On Viscous Flow around Marine Propellers
- Hub Vortex and Scale Effect -

By Isao FUNENO (Member)

Recently, the simulation technology by CFD (Computational Fluid Dynamics) has made remarkable progress. The method has become to be applied to viscous flow problems around not only various ship hulls but also marine propellers. Consequently many difficult problems on flow analysis on account of remarkable viscous effect are being resolved. There are also many subjects related to viscous effect of the flow around marine propellers. In this paper the author presents the topics about hub vortex and scale effect of propeller performance characteristics. The used analytical method consists of commercial CFD software and multifunctional grid generation software. As a result, the structure of flow around hub vortex and the difference of scale effect due to propeller shapes were revealed. These results show that the method is useful for design of such energy-saving devices as PBCF (Propeller Boss Cap Fins) or Kawasaki RBS-F (Rudder Bulb System with Fins), for ship power estimation with higher accuracy and for improvement of propellers performance.

Keywords: CFD, Propeller, Hub Vortex, Scale Effect, Cavitation

1. Introduction

With recent improvement of simulation technology by CFD (Computational Fluid Dynamics), it has become possible to apply the method to design of hydraulic machinery much more extensively. The improvement of simulation technology is also supported by remarkable progress of computer technology. For example, with advanced parallel computing technology by PC (Personal Computer)-cluster, it has become possible to conduct supercomputer-level high-performance computation at a low cost. It is a merit of the analytical method by CFD simulation able to obtain much more extensive flow data for three-dimensional complex geometries compared with those by experiments. Using these flow data, it is possible to support advanced design of hydraulic machinery. In ship hydrodynamic investigation at the preliminary design stage also, CFD simulation technique has become to be applied to viscous flow problems around not only various ship hulls but also marine propellers.

Conventional design methods of propellers consisted of the theoretical analysis based on inviscid flow and the experimental data by tank tests. To improve the propeller design method, it is necessary to take viscous effect of fluid into consideration. The author has presented the applicability studies of steady and unsteady viscous flow analysis around propellers. The author also has presented the study of tip vortex, which is related to underwater noise of low-level noise propellers equipped to oceanographic research ships. Since this tip vortex is strongly influenced by viscous effect of fluid, we have to take this effect into consideration in noise estimation.

In addition, as subjects we have to take viscous effect into consideration, there are hub vortex and scale effect on propulsive performance. Hub vortex strongly influences propulsive performances as effectiveness of such energy-saving devices as PBCF or Kawasaki RBS-F shows. Ouchi et al., Yang et al. and Atlar et al. investigated hub vortex. However, the author can not find out the detailed investigation about hub vortex by applying thoroughly CFD simulation technique. On the other hand, propeller scale effect is one of the old but the most difficult problems. The importance remains still unchanged.
The scale effect affecting performance characteristics are essentially viscous in nature, and it is mainly due to boundary layer phenomena dependent on Reynolds number. Okamura\textsuperscript{10} and Ito\textsuperscript{11} presented the investigation about the scale effect with boundary layer theory. Furthermore Uto\textsuperscript{12, 13} and Stanier\textsuperscript{14} recently presented the investigation by using their CFD simulation techniques. However, it is necessary to promote further verification and investigation. Consequently hub vortex and scale effect on performance characteristics of propellers still remain insufficiently solved because of its difficulty in analysis.

This paper presents a study of hub vortex and propeller scale effect taking viscous effect into consideration. The author has considered that application of CFD simulation technique is realistically the most suitable as the method for the study. As a result, the author has obtained several kinds of information on its effectiveness and further issues we have to tackle from now. This paper describes outline of the CFD simulation technique, the results and the discussions of the study.

2. Computational Method

2.1 Numerical Procedure

The Reynolds-Averaged Navier-Stokes (RANS) equations for incompressible flow are applied to analyze the viscous flow around a propeller. The numerical procedures to solve the RANS equations are as follows. Firstly the RANS equations are discretized based on the finite volume method. Next the discretized equations are computed numerically based on SIMPLE method. As the discretization method of the convection terms of the RANS equations, MARS (Monotone Advection and Reconstruction Scheme) is adopted in order to reduce the numerical viscosity. This scheme is a kind of TVD (Total Variation Diminishing) scheme. The effects of turbulence are modeled by the RNG k-ε model and the wall function boundary conditions are used. In order to implement a rotating propeller for open water tests condition, the equations in rotating reference frames are modified by taking the Coriolis and centrifugal forces into account consideration and then the source terms corresponding to these forces are added as the body forces in the relevant equations. In such a relative co-ordinate system, the reference frame of the solution domain rotates with a constant angular velocity about the axis of rotation. The space discretization is based on a block-structured grids with unstructured grids manner because the computations of the flow around complex geometry such a propeller require a highly flexible grids system.

