Numerical Study on Characteristics of Separated Flows around an Extremely Blunt Ship with and without a Stern Tunnel

by Nang Tin Tin Htwe*, Takanori Hino*, Member Kazuo Suzuki*, Member

Summary

A novel ship concept which is called ULBS (Ultra Large Block coefficient ship) to reduce CO2 emission form sea transportation is under investigation at Yokohama National University. Since ULBS is supposed to have a very blunt hull and use of various flow control devices are essential for better hydrodynamic performance, flow field analysis around a ship is crucial for a design of ULBS. Computational Fluid Dynamics (CFD) is expected to be an efficient design tool for unconventional hull forms such as ULBS. To investigate the flow characteristics of ULBS, free surface viscous flows around the ship with and without a stern tunnel are computed. Total resistance coefficients are compared with each other and with experimental data. Grid convergence study is performed with respect to resistance for the verification of the results. The three dimensional vortical structures in the stern of the two hulls are analyzed and compared with each other. Large scale flow separations behind the stern due to the bluntness of a hull and effects of a stern tunnel to flow fields are discussed.

1. Introduction

Demands for efficient sea transportation have been increasing due to globalization of world economy. At the same time, global environment protection becomes more important than ever. In view of these circumstances, the applied ship hydrodynamics group of Yokohama National University proposes a new concept of a cargo ship which is called Ultra Large Block coefficient Ship, ULBS. ULBS is a very blunt ship (block coefficient $C_b > 0.95$) with a large breadth-to-length ratio ($B/L > 0.2$), which enables larger cargo capacity. The extreme bluntness, however, yields higher resistance and poor performance. In order to minimize this deficit, ULBS is designed to sail at low speed. Furthermore, various flow control devices are attached to improve hydrodynamic performances.

Since ULBS has an extremely blunt hull form, it is expected that large scale separations are dominant in a flow field around a hull. Therefore, understanding of flow properties is crucial for the better design of a hull form and flow control devices. Conventionally, ship hydrodynamics analysis has been carried out for a streamlined body. Even with a blunt ship such as VLCC, a stern part is streamlined for better propulsive performance. CFD applications for such cases are in $[1,2]$. However, due to the large $C_b$, the stern of ULBS is not streamlined and its shape is similar to a box. In the present study, a CFD (Computational Fluid Dynamics) method currently used for practical ship flow computations is applied to the flow simulations around ULBS. Based on the experiences of the preliminary study $[3]$ in which CFD analysis is applied to box-shaped ships, higher-order turbulence model is adopted in anticipation of more accurately simulating flow separations.

Thus, the objective of the present study is to assess a currently used CFD method with an advanced turbulence model for simulating strongly separated flows around an extremely blunt ship. In addition, differences of flow structures between a bare hull and a hull with a stern tunnel is examined using numerical results together with available experimental data.

2. Numerical Procedure

2.1 Flow Solver

The flow solver used in this study is SURF (Solution algorithm for Unstructured RaNS with FVM) which is under development at National Maritime Research Institute $[4,5,6]$. The governing equations used are the three-dimensional Reynolds averaged Navier-Stokes equations for incompressible flows. In order to couple pressure with a velocity field, artificial compressibility is introduced into the continuity equation. In order to account for the unsteadiness, the dual time stepping is employed in which the physical time and $r^*$ is the pseudo time for artificial compressibility and the sub-iteration with $r^*$ is performed at each time step in $t$.

