RANS Simulation of KVLCC2 using Simple Body-Force Propeller Model With Rudder and Without Rudder

by Yan Naing Win*, Member
Keisuke Akamatsu**, Member
Frederick Stern****, Member

Ping-Chen Wu**, Member
Hiroshi Okawa***, Member
Yasuyuki Toda ***, Member

Summary

The viscous flow simulation is carried out around the KVLCC2 tanker model by the simple and effective way using a new body-force distribution model for the propeller-hull and propeller-hull-rudder interaction. The simple body-force model based on quasi-steady blade element theory is coupled with the Reynolds averaged Navier-Stokes (RANS) code CFDSHIP-IOWA. The captive tests in the trimmed condition and even-keel condition are computed with rudder and without rudder. The computational condition is set up according to the experiments in Osaka University towing tank and the output flow fields are analyzed in details especially in the wake field around the rudder and propeller. The computational results are not only validated with the experiments but also compared with the real propeller computation and other body-force models in order to find out the advantage of the current method. Summarizing the results, the present study could provide the complicated wake field patterns behind the tanker hull form which are as close as the experiment by simply using the new body-force distribution model and the current method can predict the wake field superior to the other body-force models.

1. Introduction

The study of propeller-hull and propeller-hull-rudder interaction is important to predict the efficiency of the propeller as well as its influences on the resistance of the ship hull. By the Computational Fluid Dynamics (CFD) method, the viscous flow computation of the ship hull is normally coupled with some propeller programs either by viscous method or by potential flow method. Compared to the high cost and complexity of viscous flow method and to some well-known body-force models, the propeller model utilized in this paper has been proved as the easier and effective body-force model in predicting the complicated ship wake flow by Win et al.1)2) The advantages of the propeller model which is computed by the quasi-steady blade element theory can be summarized as; (1) the coding is very simple (2) no special type of grids is required and any grid can be used to compute the body forces (3) it is unnecessary to extract the effective wake for the inflow to the propeller blade from the total flow field. The effective wake usually is required in potential theory based body-force models. Moreover, the capabilities for dealing with side-flow as well as various propeller loads make this method much effective to be utilized in the propulsion research field1).

The purpose of this study is to show the capability of the propeller model in the tanker hull form. This research, therefore, will mainly focus on the propeller-hull and propeller-hull-rudder interaction of KVLCC2 by coupling the Reynolds averaged Navier-Stokes (RANS) code CFDSHIP-IOWA with the aforementioned propeller model. Especially, the rudder flow field is the main concern since it is not considered in the previous work1)2). The propeller hub vortex and rudder flow in propeller wake are well-resolved by the propeller model. The present work will be useful for the future work to investigate the energy-saving device on the rudder, such as bulb and fins.

The computational results are validated against the Experimental Fluid Dynamics (EFD) data which has been carried out in Osaka University towing tank. The two cases are mainly computed: with-rudder and without-rudder case. The wake field behind the ship without rudder is much easier to be analyzed. The propeller-hull-rudder interaction and rudder flow are investigated by the with-rudder case. The results are also compared with the results of real propeller geometry computation and some body-force models to understand the prediction level of the current model and the pros and cons.

2. Background

2.1 CFD Method

The numerical code utilized in this study is the RANS solver CFDSHIP-IOWA which is an unsteady single-phase level-set solver with dynamic overset grids3). It is designed mainly for ship applications using either absolute or relative inertial non-orthogonal curvilinear coordinate system for arbitrary moving but non-deforming control volumes. Captive, semi-captive, and full 6-DOF capabilities for multi-objects with parent/child hierarchy are available but 6-DOF (Degree of
formulations of the propeller model has already been explained by Win et al. and Tokgoz et al.

2.3 KVLCC2 geometry
The main particulars of KVLCC2 for the full scale, Osaka University (OU) model and INSEAN (Istituto Nazionale per Studi ed Esperienze di Architettura Navale) model scales are listed in Table 1. The OU model is used for the experimental work with the scale ratio 1/100 and the actual propeller model computation used the INSEAN particulars with NMRI (National Maritime Research Institute) propeller diameter. The geometry of KVLCC2 with propeller and rudder is shown in Fig. 1.

