Local flow assessment of the Japan Bulk Carrier using different turbulence models

A Bekhit* and F Popescu†

1“Dunarea de Jos” University of Galati, Faculty of Naval Architecture, 47 Domneasca Street, 800008, Galati, Romania
2“Dunarea de Jos” University of Galati, Faculty of Engineering, 47 Domneasca Street, 800008, Galati, Romania

E-mail: adham.bekhit@ugal.ro

Abstract. Ship resistance and powering represent the most important aspects in the initial design stage of the ship. Based on their estimation the basic milestone for selecting the main engine and the propulsion system is established. The majority of ships in the international fleet nowadays rely on the screw propeller working in the wake zone behind the ship. The wake flow of the ship has a direct impact on the propeller performance and the propulsion efficiency. Accurate prediction of the nominal and effective wake is crucially important to provide a proper understanding of the flow where the propeller will perform. From this point of view, the wake flow of the Capesize Japan Bulk Carrier (JBC) is assessed using a viscous flow Computational Fluid Dynamics (CFD) method. Numerical simulations are performed to predict the nominal and effective wake of the ship by making use of the viscous flow solver ISIS_CFD of the FINE™/Marine software 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) equations. Closure to turbulence is achieved using different turbulence models in order to investigate their accuracy in predicting the complex wake flow of the ship. Two-phase flow approach is used to model the air-water interface where the Volume of Fluid method is implemented to capture the free-surface. The results for both nominal and effective wake are assessed against the experimental data provided by the National Maritime Research Institute (NMRI) and Yokohama National University in Japan that were presented in the seventh Workshop on CFD in ship hydrodynamics (Tokyo2015). The results validation showed a reasonable agreement compared to the experimental data for both nominal and effective wake. As it was expected, some turbulence models showed to be more accurate in predicting ship wake, especially the Shear Stress Transport (K-ω SST) and Explicit Algebraic Reynolds Stress (EASM) Models. A special investigation of the flow vortices is also taken into consideration.

1. Introduction

Ship resistance and powering are the two most important aspects in the initial design stage of a ship. Their accurate prediction is crucially important, since the following aspects regarding ship operability, speed and performance may be directly or indirectly influenced by their estimation. Once the ship powering is predicted, the following stage is to select the main engine and propulsion system. The vast majority of commercial ships in the international fleet are relying on the rotating propeller that is working in the wake zone behind the ship. This imposes another challenge for the naval architect to
predict accurately the flow configurations in the propeller working zone in order to avoid any undesired problems such as noise, vibration, cavitation, and/or loss of propulsion. Heading from this point of view, the importance of the wake flow analysis is augmented and rather becomes extremely crucial for the high-block coefficient ships, as in case of tankers and bulk carriers where a significant flow separation and vortex formations are expected to occur. For this reason, this study investigates the local flow of the Japan Bulk Carrier (JBC hereafter) ship on three different levels: the first is for the nominal wake, i.e. the ship is not equipped with the propeller; the second is for the propeller open water (POW) condition and finally the effective wake were the propeller is connected and working behind the ship. The study is based on the Computational Fluid Dynamic (CFD) method that becomes more popular in the ship hydrodynamic field as the quality of the predicted results in the recent two decades are competing head to head with the classic experimental method from the accuracy point of view. Besides, one may consider an advantage of the CFD compared to the experimental method regarding its flexibility and recently the associated low cost, which resulted after the significant development in the numerical modelling and the associated revolutionary progress in the computational power. These improvements resulted in less physical modelling assumptions, the possibility to use finer grids and less turnaround simulation time. Turbulence modelling has played a significant role in predicting the flow configurations in CFD for ship hydrodynamics. Implementation of turbulence models such as Reynold’s-Averaged Navier-Stokes (RANS), Large Eddy Simulation (LES) and Hybrid RANS-LES models increased the reliability of CFD and helped understanding more sophisticated phenomena such as flow separation and hull-fluid interactions. In this study, the RANS method are applied to solve and predict the wake flow of the JBC ship model in a resistance, open water propeller, and self-propulsion simulations. Closure to turbulence is achieved by applying different turbulence models that are available in the ISIS-CFD solver used in this study, such as for resistance simulation the Spalart-Allmaras, K-ω, the baseline K-ω BL, the shear stress transport K-ω SST, the enhanced K-ω SST 2003 and finally, the Explicit Algebraic Stress Model (EASM) were used. While for the POW and self-propulsion simulations, the EASM and the DES models were applied. More details about the turbulence models can be found in [1-3].