The final form is written as follows:

$$\frac{\partial q^*}{\partial t} + \frac{\partial q}{\partial x} + \frac{\partial \left( e - e^* \right)}{\partial y} + \frac{\partial \left( f - f^* \right)}{\partial y} + \frac{\partial \left( g - g^* \right)}{\partial z} = 0$$

(1)

where

$$q = [ p \ u \ v \ w ]^T$$

$$q^* = [ 0 \ u \ v \ w ]^T$$

In the above expressions all the variables are made dimensionless using the reference density $\rho$, velocity $U$ and length $L_{pp}$. Pressure $p$ is modified as

$$p = p^* + \frac{z}{Fr^2}$$

where $p^*$ is the original pressure and $Fr$ is the Froude number, with $z$ being the vertical coordinate. The velocity components in the $(x, y, z)$ direction is expressed as $(u, v, w)$. The inviscid fluxes

---

* Yokohama National University
Received 7th June 2013
e, f and g are defined as

\[
e = \begin{bmatrix}
\beta u \\
\beta v \\
wv
\end{bmatrix}, \quad f = \begin{bmatrix}
vu \\
v^2 + p \\
wv
\end{bmatrix}, \quad g = \begin{bmatrix}
wv \\
wv \\
v^2 + p
\end{bmatrix}
\]

where \(\beta\) is a parameter for artificial compressibility. The viscous fluxes \(e^v, f^v\) and \(g^v\) are written as:

\[
e^v = \begin{bmatrix}
\tau_{xx} \\
\tau_{xy} \\
\tau_{xz}
\end{bmatrix}, \quad f^v = \begin{bmatrix}
0 \\
\tau_{xy} \\
\tau_{xz}
\end{bmatrix}, \quad g^v = \begin{bmatrix}
0 \\
\tau_{xx} \\
\tau_{xz}
\end{bmatrix}
\]

where

\[
\tau_{ij} = \frac{1}{Re} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial t} \right) + \tilde{\tau}_{ij}
\]

and \(Re\) is the Reynolds number defined as \(UL_{pp}/v\) with \(v\) being the kinematic viscosity. \(\tilde{\tau}_{ij}\) is the Reynolds stress terms determined by an appropriate turbulence model.

For the inviscid fluxes (convection terms and pressure gradient terms), the second order upwind scheme based on the flux-differencing splitting of Roe[7] with the MUSCL approach is employed. The viscous fluxes are evaluated by the second order central scheme. Thus, the overall accuracy in space is the second order. The backward Euler scheme is used for the time and the pseudo time integrations. Local time stepping method is used for the pseudo time \(t^*\), in which time increment is determined for each cell in such a way that the CFL number is globally constant. On the other hand the constant time interval is used for the physical time \(t\) in order to maintain time accuracy. The linear equations derived from the time linearization of the fluxes are solved by the Gauss-Seidel iteration.

Free surface is an interface between air and water in the present applications. Free surface conditions consist of kinematic and dynamic conditions and they are implemented in the interface capturing framework. The kinematic condition is the condition that fluid particles on a free surface remain on an interface. This condition is implemented based on the localized level set method[9] which improves the efficiency of the original level set approach[9]. Since most of ship hydrodynamics applications require a flow field of water region only, single-phase flow approach is used, i.e., the flow equations are solved only in a water region. Flow variables in an air region are, therefore, extrapolated from a water region in such a way that the dynamic condition on free surface boundary is satisfied.

2. 2 Turbulence Model

A turbulence model is essential for simulating high Reynolds number flows of practical interests. In this study, the Explicit Algebraic Reynolds Stress Model (EARSMS) model[10] which is based on the \(K - \omega\) model[11] is used. Since separations are expected to be dominant in the flow fields around an extremely blunt ship, it is considered that the higher-order models are better suited than the standard eddy viscosity models.

The Reynolds stress tensor \(\bar{\tau}_{ij}\) is given by the explicit algebraic relation of the average velocity fields as

\[
\bar{\tau}_{ij} = \frac{2}{3}K\delta_{ij} - 2\nu^* \left( S_{ij} + \frac{2}{3} \alpha_4 \left( S_{ik} W_{kj} - W_{ik} S_{kj} \right) \right)
\]

where the non-dimensional strain-rate \(S_{ij}\) and vorticity tensors \(W_{ij}\) are defined as

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

and

\[
W_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)
\]

\(v^*\) is the kinematic eddy viscosity determined from the turbulent energy \(K\) and the turbulence frequency \(\omega\) of the \(K - \omega\) model[11]. The other parameters are given using the average velocity components.