![Fig. 1 Geometry of KVLCC2 (Hull, Propeller, Rudder)](image1)

![Fig. 2 Trim condition of KVLCC2 experiment](image2)

2.4 Computational Domain, Grids and Boundary Conditions
The computational mesh is generated according to the non-dimensional coordinates of the ship and propeller as shown in Table 1 by using grid generation software Gridgen. The ship hull is trimmed by bow-down 0.13 degree provided by the experiment as illustrated in Fig. 2 where Le is length between perpendicular, zf is the forward draft change, za is the aft draft change, C.F is the center of floatation, FP is the forward perpendicular and AP, aft perpendicular. The experiment is carried out at the full load condition for sinkage and trim condition. The Froude number is 0.142 and the carriage speed is set at 0.795 ms\(^{-1}\) so that the Reynolds number utilized is 2.05 \times 10^{6}. On the other hand, the grid for comparing with the real propeller computation (section 3.6) is generated by INSEAN data in even-keel condition.

The domain ranges \(-2.5<X<0.5, -1<Y<1, -1<Z<0.22\) where \((X,Y,Z)\) are non-dimensional Cartesian coordinates by the ship length where \(X\) is in axial (ship length) direction positive to the downward of the ship, \(Y\) is positive in starboard side direction and
Z is in upper direction, respectively with the origin at the intersection of the center plane at FP and at still water surface.

The computational domain is illustrated in Fig. 3 with several blocks overset within the domain. The whole domain is composed of 9 blocks for without-rudder case (12.9 million grids) and 11 blocks for with-rudder case (13.1 million grids) in the H-O topology which are listed in Table 2 with corresponding grid size. The boundary layer grids for port and starboard sides are for capturing the detailed boundary layer flow along the hull. The tail blocks are used to describe the tanker stern part in detail to resolve wake field in high accuracy. The hub blocks are used to describe the tanker stern part in detail to resolve wake field in high accuracy. The hub blocks are used to capture the accelerated flow downstream of the propeller. The outermost background grid is generated to deal with the free surface and far field boundary conditions. All of these blocks are generated separately and then combined and assembled by SUGGAR which determines the static overset structure among the blocks. To integrate the area, forces and moments properly, USURP code is performed to give weights on the cells overlapping on the solid surface.

The boundary conditions are as shown in Table 3 with the mathematical expression in Table 3 where \( (U, V, W) \) are velocity components non-dimensionalized by ship speed \( U_0 \), \( \rho \) is the transient pressure, \( k \) is kinetic energy of turbulence, \( \omega \) is dissipation of turbulence kinetic energy, \( f_s \) in \( k_{fs} \) and \( \omega_{fs} \) refers to the free stream and rotational effect is set on each grid point on the hub surface at the same revolution rate with the propeller and implemented into the code as user defined boundary condition. This condition is illustrated in Fig. 4 for one X-section of the hub. The axial velocity \( U \) will be zero according to the no-slip condition but there will be \( V \) and \( W \) according to the rotation. The values of \( V \) and \( W \), on the hub surface can be computed according to Eq. (1) and Eq. (2) where \( r_{hub} \) is the hub radius at the grid point on the hub surface and \( n \) is the propeller revolution rate.

\[
V = 2\pi r_{hub} n \sin \theta \\
W = -2\pi r_{hub} n \cos \theta
\]  

Table 3 Boundary conditions

| Type               | I    | J    | K    | Total       |
|--------------------|------|------|------|-------------|
| Boundary Layer (Starboard) | O    | 154  | 50   | 144         |
| Boundary Layer (Port)     | O    | 154  | 50   | 144         |
| Tail part (Starboard)     | O    | 55   | 50   | 40          |
| Tail part (Port)          | O    | 55   | 50   | 40          |
| Hub (Starboard)           | O    | 55   | 50   | 40          |
| Hub (Port)                | O    | 55   | 50   | 40          |
| Propeller block           | O    | 27   | 126  | 101         |
| Rudder (Starboard)        | O    | 44   | 43   | 70          |
| Rudder (Port)             | O    | 44   | 43   | 70          |
| Wake Refinement           | H    | 281  | 144  | 151         |
| Background               | H    | 216  | 121  | 151         |

