7.357     | 0.240 | 5.240 | 3.517 | 1.723 |
| 3    | 12:30 | 1.577     | 10.379    | 0.280 | 5.425 | 3.417 | 2.008 |
| 4    | 13:30 | 1.802     | 13.580    | 0.320 | 5.436 | 3.334 | 2.102 |
| 5    | 14:30 | 2.027     | 17.534    | 0.360 | 5.547 | 3.264 | 2.284 |
| 6    | 16:35 | 2.252     | 25.668    | 0.400 | 6.579 | 3.202 | 3.377 |
| 7    | 17:40 | 2.477     | 35.109    | 0.440 | 7.438 | 3.148 | 4.290 |

#### 5.2. Free-surface

The wave configuration at the ship extremities is monitored by the help of two cameras to capture the wave in the vicinity of the ship model and any possible greening effect. The wave profiles on the ship hull for instantaneous snapshots captured during simulation are presented in figure 11. The amplitude of the wave generated close to the forward perpendicular is increasing as the ship speed increases, which tend to cove the fore area of the model during ship motion; however, no greening effect was observed during the experimental test due to the fact that the flare configuration tends to push the water outward from the hull. It is worth mentioning that the rise of flare was ignored during the model preparation; yet, no green-water on the deck was observed. The stern area confirms the observation from the numerical analysis which unveiled that the transom remains dry during the test, besides rooster-tail-like wave formation remains behind the hull.
6. Results validation
One of the most classic methods used to validate any numerical solution is to compare it directly with measured data in the towing tank or the data collected from the sea trials. The process is performed generally through a simple percentage error formula computed as 

$$
\varepsilon\% = 100 \times \frac{(EFD - CFD)}{EFD}
$$

which can give a primary indication about the accuracy of the numerical solution. However, the new
systematic verification and validation method proposes a similar approach to validate the numerical solution against experimental data based on the estimated errors and uncertainties in the numerical solution. The reason behind this approach is related to the fact that, in some cases, it is possible at the numerical errors can eliminate each other which consequently can result in a wrong impression about the numerical results, even when they have a good coherence compared to the EFD data. For this reason, the results obtained in the numerical simulations are validated with both approaches, the classic comparison and the systematic validation method recommended by the ITTC procedures for verification and validation of the CFD in ship applications [8] as previously described. In addition, for the sake of validating also the experimental data, the results measured in the towing tank of the “Dunarea de Jos” University of Galati are also compared with the measured data in the INSEAN towing tank reported in [2], based on the ITTC 57 extrapolation method using an in-house code to compare between two different, geometrically similar, ship models.

6.1. Resistance validation
Table 7 provides a quantitative simple comparison between the numerically obtained total resistance using the three different approaches and the measured data in the towing tank. The results show a reasonable agreement between the CFD results and the EFD data with an absolute average error within 4.2% which can be considered acceptable for resistance applications. The error seems to be higher for the lower ship speeds with under predicted values and lower with over prediction as the ship speed increases. This might be enhanced by imposing higher grid quality based on the lowest ship.

On the other hand, the validation parameters for total ship resistance are tabulated in table 8 after correlating the estimated uncertainties to the EFD data, where the numerical uncertainties are included in the validation process to reproduce the validation uncertainty $U_V$ based on imposing the data uncertainty $U_D$ from the experiment, such that $U_V^T = U_D^T + U^N_{SN}$. It is worth mentioning that the data uncertainty was set to the same level of the data uncertainty from the INSEAN and the IIHR towing tank as $U_D = 1$.

Table 7. Total resistance comparison between EFD and CFD results.

| $U$ [m/s] | Total Resistance $R_T$ [N] | Error |
|-----------|---------------------------|-------|
|           | CFD (GD) | CFD (DW) | CFD (AW) | EFD  | $\varepsilon_{GD-\text{EFD}}\%$ | $\varepsilon_{DW-\text{EFD}}\%$ | $\varepsilon_{AW-\text{EFD}}\%$ |
| 1.126     | 4.910    | 4.947    | 4.950    | 5.177 | 5.157 | 4.443 | 4.385 |
| 1.351     | 6.850    | 6.881    | 6.941    | 7.357 | 6.891 | 6.470 | 5.654 |
| 1.577     | 9.696    | 9.814    | 9.830    | 10.379| 6.581 | 5.444 | 5.290 |
| 1.802     | 12.973   | 13.176   | 13.164   | 13.580| 4.470 | 2.975 | 3.063 |
| 2.027     | 17.194   | 17.458   | 17.468   | 17.534| 1.939 | 0.433 | 0.376 |
| 2.252     | 25.846   | 26.492   | 26.528   | 25.668| -0.693| -3.210| -3.350|
| 2.477     | 36.308   | 37.292   | 37.422   | 35.109| -3.415| -6.218| -6.588|
|           | Average  | 4.164    | 4.170    | 4.101 |

Comparing the estimated errors from table 7 with the validation uncertainty in table 8, we can notice that the results are not validated at the validation uncertainty level since the average error $E$ for the three approaches is more than the validation uncertainty $U_V$. Nevertheless, the difference between $E$ and $U_V$ is very small.