3. Ship Models and Flow Conditions

3. 1 Ship Models

Ship hull forms used in this study are conceived as a typical ULBS (Ultra Large Block coefficient Ship) hull using mathematical formulations. Two ship models are considered. One is called “Bare Hull” hereafter and its waterline at bow is expressed as a semi-ellipse followed by the parallel part which continues to the aft-end. Frame lines are wall-sided with a bilge circle at the bottom. The other model is called “Stern Tunnel” which is based on “Bare Hull” and a stern tunnel is attached in the aft part. The width and the depth of a tunnel is 1/3 of breadth \(B\) and 1/2 of the draft \(d\). Tunnel has a wedge-shape with its length of 1/5 of \(L_{pp}\). Two hull shapes are shown in Fig. 1 and the principal particulars are tabulated in Table 1.

![Fig. 1 Hull forms of “Bare Hull”(top) and “Stern Tunnel” (bottom).](image)

<table>
<thead>
<tr>
<th>Table 1 Principal particulars of ship models.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Breadth ((B/L_{pp}))</td>
</tr>
<tr>
<td>Draft ((d/L_{pp}))</td>
</tr>
<tr>
<td>Wetted Surface Area (S/L_{pp}^2)</td>
</tr>
<tr>
<td>Block Coefficient (C_b)</td>
</tr>
</tbody>
</table>

3. 2 Computational Conditions

Reynolds number \(Re\) and Froude number \(Fr\) of the computations are set to be the same as in the experimental condition, which are \(Re = 8.4 \times 10^5\) and \(Fr = 0.15\). The experiments were carried out in the circulating water channel (its measuring section is 2.5m
Numerical Study on Characteristics of Separated Flows around an Extremely Blunt Ship with and without a Stern Tunnel

(1.2m (length) × 0.6m (width) × 0.6m (water depth)) of Yokohama National University with ship models of \( L_p = 1.5m \).

Computational grids are generated using the commercial code Pointwise. Grids for “Bare Hull” and “Stern Tunnel” are designed to have the same properties as much as possible in order to minimize the effects of grid differences. Computational domain is similar to a quarter of an ellipsoid with \(-2 \leq x \leq 2.6, -2 \leq y \leq 0\) and \(-2 \leq z \leq 0.05\), while a ship is located \(-0.5 \leq x \leq 0.5\). Only the port side is discretized due to symmetry. Each grid consists of 8 structured grid blocks and a total number of cells are 4,579,328.

Fig. 2 shows the partial views of the computational grids. Note that the bow grids are identical for both models.

Computations are performed using the unsteady flow mode of the solver with the dual time stepping and non-dimensional time interval of \( \Delta t = 0.002 \) is used for the physical time marching.

4. Verification of Computed Resistance Coefficients

In order to verify the computational solutions obtained, the simulation numerical uncertainty is estimated. In particular, grid uncertainty \( U_g \) for the total resistance coefficient \( C_t = R_t/0.5 \rho U^2 L_p^2 \) where \( R_t \) is the total resistance is estimated using three systematically refined grids. The procedure is in accordance with the Recommendations of ITTC (12, 13).

The computed results of time-averaged resistance values of \( C_t \) using three grids are shown in Table 2 and results of grid uncertainty analysis are shown in Table 3. The grid refinement ratio \( r \) for three grids is \( \sqrt[3]{2} \), thus, the cell numbers are 4,579,328 for the fine grids shown in Fig. 2 and 1,642,004 for the medium grids and 572,416 for the coarse grids. In the “Bare Hull” case, \( C_t \) monotonically decease with grid refinement and the grid convergence ratio of \( R_G = (S_2 - S_1)/(S_3 - S_2) = 0.158 \) is obtained, where \( S_1, S_2 \) and \( S_3 \) are solutions of fine, medium and coarse grid, respectively. The grid uncertainty \( U_g \) is estimated as 0.22% of \( S_1 \) (the fine grid solution), based on the observed order of accuracy, \( P_G = 5.33 \) and the correction factor \( C_G = 5.33 \).