3 Computational Results and Discussions

3.1 Propulsive Quantities

For both with- and without-rudder cases, two steps of computations are carried out: towing condition first and then self-propulsion condition with the propeller model. The computation of towing condition is essential in order to validate the nominal wake patterns first and later coupling the propeller model to reach converged self-propulsion condition. This
procedure can prevent the numerical divergence. Also, in the coupling with propeller model, a relaxation factor is used for the body-force terms in the momentum equation for preventing divergence problem as well. The propeller revolution rates are obtained by the self-propulsion test in the experiment with the full load condition: 16.5 rps for model point, and 11.1 rps for ship point (considering the skin friction correction for the full scale ship). The final converged thrust and torque coefficients for model point are listed in Table 4 by comparing between CFD and EFD for both with- and without-rudder conditions. The computations give very close agreements to the experiments. The with-rudder case provides a little bit higher thrust coefficient than without-rudder case. The thrust deduction factor, which expresses the effect of the suction of the propeller on the hull, is defined by Eq. (3) where \( R_T \) is the bare hull resistance, \( R_T \) is the resistance with propeller and \( T \) is the thrust of the propeller. The thrust deduction factor (1-\( t \)) of the computation gives as 0.794 for with-rudder case and 0.855 for without-rudder case. The effective wake factor is calculated as in Eq. (4), where \( w \) is the wake fraction, \( J_a \) and \( J_s \) are the advanced coefficients defined as \( J_a = U_a / n D \) and \( J_s = U_s / n D \) where \( U_a \) is advanced speed and \( U_s \) is ship speed. (1-\( w \)) is determined on the basis of a thrust identity where \( J_s \) is obtained from the open water characteristics curve corresponding to the thrust measured when the propeller is running behind the ship at \( J_s = 0.489 \). As KVLCC2 is tanker shape and the wake field is wide so that the effective wake factor is low. Based on \( K_T = -0.241 J_a - 0.2998 J_s + 0.2705 \) (EFD open water characteristics curve), CFD (1-\( w \))=0.424 for with-rudder case and 0.453 for without-rudder case.

\[
t = \frac{R_T - R_{T0}}{T} \\
(1 - w) = \frac{J_a}{J_s}
\]

Table 4 Thrust and torque coefficients comparison

|     | \( K_T \) | \( K_Q \) |
|-----|-----------|-----------|
| **Without-rudder case** | | |
| CFD | 0.198 | 0.0221 |
| EFD | 0.195 | 0.0266 |
| **With-rudder case** | | |
| CFD | 0.203 | 0.0222 |
| EFD | 0.198 | 0.0228 |



3.2 Nominal wake Analysis

Understanding the flow natures of the wake of ships is important for ship design and especially, in this paper, the nominal flow field at the propeller plane is very important to evaluate the capability of the propeller model. The nominal wake is therefore extracted at the propeller plane and this kind of assessment, analyses and comparisons has been carried out with different computational results provided by several institutes as well as with the experimental results. However, in all of those works, the propeller is believed to be located at \( X = 0.9825 \) but in the current case, the measurement of the OU model gives the propeller position to be at \( X = 0.98 \). Axial velocity contour and cross-flow vectors at this station is illustrated in Fig.5 (a) for CFD, (b) for PIV measurement and their comparison in (c).

![Fig. 5 Wake flow field at the propeller plane X=0.98](a) CFD computation (b) PIV measurement (c) CFD and PIV comparison inside propeller radius)