2. Geometry and analysis conditions
The JBC (Japan Bulk Carrier) is a capesize bulk carrier equipped with a stern duct that plays the role of a wake equalizing duct or Energy Saving Device (ESD). National Maritime Research Institute (NMRI), Yokohama National University and Ship Building Research Centre of Japan (SRC) are jointly involved in the design of the ship hull, the duct, propeller and rudder. The full scale ship does not exist; however, a geometrically similar model was built and tested in the towing tank experiments planned and performed at NMRI, SRC and Osaka University. The test included the resistance tests, self-propulsion tests and PIV measurements of stern flow fields and the test results were presented for CFD validation and verification in the Tokyo 2015 Workshop on CFD in ship Hydrodynamics [4, 5]. The geometry of the 7.0 m ship model and the main dimensions for ship, duct and propeller are given in figure 1 and table 1, respectively.

The analysis conditions include three simulations: the first is dedicated for studying the local flow of the ship with and without the wake equalizing duct in resistance estimation, the second is for the propeller in the open water condition, while the final is for the self-propulsion condition for the ship with and without the duct using two different approaches; the infinite blade actuator disk and the 3D rotating propeller based on the sliding grid approach. More details regarding the two models were previously presented by the author in [6]. The difference between the previous study and the current is regarding the turbulence models used for the analysis and the use of finer grid to capture more details in the wake zone.
Figure 1. JBC model geometry highlighting: (a) stern, (b) fore, (c) duct, (d) propeller and (e) rudder.

Table 1. Principal particulars for the JBC ship model, duct and propeller.

| Particulars                               | Unit   | Value          |
|-------------------------------------------|--------|----------------|
| Length between Perpendiculars ($L_{pp}$)  | [m]    | 7.0            |
| Beam ($B$)                                 | [m]    | 1.125          |
| Depth ($D$)                                | [m]    | 0.625          |
| Draft ($T$)                                | [m]    | 0.4125         |
| Volumetric Displacement ($V$)              | [m$^3$]| 2.787          |
| Wetted Surface Area ($S_0$) (without ESD) | [m$^2$]| 12.225         |
| Wetted Surface Area ($S_0$) (with ESD)    | [m$^2$]| 12.272         |
| Block Coefficient ($C_B$)                  | [-]    | 0.858          |
| LCB ($%L_{pp}$), fwd+                      | [-]    | -2.548         |
| Duct Outlet Diameter (0.55$D_p$)          | [m]    | 0.11165        |
| Duct Cord Length (0.3$D_p$)               | [m]    | 0.0609         |
| Duct Angle of Attack                      |        | 20             |
| Duct Foil Section                         |        | NACA4420       |
| Propeller Diameter, $D_p$                 | [m]    | 0.203          |
| Propeller expanded area ratio, $A_S/A_0$  |        | 0.5            |
| Number of blades, $Z$                     |        | 5              |
| Propeller rotation direction             |        | Clockwise      |

3. Numerical Simulation Framework

3.1. Numerical solver

The numerical simulations performed in this study are carried out using the ISIS-CFD solver of the software FINETM/Marine provided under the NUMECA suite. 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) [7]. The solver implies a face-based spatial discretization that constructs the fluxes face by face, which makes it optimum 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 discretising 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
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:

\begin{equation}
\frac{\partial (\rho \bar{u}_i)}{\partial x_i} = 0
\end{equation}

\begin{equation}
\frac{\partial (\rho \bar{u}_i \bar{u}_j + \rho \bar{u}_i \bar{u}_j)}{\partial x_i} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \bar{t}_{ij}}{\partial x_j}
\end{equation}