On the other hand, \( C_t \) shows oscillatory convergence with \( R_G = -0.125 \) in the “Stern Tunnel” case and the grid uncertainty is estimated as 2.18% of \( S_1 \) from the maximum and the minimum of \( C_t \). Although the uncertainty levels are reasonably low for both cases, \( C_t \) of “Stern Tunnel” is larger than “Bare Hull” and this trend is opposite to that of the measurement shown in Table 2. In the experiment, the models with 1.5m \( L_p \) are used in the circulating water channel at Yokohama National University. The resistance is measured with the load cell with the capacity of 20 N and its accuracy is given as 0.2% of full scale. Since the uncertainty of the measurement is not available, the validation of the results is not performed.

The computed resistance components are shown in Table 4. The main difference comes from pressure resistance. Increase of pressure resistance in “Stern Tunnel” can be confirmed by viewing the hull surface pressure distributions in the stern shown in Fig. 3. Note that the fore part of the hulls are identical for both hulls. With the presence of a stern tunnel, flow inside the tunnel is accelerated as discussed in the following section and pressure decreases. Thus, the relatively higher pressure zone near the center on “Bare Hull” disappears in “Stern Tunnel”. Therefore, larger resistance in “Stern Tunnel” seems to be consistent with flow fields. Table 4 also includes the computed resistance coefficients with the medium grids using the conventional eddy viscosity model \( (K - \omega \text{SST model}^{11}) \). The SST results show that \( C_t \) of “Bare Hull” is slightly larger than “Stern Tunnel” and the trend is similar to the experiment. The time histories of computed resistance coefficients of these cases are shown in Fig. 4. The unsteadiness of the resistance histories is more pronounced in the EARS model than in the SST cases, which indicates the vortex shedding associated with the flow separation described below seems to be better captured by the EARS model.

A reason for the opposite prediction of resistance trend by the EARS model is unknown. It may be due to the numerical modeling errors of the current turbulence models for strongly separated flows or due to uncertainty of measurement. Further investigations are required both in simulations and measurements to clarify the reasons of this discrepancy.

Table 2 Grid convergence study of total resistance \( C_t (\times 10^3) \).

<table>
<thead>
<tr>
<th>Cell no.</th>
<th>Coarse</th>
<th>Medium</th>
<th>Fine</th>
<th>Exp.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bare Hull</td>
<td>572,416</td>
<td>1,642,004</td>
<td>4,706,178</td>
<td>—</td>
</tr>
<tr>
<td>Stern Tunnel</td>
<td>5.59</td>
<td>5.21</td>
<td>5.15</td>
<td>5.93</td>
</tr>
</tbody>
</table>

Table 3 Verification of total resistance \( C_t \).

<table>
<thead>
<tr>
<th>Ship</th>
<th>( R_G )</th>
<th>( P_G )</th>
<th>( C_G )</th>
<th>( U_G/%S_1 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bare Hull</td>
<td>0.158</td>
<td>5.33</td>
<td>5.33</td>
<td>0.22</td>
</tr>
<tr>
<td>Stern Tunnel</td>
<td>-0.125</td>
<td>—</td>
<td>—</td>
<td>2.18</td>
</tr>
</tbody>
</table>

\( S_1 \): fine grid solution

Table 4 Computed resistance components \( (\times 10^3) \).