Generally, it is usually accepted that the prediction for the nominal velocity at the propeller plane for U-shaped hull like KVLCC2 tanker depends strongly on turbulence model as well as on a reasonably fine grid. The refinement block which has the axial grid spacing in the \( 10^{-4} \) order is believed to capture the flow field well. The station \( X = 0.98 \) is very close to the so-called \( X = 0.9825 \) station so that Fig. 5 is believed to have similar
behaviors with other computational results which are illustrated in Fig. 3.4 of Larsson et al.[12] In comparing with these results, the axial velocity contour of 0.4 also does not cross the vertical plane of symmetry and remains parallel to it. The main stern bilge vortex is also accurately captured and the hook-shape of the axial velocity contours is also well produced. KVLCC2 hull has more U-shaped stern frame-lines and the flow around it is characterized by the gradual development of an intense stern bilge vortex which creates a strong distortion of the axial velocity contours at the propeller plane[12]. This distortion is due to the transport of low momentum fluid from the vicinity of the hull to the center of the flow field under the action of an intense longitudinal vortex. Under the main vortex, there exists the secondary counter-rotating vortex close to the vertical plane of symmetry and this can be seen clearly just near below the hub shown in the vector field. The main difference between CFD and PIV is the area near the hub surface because the changes of the small values are visualized in a wide area on a 2-D flat diagram.

The volume average axial velocity is calculated by Eq. (5), i.e. nominal wake factor or mean value of u/U in Fig. 5 inside the propeller radius. \( u_k = 0.389 \) for CFD and 0.383 for EFD on propeller plane for without-rudder case. The nominal \( u_k \) values are lower than the effective \((1-\omega)\) of 0.453 in the previous section indicating the propeller-hull interaction (vorticity distribution around hull was changed by propeller induced velocity).

\[
 u_k = \frac{1}{m(r_p-r_0)} \int_{r_0}^{r_p} u dA
\]

3.3 Wake Field Analysis of Without-rudder Case

For the wake behind the rotating propeller, the rudder is not fitted at first and the computation is carried out with propeller only as the wake field is easier to be analyzed. Analyzing the downstream part of the propeller can give the information how accurate the propeller model can predict the wake field. The converged bare-hull solution is obtained first which has been discussed in section 3.2 and the propeller model is coupled with the RANS solver. The propeller revolution rate is set up at the self-propulsion point of the model.

The two stations are analyzed for this case at same locations with EFD; \( X=0.989 \) and \( X=1 \) (AP) and both locations are downstream of the propeller where the flow field can be affected mostly by the rotating propeller. The two figures represent the flow field solutions at those stations; Fig. 6 for \( X=0.989 \) and Fig. 7 for \( X=1 \) with the comparison of the corresponding EFD results. By comparing the two computational results, it is distinct that the flow accelerated towards downstream whereas the highest axial velocity contour level 1.5 at \( X=0.989 \) (Fig. 6 (a)) jumps up to 1.6 at \( X=1 \) (Fig. 7 (a)). The iso-contour shapes are like twisted to the starboard side because of the flow rotation driven by the right-handed propeller. The hub vortex can be seen near the shaft center line (\( Z=-0.04688 \)) but vortex center is shifted a little bit to the left of the center line (\( Y=0 \)) and that behavior can be clearly seen in the cross-flow vector pattern of Fig. 7(b).

In the comparison, CFD velocity is generally under-predicted than EFD one and that can be seen in both figures. For the station \( X=0.989 \), the highest axial velocity field shows 1.6 for EFD whereas only 1.5 is achieved for CFD even though the flow patterns of both are quite similar. Moreover, the lowest contour level 0 can be observed at the shaft center line (\( Z=0.04688 \)) for CFD in a small core region. It means there is no flow velocity or flow separation occurs at that region. When the station becomes more far downstream at \( X=1 \) (Fig. 7 (a)), the CFD lowest contour lever near the hub vortex recovers to 0.6 which is still much higher than EFD’s. If only the computation is taken account, the reason may be concluded as the boundary layer effect of the hub because this location is closer to the region of the tip of the hub and it may be influenced by the boundary layer. But, this kind of behavior cannot be observed in the EFD case at the same station where the lowest contour at the hub vortex is around 0.9 so that the reason is too vague to be judged.

At the aft perpendicular station, the highest contour level 1.6 of EFD is much wider than CFD one showing the under-prediction of the computation. However, the upper region which is out of the regime where the propeller affects strongly is quite similar for both solutions like the patterns of contour level 0.8, 0.7 and 0.6. The location of the hub vortex of EFD is also shifted a bit to the left similar to CFD one and the cross-flow patterns are also in good agreement. However, it can be seen that vector length of CFD is shorter than EFD one which shows that the computation is dissipated.