The solution method is based on the commercial CFD software STAR-CD\textsuperscript{15}. The code has the implementation of the above-mentioned all functions and has been tested intensively on versatility and accuracy of the propeller flow computations.

2.2 Grid Generation

One of the difficulties in the computations of viscous flow around a propeller is due to a complexity of the propeller geometry when we generate the appropriate grids to obtain higher accurate solutions. Compared to other lifting bodies, such wings on airplanes, for example, there are the following peculiar geometry of a propeller:

- Periodicity in circumferential direction,
- Strong twisting of the central plane between the back and face surface of a blade,
- Juncture of a hub and blades,
- Mounting of boss cap.

It is necessary to generate the suitable grids that keep the orthogonality as possible in order not to degrade the computational accuracy and efficiency (convergence rate). Thus the computation domain around a propeller was divided optimally into many blocks and the suitable grids were generated at each block by using the unstructured grid technique. That is, the grids were generated by the following manners:

- O-type grid system for the group of blocks around the blade surfaces,
- H-type grid system for the other group of blocks.

For the computations of viscous flow around a propeller for open water tests condition in an uniform flow, only one blade was modeled and the periodic boundary conditions were employed at the outer boundaries in circumferential direction (so-called sector model).

The above-mentioned grid generation was carried out effectively by using the advanced grid generation software GRIDGEN\textsuperscript{16}. It is necessary to generate the cells less distorted and with appropriate density of
grids according to the gradient of velocity and pressure. Thus this grid generation software allowed us to adjust flexibly the orthogonality and the smoothness of grid intervals to obtain as accurate solutions as possible.

2.3 General Conditions

The propeller geometries computed in this paper are the Seiun-maru conventional propeller (CP, MAU type) and the highly skewed propeller (HSP)\(^\text{[7]}\). Table 1 shows the principal particular of these propellers. Fig. 1 shows the distribution of the propeller surface grids. Fig. 2 shows the arrangement of the global grids. The extents of the computational domain were set as follows. The inlet, outlet and circumferential boundaries were located at \(3.0 D\), \(7.0 D\) and \(5.0 R\) respectively from the propeller center point which is the point of intersection of the generator line and the axis of rotation (\(D, R\): diameter and radius of a propeller respectively). The numerical grids consisted of 46 blocks and included about 260,000 cells.

In order to reduce the time needed to obtain a solution, the computations were carried out by using the parallel computing technique, which each computational domain was assigned to each CPU. Thus the computations were performed with the parallel computer equipped the two sets of 1GHz Intel CPU. For the reference, the required CPU time was about 6.0 hours per condition.

Table 1  Principal particulars of Seiun-maru propellers.

<table>
<thead>
<tr>
<th>Type of Propeller</th>
<th>CP</th>
<th>HSP</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter [m]</td>
<td>3.600</td>
<td>3.600</td>
</tr>
<tr>
<td>Pitch Ratio at 0.7R</td>
<td>0.950</td>
<td>0.944</td>
</tr>
<tr>
<td>Expanded Area Ratio</td>
<td>0.650</td>
<td>0.700</td>
</tr>
<tr>
<td>Bass Ratio</td>
<td>0.1972</td>
<td>0.1972</td>
</tr>
<tr>
<td>Number of Blades</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Blade Thickness Ratio</td>
<td>0.0442</td>
<td>0.0496</td>
</tr>
<tr>
<td>Skew Angle [deg.]</td>
<td>10.5</td>
<td>45.0</td>
</tr>
<tr>
<td>Rake Angle [deg.]</td>
<td>6.0</td>
<td>-3.03</td>
</tr>
<tr>
<td>Blade Section</td>
<td>MAU</td>
<td>Modified SRI-B</td>
</tr>
</tbody>
</table>

Fig. 1  Distribution of grids on surface of propellers.

Fig. 2  Arrangement of global grids around propeller.

3. Flow around Hub Vortex

3.1 Objectives

Fig. 3 shows a hub vortex cavitation observed at a cavitation tunnel. It is likely that the hub vortex cavitation is generated by a large vortex at the aft end of a boss cap, which is formed by twisting vortex filaments from the blade roots, as Fig. 4\(^\text{[8]}\) shows. Hitherto, we have had a little interest in the flow around a boss cap, which has been considered to exert a little influence on the propeller performance characteristic. However, the flow around a boss cap produces important effect on the propeller performance characteristic, as effectiveness of such energy-saving devices as PBCF or Kawasaki RBS-F shows.