<table>
<thead>
<tr>
<th>Ship</th>
<th>Total ( C_t )</th>
<th>Pressure ( C_p )</th>
<th>Friction ( C_f )</th>
<th>Total(SST) ( C_t )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bare Hull</td>
<td>5.15</td>
<td>4.22</td>
<td>0.93</td>
<td>5.43</td>
</tr>
<tr>
<td>Stern Tunnel</td>
<td>5.51</td>
<td>4.56</td>
<td>0.95</td>
<td>5.39</td>
</tr>
</tbody>
</table>

Fig. 2 Partial views of computational grids. Bow grid (top), Stern grid (“Bare Hull”) (bottom left) and Stern grid (“Stern Tunnel”) (bottom right).
5. Results and Discussions

5.1 Effect of Turbulence Models

In order to examine the effect of turbulence models on flow fields, comparison are made between the results with EARSM and the $K - \omega$ SST model.

Fig. 5 shows the instantaneous streamlines with contours of $u$ (velocity in $x$-direction) on the center plane around the bow bottom together with the visualization of the flow field in the experiment. These plots together with the following plots of flow fields are produced using the snapshot data of unsteady flow fields, since the snapshots can display the detailed flow structure such as vortex shedding.

In the $K - \omega$ SST case, only one vortex is observed under the bow bottom, while in the EARSM case multiple vortexes with a small scale are captured, which is closer to the visualization result in the experiment. This is also consistent with the fact that the resistance histories of the EARSM results show the larger fluctuations than the SST results in Fig. 3. In addition, the vortical flow structures in the stern region of “Stern Tunnel” shown in Fig. 6 indicate the better resolution of the EARSM result as discussed below. It is thought that EARSM turbulence model can capture the more detailed flow features compared with the $K - \omega$ SST model. Therefore the EARSM model will be used in this study with the anticipation that separated flows in stern regions can be predicted better than the eddy viscosity models.

5.2 Vortical Flow Fields Near the Bottom

Fig. 6 shows the bottom views of the vortical flow fields for “Bare Hull” and “Stern Tunnel”. In the plot, iso-surfaces of the second invariant of the velocity gradient $Q = 5$ are shown. $Q$ is defined as

$$Q = \frac{1}{2} (W_{ij}^2 - S_{ij}^2)$$

where the rate of strain tensor $S_{ij}$ and the vorticity tensor $W_{ij}$ are defined in Eqs. (3) and (4), respectively. The $Q$-criterion14) is used here to identify the coherent vortical structures. If $Q$ is positive, the Euclidean norm of vorticity tensor dominates the rate of strain which evidences the rotation of the flow. The flow fields around the bow of “Bare Hull” and “Stern Tunnel” are essentially identical because of the same geometry of the fore part. Small discrepancies come from the fact that flows are unsteady and fluctuating slightly. Also shown in Fig. 6 is the magnified view of the bow part of “Stern Tunnel”, where a series of vortex structures shedding from the bow are observed and this is consistent with streamlines shown in Fig. 5. This feature in the bow is common between two hulls. On the other hand, the flow fields in the stern show significant differences between two hulls and “Stern Tunnel” hull generates more complicated flows than “Bare Hull” as shown in the bottom of Fig. 6. Stern of “Bare Hull” has a simple shape with continuing parallel part and the vortices are generated at the edges of the stern. On the other hand, in case of “Stern Tunnel”, vortices are generated on the walls of the tunnel in addition to the stern edges of a hull and the vortices are shed to downstream. The SST result for “Stern Tunnel” does not indicate the vortex shedding characteristics behind the hull compared with the EARSM result.
Fig. 6 Iso-surfaces of the second invariant of velocity gradient $Q = 5$ colored by $u$ velocity around the hull (bottom view). “Bare Hull” (top left), “Stern Tunnel” (top right), magnified view at the bow of “Stern Tunnel” (middle left) and magnified views at the stern “Stern Tunnel” (SST) (middle right), “Bare Hull” (bottom left) and “Stern Tunnel” (bottom right).