Table 8. Validation data for total ship resistance.

| Validation | $R_T$ [%]$S_1$ | $U_D$ [%]$S_1$ | $U_{SN}$ [%]$S_1$ | $U_{SN}$ [%]$D$ | $U_D$ | $E_V$ | $U_V$ |
|------------|----------------|----------------|-------------------|----------------|-------|-------|-------|
| $R_T$      | 0.85           | 2.351          | 2.499             | 3.995          | 1.0   | 4.145 | 4.118 |
Similarly, the simple comparison between the measured force in the experiment for Model – UGAL and Model – INSEAN after the data extrapolation process, which is presented in table 9 shows a good agreement for the total resistance in both cases, except for the lowest speed. This might include various reasons for this difference in general, such as the dynamometer sensitivity to smaller forces, the data sampling errors, the test conditions, the aforementioned volumetric difference, etc. Yet, the overall agreement may be considered reasonable, taking into consideration the fact that the absolute average error is within 5.4%.

| Fr   | Model – UGAL | Model – INSEAN | ε%     |
|------|--------------|----------------|--------|
| 0.2  | 25.39212     | 22.58          | -12.4540 |
| 0.24 | 36.24414     | 33.76          | -7.358223 |
| 0.28 | 51.60957     | 48.82          | -5.713991 |
| 0.32 | 67.81115     | 65.88          | -2.931309 |
| 0.36 | 88.0354      | 88.702         | 0.7515  |
| 0.4  | 131.2834     | 136.35         | 3.715879 |
| 0.44 | 181.5826     | 191.23         | 5.044941 |

Average [ε%] 5.42

6.2. Free-surface results

In order to highlight the accuracy of the free-surface prediction, the non-dimensional free-surface profile is compared to the results obtained in the towing tank experiment of the Model – IIHR tank test reported in [9] at Fr=0.28. The comparison is brought to attention in figure 12, which shows that the CFD results have a very close agreement with the EFD data. The near-field free-surface close to the ship hull shows a good match with less than 0.5 % error, taking the average wave height comparison of 10 points; however, the far-field wave height recorded an over prediction discrepancy within 1.1%. Similarly, the wave profile fore and aft is plotted in figure 13 showing the comparison between the CFD results and the experimental data.

Figure 12. Computed free-surface contours compared to the tank test from Model – IIHR [9].
7. Conclusions

The free-surface flow and total resistance of the DTMB surface combatant ship model are predicted by the means of numerical and experimental methods and presented in the current study. The numerical approach was based on the use of a viscous flow solver and the experiment was performed on a geometrically similar model built and tested in the “Dunarea de Jos” University of Galati. The numerical and experimental analysis included seven different cases based on the ship model speed. The free-surface and total resistance results, numerically computed or experimentally measured, were presented and compared. A set of comparisons were performed to highlight the accuracy of both approaches, comparing the available results from this study with other results from the public domain.

Having in hand the results from the current study and its main objective that was presented in the introduction, we can list the following concluding remarks:

The numerical method showed to have a high quality and reliable solution that can provide a useful alternative for the initial design stage to predict the total resistance and the free-surface flow of unconventional ship types with a relatively acceptable level of accuracy and low cost;

The banking effect of the tank was highlighted and estimated to influence the total resistance of the ship with an added resistance due to the tank walls within 1~3% depending on the ship velocity;

A grid convergence study was performed to assess the numerical errors and uncertainty showed a monotonic convergence for the total ship resistance. Estimated numerical error was within the data accuracy with <1% difference compared to the finest grid $S_f$;

Free-surface and mass fraction were well predicted and presented for both cases; the general simulation case and the banking effect active tank walls case;

Experimental data were presented and compared to the well-recognized towing tanks such as INSEAN and the IIHR towing tanks, showing a good agreement especially for the high speed cases, with a level of accuracy within 95%;

The comparison between numerical and experimental data highlighted the accuracy of the numerical solution, with an accuracy level within 96% compared to the EFD data.

The validation study showed that the errors of the numerical approach were within the validation uncertainty.
Finally, the total outcome from this study can be considered sufficient especially for the initial design purposes in order to obtain the total resistance and the free-surface flow. Another feature to highlight is the relatively low cost of the numerical simulation estimated by the base of the physical computational time and the reliability of the obtained results. Nevertheless, the compatibility between the numerical and experimental approaches can provide the maximum benefit for a design engineer, especially with the high demand of more sophisticated hull forms and unconventional ship types required in the modern ship hydrodynamics.

8. References

[1] Larsson L, Stern F and Bertram V (eds.) 2002 Gothenburg 2000 a workshop on numerical ship hydrodynamics Chalmers University of Technology (Göteborg, Sweden) 320

[2] Olivieri A, Pistani F, Avanzini A and Stern F 2001 Towing Tank Experiments of Resistance and Trim, Boundary Layer, Wake, and Free-Surface Flow Around a Naval Combatant INSEAN2340 Model Iowa Institute of Hydraulic Research, the University of Iowa IIHR report No.421, 41

[3] Bekhit A S and Lungu A 2017. Annals of “Dunarea de Jos” Univ. of Galati, Fascicle XI – Ship Building 53-60

[4] Lungu A and Pacuraru F 2010 Proc. 25th IAHR Symp. Hyd. Mach. & Sys. IOP Conf. Ser.: Earth and Env. Sc. (Timisoara, Romania) 12 (012079) 709–718

[5] Bekhit A S and Lungu A 2018 Proc. Int. Conf. on Traffic and Trans. Eng., ICTTE, (Belgrade, Serbia) 325–332

[6] Deng G B, Queutey M and Visonneau M 2010 J. Hydrodynamics. 22(5) 476–481

[7] International Towing Tank Conference (ITTC) 2011, Practical guidelines for ship CFD applications. Rev.1 7.5-03-02-03 1–18

[8] International Towing Tank Conference (ITTC) 2014 Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures Rev.2 7.5-03-01-01 1–12

[9] Longo J, Shao J, Irvine M and Stern F 2007 ASME, J. Fluids Eng. 129(5) 524–540

Acknowledgment

This research is partly conducted by making use of the research computing facilities HPC-UGAL offered by the “Dunarea de Jos” University of Galati - www.hpc.ugal.ro.

This research work was supported by the project "EXPERT", financed by the Romanian Ministry of Research and Innovation, Contract no. 14PFE/17.10.2018.