![Fig. 6 Velocity profiles at the section X=0.989 of (a) CFD (b) EFD for without-rudder case](image)

3.4 Wake Field Analysis of With-rudder Case

In this section, the computation is carried out with the rudder fitted at zero drift angle and rudder forces will not be accessed and discussed. The main purpose is to evaluate the wake patterns downstream of the propeller by using the simple propeller model. Totally two downstream sections are therefore selected to be
analyzed; at X=1 (AP) and far more downstream at X=1.025 and are illustrated in Fig. 8 and Fig. 9 respectively with comparison of the experiment.

The comparisons are mainly carried out for the downstream part of the propeller. The computational results for the upstream part are available but would not be presented in this paper.

Fig. 7 (a) Velocity profiles of CFD, (b) cross-flow vectors of CFD, (c) velocity profiles of EFD and (d) cross-flow vectors of EFD at X=1 for without-rudder case

Fig. 8 (a) Velocity profiles of CFD, (b) cross-flow vectors of CFD, (c) velocity profiles of EFD and (d) cross-flow vectors of EFD at X=1 for with-rudder case
At the aft perpendicular position (AP) which is in the vicinity of the rudder, CFD can capture more detailed flow field near the rudder surface (Fig. 8 (a)) whereas EFD cannot capture in that place leaving blank with no measurement (Fig. 8 (c)). It is therefore a bit hard to make a qualitative comparison between the two results. Based on the available EFD result, the flow patterns on both port and starboard sides of the rudder seem to be in good agreement with CFD ones. The highest contour level of both solutions give 1.6 in non-dimensional value but it seems the port part of EFD result shows wider region for the legend 1.6. That should be counted that CFD predicts the velocity lower than EFD measured similar to the previous discussion. For the cross-flow components, CFD can predict the vector patterns near the rudder surface but EFD cannot present the condition much. In Fig. 8 (b), the two vortices can be observed on both side of the rudder around Y=-0.005 and Y=0.01. The hub vortex generated by the propeller accelerated flow is shifted a little bit to the left of the center line (Y=0) and another vortex appears to the starboard side of the rudder. By comparing the two vortices, it can be seen the port side one is much stronger than the starboard one. The main part of the hub vortex is twisted and shedding into the portside. A right rotating vortex would tend to move to the left in an upward flow field (in ship stern). It causes the downward flow along the starboard side surface of the rudder which induces the starboard side vortex. EFD seems to show the hub vortex near Y=-0.005 but there is no tentative shape at Y=0.01. Generally, the cross-flow component structures of the two results are also close enough.

For the clear comparison, another far downstream section is selected at X=1.025 where EFD can give full flow field layout. This location is more far away from the propeller plane and the right-handed rotating behavior of the propeller gives the flow field shape twisted as shown in Fig. 9 (a) and (c) with port side higher and with starboard side lower. Both of the results give similar shapes with the same highest axial velocity level 1.6 but again, CFD gives under-predicted solution. A hub vortex can be observed in both results near the cross-point of Y=-0.005 and Z=-0.048. The starboard vortex cannot be seen in EFD again similar to the aft perpendicular position but it is distinctly observed in CFD. Another vortex appears near the cross-point of Y=-0.005 and Z=-0.062 in EFD but this pattern is lack or unclear in CFD. Moreover, the cross-flow vectors near the center line (Y=0) of EFD in Fig. 9 (d) gives strong jet-like flow at around Z=-0.03 to upwards and Z=-0.06 to downwards. The CFD counterpart in Fig. 9 (b) gives shorter and less dense vectors. Generally, it can be concluded that cross-flow components of CFD is lower than corresponding EFD values. In this location the effects of flow field from the hull, propeller and rudder is combined and interacting to each other. Thus, the detailed analysis is drawn at each vertical position (Z=-0.03, -0.035, -0.04, -0.045, -0.05, -0.055 and -0.06) with the comparison between CFD and EFD to understand how different the results are. In Fig. 10, the patterns are much closer between the two results with comparatively lower velocity of CFD. The differences are more obvious in the propeller plane region and better agreement is obtained outside that region.