5.3 Flow Structures behind the Stern

Sketches of the main vortex structures behind the stern are shown in Fig. 7. The following terms are defined to discuss the flow structures behind the stern:

<table>
<thead>
<tr>
<th>Term</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>HSV</td>
<td>Vortex generated at the side edge of a main hull</td>
</tr>
<tr>
<td>HBV</td>
<td>Vortex generated at the bottom edge of a main hull</td>
</tr>
<tr>
<td>TSV</td>
<td>Vortex generated at the side edge of a tunnel</td>
</tr>
<tr>
<td>TBV</td>
<td>Vortex generated at the bottom edge of a tunnel</td>
</tr>
</tbody>
</table>

In case of “Bare Hull”, two vortex structures, HSV and HBV, are dominant, while four structures, HSV, HBV, TSV and TBV, are present in “Stern Tunnel” and flow field becomes more complicated.

In Fig. 9, vorticity magnitude distributions on a center plane $y = 0$ and on the section of $y = -0.067$ which corresponds to the middle of a half-span of the main hull beside the stern tunnel are shown. On the center plane, vorticity develops horizontally from the bottom edge of the stern in case of “Bare Hull” (HBV), while vorticity goes up with the same angle of the tunnel bottom in “Stern Tunnel” case (TBV). Also, the high vorticity region of HBV in “Bare Hull” is larger than TBV in “Stern Tunnel”, which shows separation behind the stern of “Bare Hull” is stronger than that of “Stern Tunnel”. On the plane of $y = -0.067$, vorticity develops horizontally in both cases, since these vortices can be considered to be HBV. Again, the intensity of the vorticity is larger in “Bare Hull” than in “Stern Tunnel”.

The same flow structures can be captured in the iso-surfaces of $Q = 5$ in Fig. 10. It is observed that a series of vortices comes from the bottom edge of the stern in “Bare Hull” case (HBV), and vortices with weaker intensity come from the bottom of stern tunnel in “Stern Tunnel” case (TBV). Also the vortices from the side of the hulls (HSV) is larger in “Bare Hull” than in “Stern Tunnel”.

In Fig. 11, velocity distributions and streamline behind the stern on the center plane. The re-circulation region in “Bare Hull” is much larger than in “Stern Tunnel”. The multiple swirling can be observed in “Bare Hull” and this is related to the series of iso-surfaces of $Q$ in HBV region in Fig. 10. Flow directions behind the stern are close to horizontal in “Bare Hull” and upward in “Stern Tunnel” as expected.

Fig. 12 shows the similar plots on the free surface. Again the re-circulation region of “Bare Hull” is larger than “Stern Tunnel”.

Fig. 7 Sketches of the flow structures behind the stern “Bare Hull” (left) and “Stern Tunnel” (right).

Fig. 8 Iso-surfaces of the vorticity magnitude $= 20$ colored by $u$ velocity near the stern (top view). “Bare Hull” (left) and “Stern Tunnel” (right).
One big swirling flow with counter-clockwise rotation is observed in “Bare Hull”, while two main vortices are present in “Stern Tunnel”. One in the outside is in the same direction with “Bare Hull” and these are related to HSV. The other vortex inside is in the opposite rotation and related to TSV.

5.4 Velocity Fields behind the Stern

Figs. 13 and 14 show the velocity distributions at the section of $x = 0.5533$, 5.33% $L_{pp}$ behind the stern end. “Bare Hull” has a simple wake with the minimum velocity in the center line and near the free surface. On the other hand, the dominant wake of “Stern Tunnel” is behind the main hull part and the wake of a tunnel shows higher velocity due to acceleration in a tunnel. In the cross-flow vector plots in Fig. 14, the velocity directions are completely different between two hulls, which reflects the variation of re-circulation structures due to the stern tunnel. In “Stern Tunnel”, upward velocity are observed near the center plane, which also shows the acceleration due to the stern tunnel.

Fig. 15 shows the vorticity distributions at the section of $x = 0.5533$. These distributions clearly show HSV and HBV in “Bare Hull” and HSV, HBV, TSV and TBV in “Stern Tunnel”.