Nevertheless, there is insufficient investigation of the flow around a boss cap. We can find out only Ouchi\(^\text{[7]}\) and Atlar\(^\text{[9]}\) experimental investigation and Yang\(^\text{[8]}\) numerical analysis based on inviscid theory.
Since the hub vortex is strongly influenced by the viscosity of fluid, it is difficult to analyze the flow around a boss cap. In order to promote the investigation in this field, it is necessary to introduce the computation taking viscous effect into consideration. Thus it is the purpose of this paper to investigate the structure of the flow around a hub vortex and a boss cap by using the CFD simulation technique.

Fig. 3 Photograph of hub vortex cavitation of HSP.

Fig. 4 Sketch patterns of hub vortex[18].

3.2 Conditions

In this paper, Seium-maru HSP was selected as the subject for the investigation. Two boss caps were prepared, one is cone type and another truncated type. Fig. 5 shows the principal dimensions of these boss caps. The truncated type was modeled after the actual boss cap of the ship. The cone type was designed temporarily. Furthermore, the propeller boss cap on the upstream side used in open water tests was modeled after the cone type boss cap. The diameter of the propeller was 0.22 meter in model scale. The inlet boundary condition was applied as uniform flow. The propeller running conditions were from the lower advance coefficient: $J$ as higher loading condition to the higher $J$ as lower loading condition. Further, Reynolds number: $R_{n,D}$ defined by the following formula was about $4.3 \times 10^5$:

$$ R_{n,D} = \frac{nD^2}{\nu} $$

(1)

Where, $n$: number of revolution of propeller, $\nu$: coefficient of kinematic viscosity.

Fig. 5 Definition of cone and truncated boss caps in full scale.

3.3 Results

Fig. 6 shows the streamlines around the boss caps. Fig. 7 shows the limiting streamlines on the surface of the propellers. We can find that the streamlines of both the boss cap types are twisted toward the same direction to the propeller rotating direction with going downstream from the blade roots to the aft end of the boss caps and the hub vortex is formed behind the boss caps. Also the rotating direction of the hub vortex is opposite to that of the tip vortex. These phenomena agree well with the results from the viewpoint of the lifting line theory. It is found that the location where the streamlines around the truncated boss cap are twisting together is farther aft from the aft end of the boss cap than that of the cone type. And also the flowing direction of the limiting streamlines on the aft end face of the truncated boss cap is the same direction with the propeller revolution. Fig.
8 shows the circumstances of the flow visualization around the boss cap of the similar propeller by the tuft method\(^8\). We have observed that the tufts near the surface are bending in the same direction with the rotating direction of the propeller. This observation corresponds very well to the computed results. Fig. 9 shows the contour of the pressure coefficient: \( C_{pn} \) and the velocity vectors on the surface perpendicular to the revolution of the propeller. The definition of \( C_{pn} \) is as follow:

\[
C_{pn} = \frac{p - p_0}{\frac{1}{2}\rho(VnD)^2}
\]

(2)

Where: \( p \): static pressure, \( p_0 \): standard pressure, \( \rho \): density of water. These figures show that the boundary layer flows generated on the surface of the boss cap are going downstream toward the aft end of the boss cap. In case of the cone type there is the stagnation domain in the aft end of the boss cap. On the other hand, behind the aft end face of the truncated type there is the swirling dead water domain. Also in both the types there are the minimum pressure regions behind the aft end of the boss caps. Fig. 10 shows the pressure distribution along the axis of revolution. The horizontal axis shows the values of the downstream distance from the aft end of the boss cap, which are non-dimensionalized by the diameter of the propeller. These figures show data under the various propeller loading conditions. Though the location of the minimum pressure of the truncated type is farther downstream than that of the cone type, in case of the cone type the magnitude of the negative pressure is larger and the pressure recovers more rapidly with going downstream than those of the truncated type. Fig.
11 shows the changes of the minimum pressures with the advance coefficient for each boss cap type. We can find that the truncated type is favorable for the lower inception of the hub vortex cavitation. Fig. 12 shows the non-dimensionalized vorticity around the axis of revolution $\omega_n/\omega$. In case of the cone type, there are the peak values of the vorticity at the aft end of the boss cap and the vorticity is attenuated more rapidly than that of the truncated type. Fig. 13 shows the changes of the maximum vorticity with advance coefficients for each type. We can find that the maximum vorticity becomes larger with increasing of the propeller loading and the values of the cone type are larger than those of the truncated type. Therefore the magnitude of negative pressure of the truncated type becomes smaller than that of the cone type.