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 \( \bar{t}_{ij} \) is the mean viscous stress tensor components for Newtonian fluid under the incompressible flow assumption, and it can be expressed as

\begin{equation}
\bar{t}_{ij} = \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)
\end{equation}

3.3. Computational domain and boundary conditions

For resistance, 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 in \((x-y-z)\) directions are set as \((5.0L_{PP}-2.0L_{PP}-2.0L_{PP})\) as illustrated in figure 2(a). As for the self-propulsion simulation, the same dimensions in resistance are typically implied except for the fact that the symmetry condition is not applicable here and the full ship is introduced in the domain. For this reason, the lateral dimension of the self-propulsion domain in \(y\)-direction is set as \(4.0L_{PP}\), as figure 2(b) bears out. The undisturbed free-surface level is set at the design draft of the ship located at \(0.4125m\) from the ship baseline. A global earth-fixed reference frame is set for the resistance and self-propulsion simulations at the point \((0, 0, and 0)\) which results from the intersection of the ship baseline with the aft perpendicular (A.P). The boundary conditions are set for the boundaries such that: the flow inlet is set at \(2.0L_{PP}\) from the reference point, while the outlet is set at \(3.0L_{PP}\) from the reference. The far field side boundaries are set at \(2.0L_{PP}\) from the ship centreline and finally, the prescribed pressure top and bottom boundaries are located at \(0.5L_{PP}\) and \(1.5L_{PP}\) from the undisturbed free-surface level, respectively. Symmetry boundary is set at the ship centreline only in the resistance simulation.

On the other hand, for the POW simulation, the domain is having a cylindrical configuration whose dimensions are \(6.0\) times the propeller diameter \((D_p)\) in the circumferential direction and \(8.0\) times the propeller diameter in the longitudinal direction as illustrated in figure 2(c). A global reference frame is set at the point \((0, 0 and 0)\) which results from the intersection of the propeller generator line and the shaft axis. The velocity inlet condition is set at \(2.0L_{PP}\) from the reference point, while the outlet is set at \(3.0L_{PP}\) from the reference. No slip condition is chosen for the entire ship hull, except the deck where it is assumed to remain in air during the simulation; hence it was set to a slip condition. Besides, for the self-propulsion simulation, also the propeller shaft is set to a slip condition to avoid any discontinuity at the interface zone between the hull and propeller for the sliding grid.

On the other hand, for the POW simulation, the domain is having a cylindrical configuration whose dimensions are \(6.0\) times the propeller diameter \((D_p)\) in the circumferential direction and \(8.0\) times the propeller diameter in the longitudinal direction as illustrated in figure 2(c). A global reference frame is set at the point \((0, 0 and 0)\) which results from the intersection of the propeller generator line and the shaft axis. The velocity inlet condition is set at the inflow boundary located at \(2.0D_p\) upstream. The frozen pressure condition is applied on the exit boundary located at \(6.0D_p\) downstream. The velocity
inlet condition is also applied on the domain cylindrical wall. For the sake of capturing more details in
the wake zone, a cylindrical refinement zone is added in the vicinity of the propeller extended from
0.25\(D_p\) upstream and 4.0\(D_p\) downstream, while its diameter is set to 1.5\(D_p\) distributed evenly above
and underneath the propeller shaft axis. No slip condition is applied on the propeller blades, boss and
cap, while the slip condition is used for the propeller shaft.

![Computational domains and boundary conditions](image)

Figure 2. Computational domains and boundary conditions for the three simulation cases (a) resistance, (b) self-propulsion, (c) POW.

3.4. Discretization Grid

The grids are generated for the three simulation conditions using the automatic unstructured
hexahedral grid generator HEXPRESS\textsuperscript{TM} provided in the Fine\textsuperscript{TM}/MARINE package. In order to
capture more details of the flow in the zones of concern, special set of local refinement zones were
created in the free-surface, forward and more intensely in the wake zone of the ship and propeller. The
number of grid cells used for discretization is summarized in table 2, while the grid configurations are
presented in figure 3 showing the forward, aft and free-surface refinement zones for the resistance
simulation in figure 3(a), the aft zone, propeller and duct in the self-propulsion simulation in figure
3(b) and finally the propeller and wake refinement zone in figure 3(c).