3.5 Wake Field Analysis of With-rudder Case at Ship Point

In Section 3.2 and 3.3, the propeller revolution rate is set at 16.5 rps for model point. This section will mainly focus on the CFD and EFD with the propeller revolution rate 11.1 rps for ship point. The flow fields will be analyzed only at X=1.025 where the combined effect of the propeller-hull-rudder interaction on the wake field can be observed clearly and the comparison with the corresponding EFD result is illustrated in Fig. 11.
In both results (Fig. 11), the flow field patterns are quite similar to the previous solutions but lower axial velocity profile is observed because of the lower revolution rate. The highest axial velocity level in both solutions give 1.1 respectively, but a little bit lower estimation in the computation is observed and that behavior is consistent with that of the previous sections. The experiment here gives a little bit fluctuation which seems to have much noise. Unlike the model point case, EFD in this case shows the distinct vortex pattern in the starboard side of the center plane near the cross-point of Y=0.01 and Z=-0.05. It is induced by the downward jet flow along the starboard rudder surface. Moreover, the CFD result shows the third vortex near the cross-point of Y=0 and Z=-0.06 clearly that cannot be observed well in the model point. It is like a tip vortex across the rudder bottom edge from starboard to portside. For the upstream stations, CFD and EFD show better agreement. Due to the article length and main interest in rudder flow, the upstream comparison will not be showed here but the data is available.

3.6 Comparison with Real Geometry Propeller Computation and Other Body-Force Propeller Models

The propeller model has been validated with two sets of EFD results. To demonstrate the capability of the propeller model, the corresponding results for the real propeller geometry simulation, axisymmetric model and Yamazaki model using the same CFDSHIP-IOWA code are provided by Hosseini et al. In axisymmetric model, the body forces inside the actuator disk is prescribed by open water curve and analytical distribution of force. In the present simple body force propeller model, the drag coefficient (Cd) is set as 0.02 in blade element force calculation and this was determined by comparing EFD and CFD of open water characteristics. Note that the trip wires at 10% chord position of both sides were attached for small size propeller model. For Yamazaki model (lifting line), Cd=0.01. These three cases are carried out to predict the self-propulsion point by free running simulation with free to heave and pitch conditions.
computations are based on captive model with even-keel condition at different propeller revolution rates. The sinkage and trim is negligible since they are small values. The current study will assume that the ship wake solutions should give similar results at the same propeller revolution rate so that the wake patterns will be mainly compared and discussed in this section. The test conditions and particulars of the real propeller computation are shown in Table 1 in which the combined particulars of the INSEAN ship hull, speeds and NMRI propeller diameter are used. The ship length is 7 m with the ship speed 1.179 m/s so that Reynolds number is $8.237 \times 10^6$. The Froude number is 0.142 and the results for each case are listed in Table 5.

Our computations are carried out for three revolution rates corresponding to the cases in Hosseini et al. The comparisons are made based on these solutions which are illustrated in Fig. 12 to Fig. 15. The axial velocity contours at the aft perpendicular X=1 provided by Hosseini et al. for self-propulsion case in the free-running condition are shown in Fig. 12 (a), Fig. 13 (a) and Fig. 14 (a). According to the different thrust that is produced by each model, the different propeller revolution rate is required to achieve self-propelled. The axisymmetric model has the highest revolution rates of 12.275 rps. The real geometry computation has the lowest rps and highest thrust coefficient to produce the highest axial velocity fields than the other two models do. Here, it should be noted that the real propeller geometry result is not time-averaged and it is just the solution at one instant time while the other models are time-averaged solutions. The time-averaged solution for several time steps with corresponding blade positions are not available in this paper. Thus, the flow field patterns will be generally compared.

Fig. 12 shows the comparison of the wake field at X=1 between the simple body-force model and the real geometry results. The two solutions could provide the similar trends with the highest contour of 1.7 but the simple body-force model predicts much smaller area than the real geometry one. It is consistent with the lower thrust and torque coefficients shown in Table 5 for the simple model. Since the same RANS code is used the dissipation should be same and the reason can be concluded like the aforementioned fact that the real geometry solution is not time-averaged. It is also distinct by judging from how the actual propeller can give the highest flow velocity by the lowest propeller revolution rate among the models.