Fig. 10 Pressure distribution along axis of revolution.

Fig. 11 Changes of minimum pressure with advance coefficient.

Fig. 12 Vorticity distribution along axis of revolution.

Fig. 13 Changes of maximum vorticity with advance coefficients.
3.4 Discussions

The results have revealed the structure of the flow around the cone and truncated boss caps. Let us study the mechanism why the minimum pressure and the maximum vorticity of hub vortex of the truncated type are smaller than those of the cone type. Fig. 14 shows the stream of particles, of which the radius of a sphere is in proportion to magnitude of vorticity along the representative streamlines of both the types. We find that the vorticity generated from the blade roots increases involving the vortex in the boundary layer on the hub and boss cap. In case of the cone type, these vortices are concentrated at the aft end of the boss cap and then the hub vortex with large vorticity is generated. On the other hand, in the truncated type, because of the boss cap surface terminated suddenly on the way to the downstream, the flow separation is occurred and the development of the boundary layer is stopped, and the vortices are diffusing rapidly and flowing out. Therefore we guess that the hub vortex of the truncated type is weakened. Thus we expect that weakening the hub vortex will reduce the rotating energy loss with the truncated boss cap. Fig. 15 shows the propeller performance characteristics in open water of both the boss cap types. We can find that the propeller efficiency of the truncated type is a little larger than that of the cone type. Fig. 16 shows the measured propeller performance characteristics of the similar propeller\(^5\), and the propeller efficiency of the truncated type is a little higher than that of the cone type.

Consequently we have obtained the fundamental knowledge of the flow around the boss caps and the hub vortex not revealed analytically so far. These knowledge are useful for designing better PBCF or Kawasaki RBS-F.

Fig. 14 Stream of particles, of which radius of sphere is in proportion to magnitude of vorticity \((J=0.30)\).

![Diagram showing stream of particles](image)

Fig. 15 Comparison of propeller open performance characteristics.

![Graph showing propeller performance characteristics](image)

Fig. 16 Comparison of open performance characteristics of similar propellers\(^5\).

4. Scale Effect of Propeller Performance

4.1 Objectives

The scale effect affects the propeller performance characteristics and further the accuracy of ship power estimation considerably. The scale effect is essentially viscous in nature, and it is mainly due to boundary layer phenomena dependent on Reynolds number. Namely the flows over the blades of a model propeller \((D = 200 \text{–} 300\text{mm})\) in open water tests are involved with the laminar boundary layer as about 50 – 60% of the blade area. On the other hand, the turbulent boundary layer flows are developed fully over the
blades of full scale propellers. This subject is still one of the old but the most difficult problems even now and we have no definite view of our own. So far the scale effect of the propeller performance is integrated into the estimated values with the empirical coefficients. Thus the correction methods are somewhat in unreasonable aspect. It is to be desired that we establish the more reasonable estimation methods of the scale effect.

In the last a few decades, there were some studies on the propeller scale effect integrated with the lifting surface theory and the boundary layer theory. Nevertheless, it is necessary to deal with hydrodynamical characteristic more exactly. Consequently there were some study reports by using the RANSE codes, but there was no report with sufficient validation and verification. This paper describes the more reasonable estimation method of the viscous scale effect of the propeller performance characteristic by using CFD simulation technique. Furthermore difference of the scale effect due to the propeller geometry is discussed.

4.2 Conditions

The propeller geometries computed in this paper are the conventional propeller (CP, MAU type) and the highly skewed propeller (HSP) equipped to the training ship 'Seiun-maru'. The Kemp's Reynolds number: \( R_n(K) \) defined by the following formulation is introduced as an affecting index of the scale effect.

\[
R_n(K) = \frac{C_{0.7} \sqrt{V^2 + (0.7\pi n D)^2}}{\nu}
\]  

(3)

Where, \( C_{0.7} \): chord length at 0.7 \( R \) in radius, \( V_\alpha \): propeller speed of advance.

The computations were carried out in Reynolds numbers between model and full scale by using the grid arrangement of model scale, provided that the propeller in every scale has turbulent flow over the blade surface. Consequently, however, the minimum spacing of grids: \( y^+ \) were set about 7 to 170 in model scale to full scale respectively. The inlet conditions were employed with uniform flow. The loading conditions of the propellers were unified \( J=0.60 \) as the design point.