![Discretization Grid](image)

Table 2. Number of grid cells for simulation conditions and approaches.

| Simulation               | Ship Resistance | Self-Propulsion |
|--------------------------|-----------------|-----------------|
|                          | JBC w/o ESD     | JBC w. ESD     | A.D.\textsuperscript{a} w/o. ESD | A.D. w. ESD | S.G.\textsuperscript{b} w/o. ESD | S.G. w. ESD | POW |
| Number of grid cells (M) | 23.563          | 25.077          | 19.883          | 21.750       | 25.846                      | 27.353       | 34.633 |

\textsuperscript{a}A.D. = The Actuator Disk approach, \textsuperscript{b}S.G. = Sliding Grid approach.
3.5. Solution strategy and available resources

All the computations are performed on a High Performance Computing (HPC) machine with available 120 cores at 2.5 up to 3.3 GHz. For resistance prediction, the simulation is performed for 30 seconds to ensure adequate numerical convergence for resistance and vertical motions. The flow is accelerated based on a steady quasi-static approach for a selected period based on the ship speed to satisfy the condition $T_{acc} = 2L_{pp}/U$ in order to avoid any numerical instabilities in the beginning of the simulation. 10 non-linear iterations are used with a time step $\Delta t$ is chosen based on the ITTC formula [9] for RANSE simulations with two-equations turbulence models as $\Delta t = 0.005U/L_{pp}$ also kept the same for the one-equation Spalart-Allmaras model, while for the EASM model the time step is set as $\Delta t = 0.001L_{pp}/U$. In the self-propulsion the simulation is performed in two levels; in the first, the same strategy used in the resistance is applied for the first run to stabilize the ship resistance and vertical motion with only 8 non-linear iterations and also for 30 seconds, while in the second run, unsteady simulation with 12 non-linear iterations is used with a smaller time step to stabilize the propeller thrust that should provide 200 time-steps/propeller rotation.

As for the POW simulation, unsteady approach with a rotating frame technique is applied to reduce the simulation time, with a time step set to provide 100 time steps/propeller rotation and only 5 non-linear iteration and total simulation time is 6 seconds that was increased to 10 for the low advance

Figure 3. Discretization grids for simulation cases: (a) resistance (b) self-propulsion, and (c) POW.
velocity $J=0.1\text{–}0.3$, where $J=U/nD_p$ (to stabilize the flow). For the DES simulation in self-propulsion and POW cases, the time step was chosen with an order of magnitude with $10^{-1}$ less than that used in the EASM model. The total simulation time for each case is within 16–118 physical hours, which are definitely influenced by the grid resolution and the corresponding time-step.

4. Results and Discussions

4.1. Resistance simulation
For the resistance simulation, corresponding to the tank test where the local flow velocity contours were presented for three measuring sections as reported in the Tokyo 2015 workshop named as sections S2, S4 and S7 which are measured at distances 0.2625m, 0.11m and 0.0m, respectively from the A.P. as illustrated in figure 4. The numerical results for the axial velocity contours in $x$- direction are also presented for the same section in figures 5, 6 and 7 showing the computed results after 30 seconds of simulation using different turbulence models and arranged based on the accuracy of the obtained results compared to the experimental data (EFD).

Figure 4. PIV measuring sections.

Figure 5. Section S2 EFD and CFD results for the axial velocity contours computed at $T=30\text{ s}$ for ship without ESD using different turbulence models.
Following the flow configuration of the obtained and measured results, one can observe that the flow is characterized by the formation of an intense stern bilge vortex which leads to a noticeable distortion of the axial velocity contours at the measuring sections. This distortion results from the transport of low momentum fluid from the vicinity of the hull to the centre of the flow field under the effect of an intense longitudinal vortex. Besides, it can also be observed the existence of a secondary counter-rotating vortex close to the vertical plane of symmetry. Which leads to the formation of the so-called hook-shape of the velocity contours which is clearly visible in the towing tank data and the CFD results.