To understand the time-averaged effect, the current body-force model is compared with the other models: Yamazaki model in Fig. 13 (11.17 rps) and axisymmetric model in Fig. 14 (12.275 rps). In both figures, it can be seen that the simple body-force results show the higher wake velocity than the other two models. The contour shapes are quite similar among three models. However, while the current model gives the contour line of 1.7 in the wider range, the Yamazaki model can predict only up to the level of 1.6 and the axisymmetric model shows smaller area for the contour line of 1.7. Correspondingly, in the comparison of the thrust and torque coefficients for each case in Table 6, the current model gives higher quantities. It can be concluded that by the same time-averaged modelling, the simple body-force model can predict higher wake velocity than the other two models even though it was proved as under-prediction in comparing with the experiments.

In order to understand the vortex structure, the Q criterion iso-surface with $Q=10000$ colored with the axial vorticity contours viewing from the stern is illustrated in Fig. 15 for the real geometry propeller and the current model. For the real geometry result, the propeller tip-vortex and hub vortex is shedding with an effective helical pitch. Due to the time-averaged effect, in the current body-force model the propeller vortex is formed as a ring shape and the hub vortex is in a long finger shape pointing downstream. As the rotating hub effect is included in both computations, the strong hub vortex can be seen shift towards the portside of the rudder. The application of the propeller model to the energy saving device installed on the rudder is expected because the vortex effect around the rudder could be represented.

| Propeller Model | Method       | $K_T$   | $K_Q$   | RPS    |
|-----------------|--------------|---------|---------|--------|
| Real Geometry   | Free Running | 0.192   | 0.0237  | 10.52D |
| Axisymmetric    | Captive      | 0.147   | 0.0188  | 12.275 |
| Yamazaki        | Captive      | 0.174   | 0.0193  | 11.170 |
| Simple model    | Captive      | 0.187   | 0.0214  | 10.520 |
| Simple model    | Captive      | 0.199   | 0.0220  | 12.275 |
| Simple model    | Captive      | 0.192   | 0.0217  | 11.170 |

4. Conclusions

The RANS computation of the tanker hull form KVLCC2 using simple body-force propeller model has been carried out with the corresponding wake analysis behind the ship where the propeller-hull and propeller-hull-rudder interaction affects most. The computational results for the upstream part of the propeller as well as at the other stations that are not discussed in this paper can be available via the authors.

For the main purpose, it has been proved that a very simple propeller model can predict the complicated flow field of the tanker. The ship wake field patterns resulted from the current computations have been discussed. The pros and cons are pointed out in the comparison with its counterpart results. Several test cases like without-rudder, with-rudder at the self-propulsion point of the model as well as at the ship have been computed and the wake fields are analyzed in details with the comparison to the experimental data. Moreover, the results are also compared with the real geometry propeller computation as well as with other body-force models to find out the advantage of the current propeller model. The simple body-force model gives under-predicted solutions compared to the EFD data but it gives higher solutions than the other body-force models. As the wake field is complicated phenomenon, a very fine grid type is needed to capture the flow as well as the proper turbulence modelling is required to predict the turbulent behavior. Based on either of these facts or both a further study is required to recover the dissipation problem that is found out in this research.

The body-force model is proved useful for the application of computational propulsion or seakeeping or maneuvering field whereas the high quality of the detail flow field around the propeller blade is unnecessary. And it is still too expensive to use the real geometry propeller computation for these kinds of researches. Also, in order to cope the complicated mesh around each blade, the high quality skill of the researchers is needed.
Fig. 12 Comparison of velocity profiles between (a) the actual propeller model and (b) the simple body-force model at X=1.0

Fig. 13 Comparison of velocity profiles between (a) the Yamazaki model and (b) the simple body-force model at X=1.0

Fig. 14 Comparison of velocity profiles between (a) the axisymmetric propeller model and (b) the simple body-force model at X=1.0

Fig. 15 Comparison of iso-surface Q=10000 colored by x-vorticity between (a) the actual propeller model and (b) the simple body-force model
Moreover, some concepts of using body-force effects are still complicated as there are several steps to adjust between the RANS and propeller performance programs based on potential flow theory because the effective wake is required to be extracted from the total velocity fields. The current proposed propeller model is so simple and just a very short subroutine of the program is enough to compute the body-force terms and its competency over the other models has also been proved in this paper. As a final conclusion, the RANS simulation around the tanker KVLCC2 using the very simple body-force distribution model for both with-rudder and without-rudder cases has been successfully computed, analyzed, discussed with the qualitative and quantitative comparisons with experimental works.