Fig. 16 is the comparison of computed and measured $u$ profiles at $x = 0.5533$ and $z = -0.033$. The measurement is carried out for the models of 1.5m $L_{pp}$ at the circulating water channel using the propeller-type velocimeter the diameter of which is 3mm. The accuracy of velocimeter is given as 0.01 m/s, while the actual uniform flow velocity in the experiment is 0.4735 m/s. Although the uncertainty of the velocity measurement is not available as well as the resistance test, the accuracy of the measurement is considered to be not high.

Computed results with the $K - \omega$ SST model are also shown in the plots. General trends are well reproduced by the computations for two turbulence models. The EARSM results show the wider wake than the SST results and the measurement. Although it appears that the SST result is better in “Bare Hull” case, the prediction of the reverse flow region in “Stern Tunnel” shows the underestimation by the SST model. Considering the better production of the vortex structure under the bow bottom by the EARSM model shown in Fig. 5, the EARSM model seems better suited for complex flows. Nevertheless, performance of both turbulence models is not sufficient for accurate predictions of the present flow fields.

5.5 Wave Fields

Although the detail of wave fields around a hull is out of scope of the present study, Fig. 17 displays comparison of the free-surface elevation contours around “Bare Hull” and “Stern Tunnel”. The large bow waves in front of a bow are dominant and
they are followed by the diverging wave system with Kelvin pattern. These features are common between two hulls. On the other hand, stern waves show different patterns. “Stern Tunnel” generates larger stern waves than “Bare Hull”, since the stern tunnel accelerates the flow significantly as seen in the previous sections.

![Fig. 17 Comparison of free-surface elevation contours around the hull: “Bare Hull” (top) and “Stern Tunnel” (bottom).](image)

6. Conclusions

In order to assess the capability of a CFD method currently used in practical ship flow computations with an advanced turbulence model for simulating strongly separated flows and to evaluate the effects of a stern tunnel, flow simulations are carried out for an extremely blunt ship with and without a stern tunnel.

The verification of the computed total resistance based on three grid sequence shows reasonable uncertainty levels of the present solutions. However, the resistance decrease with a stern tunnel in the measurement is not reproduced by the simulations. Although the reason for this discrepancy is not clear, it may be due to the numerical modeling error associated with a turbulence model or due to the uncertainty of measurement.

By using the iso-surface of the second invariant of the velocity gradient tensor and the vorticity magnitude, flow structures behind the stern are analyzed. It is found that the two main vortex sheets, one from the side edge of a hull and the other from the bottom edge of a hull, are formulated in case of “Bare Hull”. In “Stern Tunnel”, on the other hand, two more vortex sheets are generated at the side wall and the bottom of a tunnel in addition to hull side and hull bottom vortices.

Also it is found that the separation zone behind the stern is larger in “Bare Hull” than in “Stern Tunnel” by the examination of velocity fields and streamlines. Computed velocity profiles behind the hull reproduce the general trend of the measured data both in “Bare Hull” and “Stern Tunnel” cases. It appears that the performance of the turbulence models is not sufficient for quantitative comparisons with the measurement, although the uncertainty of the velocity measurement must be examined further.

Though further study of turbulence modeling effects is desirable for the improvement of numerical accuracy, the overall results in general show the difference of flow structures with and without a stern tunnel well.

Another future scope is a hull form optimization of ULBS. The large areas of flow separation are found in the bow bottom and the stern tunnel shape seems not optimal. The optimization of these parts as well as the use of other flow control devices are needed for better design of ULBS.

Acknowledgment

The first author acknowledges the Ministry of Education, Culture and Science of Japan (Monobukagakusho) whose doctoral course scholarship made the present research possible.

References

12) ITTC Recommended Procedures and Guidelines, “Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures”, 7.5-03-01-01.
13) ITTC Recommended Procedures and Guidelines, “CFD, Resistance and Flow Uncertainty Analysis in CFD Examples for Resistance and Flow,” 7.5-0.3-02-01.