![Figure 6](image1.png)

**Figure 6.** Section S4 EFD and CFD results for the axial velocity contours computed at $T=30$ s for ship without ESD using different turbulence models.

![Figure 7](image2.png)

**Figure 7.** Section S7 EFD and CFD results for the axial velocity contours computed at $T=30$ s for ship without ESD using different turbulence models.
The significant challenge in the prediction of the local flow of this type of ships is to capture accurately this deformation of the velocity contours as it will be important since this phenomena happens in the vicinity of the propeller plane and it will significantly affect the propeller performance prediction in the further study in the self-propulsion condition.

The obtained results show the accuracy of the turbulence models to predict the hook-shape velocity contours with different levels. For example, the EASM, K-ω SST models showed to be the most accurate in predicting the velocity contours compared to the classic K-ω and Spalart-Allmaras models where they came in the last places. Nevertheless, it can be said that even with the under predicted velocity contours, the hook-shape was still captured with a sufficient level of accuracy for design purposes, especially compared to the previously used methods such as the methodical series and empirical based approaches where the local flow was totally not considered.

Having the fact that the EASM model gave the best obtained results, the local velocity contours configuration of the case when the ship is equipped with the ESD is represented in figure 8, showing a promising agreement between the computed results and the provided EFD data.

Figure 8. Comparison between the streamwise velocity contours measured and computed at \( T=30 \) s using EASM turbulence model for ship with ESD for sections S2, S4 and S7.

In order to take a closer look on the flow configuration and the vortices development in the wake zone of the hull, another proposal by the Workshop was given as a comparative data for the vortex formation based on the \( Q^* \)-second invariant of the axial velocity that may help visualizing the vortex formation in the stern zone, as it is illustrated in figure 9.

Figure 9. Workshop data for the second invariant iso-surface \( Q^* = 25 \) colored by helicity and the corresponding CFD results computed at \( T=30 \) s using EASM model.
Though the comparison between the CFD results and the provided data shows an underpredicted vortices structure; yet, the main vortices can be observed clearly in the vicinity of the ship boss and the propeller zone. The flow separation starts early at the bilge and transferred to the centreline as described before. Two counter-rotating vortices from the port and starboard sides can be clearly visualized and also underneath that, contra-rotating small size vortex tubes can be noticed. One more drawback in the obtained data is regarded to the choice of the length of the stern refinement zone by the author, which seems to be insufficient to capture the continuous vortices as in the provided data. However, this is with less important as the main configuration of the vortices were well predicted.

4.2. Propeller Open Water

The propeller open water test was repeated numerically for 8 advance ratio $J$ to investigate the capability of the CFD method in predicting the local flow of the propeller in the POW condition. The complete set of results including grid convergence study and more details was reported by the author in [10] based on the EASM turbulence model. This study includes the DES simulation for $J=0.2$ in order to capture more details in the flow.

The axial velocity contours computed at 10 seconds in the propeller centreline plan in the longitudinal direction can be visualized in figure 10(a) where the velocity contours are showing a maximum value at the trace of the tip vortices with a significant fluctuation downstream around the core of the vortices. The corresponding pressure is also presented in figure 10(b) showing the pressure peaks in the vicinity of the propeller and downstream again with respect to the formed vortices. The turbulent viscosity and turbulent kinetic energy are presented in figure 10(c) and (d), respectively. All the four components can represent the formation of the wake flow vortices as three significant vortices can be observed; the first is generated at the propeller blade tip, the second is at the propeller blade root and finally the third is at the propeller hub, which can also be observed completely in figure 11.

Figure 10. Wake flow configuration for the POW at $J=0.2$ showing: (a) axial velocity contours, (b) pressure, (c) turbulent viscosity and (d) TKE.
It can also be observed a significant fluctuation of the hub vortices that is apparently resulting from the interaction between the high momentum flow transferred downstream from the propeller vicinity to the low momentum flow in the far filed. This can be visualized clearly at the hub vortices in the downstream zone before the exit. It can also be noticed that the propeller tip and root vortices are combined together due to the viscous effect generating a thick vortex tube before it is transferred to the flow exit. It is worth mentioning that the grid quality should be enhanced for capturing more details in the wake zone for this analysis.