Acknowledgment(s)

This work was partially supported by JSPS KAKENHI Grant Number 24246142 and REDAS (15-12(3)). The authors of this research give their heartily thanks to Dr. Hamid Sadat-Hosseini, an assistant research scientist at IIHR, University of Iowa, USA.

References

1) Win, Y.N., Tokgoz, E., Wu, P.C., Stern, F. and Toda, Y.: Computation of Propeller-Hull Interaction using Simple Body-Force Distribution Model around Series 60 C_b = 0.6, Journal of Japan Association of Naval Architects and Ocean Engineers (JASNAOE), Vol. 18, pp. 17-27, 2013.
2) Win, Y.N., Tokgoz, E., Wu, P.C., Stern, F. and Toda, Y.: Computation of Propeller-Hull interaction using Simple Body-Force Distribution Model around modified Series 60 C_b=0.6 with hub, Proceedings of 24th International Ocean and Polar Engineering Conference, Vol. 4, pp. 759-765, 2014.
3) Paterson, E. G., Wilson, R. V., and Stern, F.: General-purpose parallel unsteady RANS ship hydrodynamic code: CFDSHIP-IOWA, IIHR report No. 432, 2003.
4) Stern, F., Kim, H.T., Zhang, D.H., Toda, Y., Kerwin, J., and Jessup, S.: Computation of Viscous Flow Around Propeller-Body Configurations: Series 60 CB=0.6 Ship Model, Journal of Ship Research, Vol. 38, No. 2, pp. 137-157, 1994.
5) Kerwin, J.E. and Lee, C.S.: Prediction of Steady and Unsteady Marine Propeller Performance by Numerical Lifting-Surface Theory, SNAME Transactions, Vol. 86, pp.218-253, 1978.
6) Simonsen, C., and Stern, F.: “RANS Maneuvering Simulation of Esso Osaka With Rudder and a Body-Force Propeller, Journal of Ship Research, Vol. 49, No. 2, pp. 98-120, 2005.
7) Yamazaki, R.: On the propulsion theory of ships on still water (improved theoretical method), Memoirs of the Faculty of Engineering, Kyushu University, Vol. 34, No. 1, pp. 65-88, 1977.
8) Winden, B., Huang, Z., and Kawamura, T.: Comparative self-propulsion simulations of the IBC bulk carrier, Journal of Japan Association of Naval Architects and Ocean Engineers, Vol. 20, pp 247-250, 2015.
9) Tokgoz, E., Win, YN., Kuroda, K., and Toda, Y.: A New Method to Predict the Propeller Body Force Distribution for Modeling the Propeller in Viscous CFD Code without Potential Flow Code, Journal of Japan Association of Naval Architects and Ocean Engineers (JASNAOE), Vol. 19, pp 1-7, 2014.
10) Noack R.: Suggar: a general capability for moving body overset grid assembly, 17th AIAA computational fluid dynamics conference, Toronto, Ontario, Canada, AIAA paper 2005-5117, 2005.
11) Boger D.A. and Dreyer J.J.: Prediction of Hydrodynamic Forces and Moments for Underwater Vehicles Using Overset Grids, AIAA paper 2006-1148, 44th AIAA Aerospace Sciences Meeting, Reno, Nevada, 2006.
12) Larsson, L., Stern, F., and Visonneau, M.: Numerical Ship Hydrodynamics, An assessment of the Gothenburg 2010 Workshop, March, 2013
13) Hosseini, H., Wu, P., and Stern, F.: CFD Simulation of KVLCC2 Maneuvering with Different Propeller Modeling, Workshop on Verification and Validation of Ship Maneouvrving Simulation Methods (SIMMAN 2014), Lyngby, Denmark, 2014.