4.3 Results

Fig. 17 shows changes of thrust coefficient: \( K_T \), torque coefficient: \( K_Q \) and open propeller efficiency: \( \eta_0 \) of each propeller with \( R_n(K) \). The filled and emptied marks denote the computed and measured data \(^{19}\) respectively. However there are two groups of measured data due to the difference of the dynamometers. We can find that the computed data show a tendency to agree with the measured ones as increasing Reynolds number. These results show that while the laminar boundary layer is predominant over the blade surface of model scale, as increasing Reynolds number the turbulent boundary layer is predominant. Among the measured data, however, there are somewhat scattering, we find roughly that HSP is more affected by the scale effect than CP. Also the computed data have the similar tendency.

Fig. 18 shows the ratio of the \( K_T, K_Q \) and \( \eta_0 \) to those of model scale. The suffix \( m \) indicates the values of model scale. The filled and emptied marks denote the values of HSP and CP respectively. We find that the ratio of \( K_T \) of HSP changes the most rapidly. Thus the scale effect affects somewhat more favorably open propeller efficiency of HSP than that of CP. Fig. 19
Fig. 18 Ratio of propeller performances to those of model scale.

Fig. 19 Pressure and frictional components of thrust and torque coefficients.

Fig. 20 Ratio of pressure and frictional components to those of model scale.

shows the pressure and the frictional components of thrust and torque coefficients. In this figure suffix p and f denote the pressure and frictional components respectively. Fig. 20 shows the ratio of each component to that of each propeller in model scale. From this figure, we find that the scale effect affects seriously the frictional component of both the thrust and torque. Also the pressure components are a little affected by the scale effect. Furthermore the pressure components of HSP are affected more seriously than those of CP.

Figs. 21 and 22 show the pressure contour on the

(a) Model scale

(b) Full scale

Fig. 21 Pressure contour on blade surface of CP (\(J=0.60\)).
face and back side of the blade surfaces of model and full scale. We find that the difference between model and full scale of HSP are more extensive than those of CP. Especially there are remarkable difference in the regions of the blade roots and the trailing edge. Figs. 23 and 24 show the limiting streamlines on the blades of model and full scale. The arcs in these figures are located at the 40%, 70% and 100% of propeller radius respectively. It can be seen that there are the three-dimensional separation bubbles spread toward the tip in the region of trailing edge of the blade root of both CP and HSP. Also there is no leading edge separation as the loading conditions are set to the design point. Besides we find that there are little difference between model and full scale in case of CP, while remarkable difference in HSP. Because the blade of HSP is a kind of swept wing, the flow near the bottom of boundary layer with declined momentum drifts toward the blade tip, and further the boundary layers are likely to be developed. Therefore HSP tend to be affected strongly by the scale effect than CP.

![Fig. 22 Pressure contour on blade surface of HSP(J=0.60).](image)

![Fig. 23 Limiting streamlines on blade surface of CP(J=0.60).](image)

![Fig. 24 Limiting streamlines on blade surface of HSP(J=0.60).](image)

4.4 Discussions

Judging from the validation with the comparison to the measured data, the estimation method of the scale effect affecting the propeller performance characteristics ($K_T$, $K_Q$ and $\eta_p$) by using the CFD simulation technique in this paper is very effectual. Namely CFD analysis of the turbulent flow in the full scale is able to estimate the propeller performances of the full scale. In the future it is necessary to secure the suggestion by increasing the proven results furthermore.

Regarding the difference of the scale effect based on the propeller geometry, it is concluded that HSP is more affected by the scale effect than CP. The measured data also show the similar tendency. Furthermore, provided that the propeller performance characteristic is divided into the pressure and frictional components, the frictional component is more affected by the scale effect than the pressure component that has rather the scale effect slightly.

5. Conclusions

This paper has presented development of the rational method to analyze the viscous flow around marine propellers by using CFD simulation technique based
on RANS equations. Several helpful guidelines for the grid generation, which is one of the difficulties in CFD analysis of marine propeller, have been proposed. Using this analytical method, we have found the detailed structure of the flow around the propeller boss caps and the hub vortex and the scale effect affecting the propeller performance characteristic, which were difficult to be analyzed in the past. The method is very valuable for feasible study and design of energy-saving devices to depress the hub vortex and also for improvement of the ship power estimation accuracy. To make this method further more mighty design tool in the future, it is necessary further to reduce the time for the grid generation and to validate the analytical method more extensively.

Acknowledgements

The author would like to thank Dr. Tamashima for providing the materials concerned with hub vortex and Prof. Yamano for many stimulating discussions.

References

16) http://www.pointwise.com/