**Figure 11.** Second invariant $Q^* = 50$ computed at $10s$, $J = 0.2$ and colored by nondimensional helicity.

### 4.3. Self-propulsion simulation

The self-propulsion simulation was performed based on two different approaches as previously described; the first is based on the actuator disk (A.D.) method which represents a quick and accurate solution for predicting the self-propulsion performance of the ship and propeller without a need of propeller modelling, besides the fact that the required simulation time and effort is significantly reduced. Unfortunately, it does not provide full details about propeller blade loading and cavitation inception, etc. The method was qualitatively and quantitatively assessed and presented by the author in [6] showing a reliable and accurate results.

In order to perform the self-propulsion using the actuator disk method, two levels of simulation must be performed, the first is to obtain the nominal wake by a standard resistance simulation, and then the actuator disk is introduced in the simulation to provide the prediction of the effective wake. Having the data from the resistance test and introducing the open water data in the solver, the self-propulsion coefficients and characteristics can be obtained based on the thrust or torque identity method.

The EFD data for the axial velocity contours measured at the section located in the propeller reference plan is plotted in figure 12 for the ship with and without the ESD at the nominal and effective wake condition, along with the computed results for the three relative velocity components $V_x$, $V_y$, $V_\theta$ for the actuator disk also located at the propeller reference plan at a distance 0.10148 m from the A.P. The comparison shows that the obtained results have a well congruence with the EFD data as figure 12 bears out. The measured and the computed results show the effect of the ESD on the flow configuration at the propeller vicinity, which indicate that the axial velocity contours are having more circular configuration compared to the ship without ESD. This means that the flow entering the propeller is more uniform, which can positively influence and enhance the propulsion efficiency. At the core of the velocity contours, a slight discrepancy can be observed which is related to the fact that in this study, the propeller hub was not connected to the ship boss during the self-propulsion simulation with the A.D. approach. Nevertheless, the effect is insignificant and the computed velocity contours resembles well with EFD data.

Similarly, the results for the axial velocity contours for the ship without ESD computed based on the 3D modelled propeller with the help of the sliding grid technique can be observed in figure 13 for the measuring sections S4 and S7, which means before and after the propeller, respectively. The comparison also reveals the fact that the computed results are within a good agreement with the measured data.
Figure 12. CFD results for nominal (a and b) and effective (c and d) axial velocity contours computed at $T=30$ s for ship with and without ESD using actuator disk method based on K-ω SST model.

Few years after the workshop was held, the rudder configuration was uploaded on the official Workshop website. This was a good opportunity to study the self-propulsion simulation for the ship with rudder as a primary step for a further research interest regarding the propeller rudder interaction. The local flow stream-wise velocity contours were brought to attention in figure 14 as a comparison between the ship with and without rudder. It was observed in the study that the rudder influences the propeller performance by enhancing the thrust forces due to the interference with the propeller as a part from the tangential velocity is transformed into an axial velocity due to the existence of rudder, which resulted in an enhanced thrust outcome. Also, the $Q^*$ second invariant is also plotted in figure 14 showing the previously mentioned propeller tip, root and hub vortices for the ship without rudder and one more vortex formation is resulting from the rudder lower trailing edge separation. More details about this study can be found in [11].
Figure 13. Comparison between the streamwise velocity contours measured and computed at $T'=33$ s using EASM turbulence model for ship with ESD for sections S4 and S7.

Figure 14. Comparison between the stream-wise velocity contours computed at $T'=30$ s using DES turbulence model for ship without ESD and with/without rudder and the second invariant $Q^*=50$ colored by nondimensional helicity.
5. Conclusions
The local flow assessment for the Japan Bulk Carrier JBC ship model of a 7.0m length was computed based on a viscous CFD method, presented and compared to the EFD data provided in the Workshop on CFD in ship hydrodynamics held in Tokyo 2015. The comparison was made for three different conditions including the resistance, POW and the self-propulsion conditions using different turbulence models. The assessment of the CFD results showed a general good agreement with the experimental data and showed the capability of the CFD method to capture accurately the challenging local flow of a high-block coefficient ship with significant flow separation. The ship was tested on two levels with respect to the existence or the absence of the ESD which is aimed at working as a wake equalizing duct in order to enhance the flow heading the propeller. The effect of the duct was properly calculated and presented in the study. Having the main objective of the current study and its outcome, we may conclude the following main points:

- In the resistance simulation, the turbulence models used for computing and predicting the flow separation showed to be sufficiently accurate with different level of reliability. EASM model showed to have the highest agreement with the EFD data, while Spallart-Allmaras showed to have the lowest. It is still worth mentioning that the simulation time for EASM is within 3 times longer than the other turbulence models. K-ω SST models showed a good balance between accuracy and simulation time.
- The stern vortices were computed and presented compared to the data provided in the Workshop showing a good agreement in predicting the vortical structures formations.
- In the POW simulation, the flow configuration was computed and presented showing the flow development in the propeller vicinity and in the downstream. The results captured accurately the formation of tip, root, and hub vortices and their development and interaction during the simulation time.
- The Self-propulsion test was presented based on both A.D. method and fully discretized propeller. The advantage of the A.D. over the fully discretized propeller is regarding the low simulation time and the simple application. The disadvantage is regarding the fact that the propeller blade loading and cavitation occurrence cannot be predicted based on this model. The results obtained using the A.D. were within a satisfying agreement compared to EFD data. As for the fully discretized propeller, the results were in a good match with the EFD data.

In general, the study was successful in predicting the local flow and the flow separation for the JBC ship model based on various turbulence models. Further investigation is planned for ship-propeller-rudder interaction in manoeuvring simulations continuing the previous research effort represented in [12-15].

6. References
[1] Wilcox D C 2006 Turbulence Modeling for CFD (3rd Ed.) (USA: D C W Industries).
[2] Ferziger J H, Peric M 2002 Computational Methods for Fluid Dynamics (3rd Ed.) (NY: Springer).
[3] Menter, F R, Kuntz M, and Langtry R, 2003 Ten Years of Industrial Experience with the SST Turbulence Model, Turbulence, Heat and Mass Transfer 4, ed: Hanjalic K, Nagano K, and Tummers M, Begell House, Inc., pp. 625 - 632.
[4] Tokyo 2015 A Workshop on CFD in Ship Hydrodynamics https://t2015.nmri.go.jp/ accessed on 18.03.2021.
[5] Hirata N 2015 http://www.t2015.nmri.go.jp/Presentations/Day1-AM2-JBC-TestData1-Hirata.pdf accessed on 18.03.2021.
[6] Bekhit A S 2018 IOP Conf. Ser.: Mater. Sci. Eng. 400 042004 1–11.
[7] Deng G B, Queutey M and Visonneau M 2010 J. Hydrodynamics. 22(5) 476–481.
[8] Queutey P, Deng G B, Wackers J, Guilmineau E, Leroyer A and Visonneau M 2012 J. Ship Technology Research 59(2) 44-58.
[9] International Towing Tank Conference (ITTC) 2011, *Practical guidelines for ship CFD applications*. Rev.1 7.5-03-02-03 1–18
[10] Bekhit A S, Lungu A 2019, AIP Conf. Proc. **2116**, 450007 (2019)
[11] Bekhit A S, Pacuraru F, Pacuraru S 2020, AIP Conf. Proc. **2293**, 420091 (2020)
[12] Pacuraru F, Lungu A 2009, AIP Conf. Proc. **1281** (1), 111-114
[13] Pacuraru F, Lungu A, Ungureanu C, Marcu O 2010, IOP Conf. Ser.: Earth and Env. Sc. **12**(1), 012032
[14] Pacuraru F, Lungu A, Marcu O 2011, AIP Conf. Proc. **1389**(1), 191-194

**Acknowledgement**

This work was carried out in the framework of the research project DREAM (Dynamics of the REsources and technological Advance in harvesting Marine renewable energy), supported by the Romanian Executive Agency for Higher Education, Research, Development and Innovation Funding – UEFISCDI, grant number PN-III-P4-ID-PCE-2020-0008.