Investigation on hydrodynamic performance of a marine propeller in oblique flow by RANS computations

Jianxi Yao

Technical University of Berlin, Germany

ABSTRACT: This paper presents a numerical study on investigating on hydrodynamic characteristics of a marine propeller in oblique flow. The study is achieved by RANS simulations on an open source platform - OpenFOAM. A sliding grid approach is applied to compute the rotating motion of the propeller. Total force and moment acting on blades, as well as average force distributions in one revolution on propeller disk, are obtained for 70 cases of combinations of advance ratios and oblique angles. The computed results are compared with available experimental data and discussed.

KEY WORDS: Propeller; Hydrodynamic performance; Oblique flow; RANS simulation; OpenFOAM.

INTRODUCTION

Numerous studies based on experiments or computations have been carried out to investigate propeller open water characteristics, which are main ways to access propeller hydrodynamic performance in open water currently. Most studies only considered the case of propeller in straight ahead flow. However, under real operating conditions working propeller behind ship usually locates in so complicated wake that the propeller shows quite different hydrodynamic performance. This is mainly due to the disturbance of ship hull to inflow before propeller, which results in each blade angle of attack of propeller becomes different and the loads acting on blades in axial direction is not symmetric anymore. So that it is necessary to study the cases of propellers in complicated flow, so as to obtain a fuller knowledge and understanding of propeller hydrodynamic performance.

On the other hand, in recent years numerical methods based on solving Reynolds-Averaged Navier-Stokes (RANS) equations have been made remarkable progress and widely applied to address problems in ship hydrodynamics, such as manoeuvre. For instance, Cura-Hochbaum (2006) predicted ship manoeuvring behaviours based on the determination of manoeuvring derivatives by means of virtual PMM (Planar Motion Mechanism) tests. A similar study was also done by Simonsen et al. (2012) who performed the standard deep water IMO (International Maritime Organization) manoeuvring simulations for a container ship model with the hydrodynamic coefficients obtained from both RANS computations and measurements. In their simulations, effects of real working propeller-on flow were approximated by either a simple prescribed body force model or a model based on a database consisting of force distributions on propeller disk for a range of advance ratios, as using real discretized propeller usually requires small length of time step, making the simulations extremely time-consuming. The experience from the first workshop on verification and validation of ship manoeuvring simulation methods-SIMMAN (Stern et al., 2008) has shown that present body force models used are required to be improved for highly accurate simulations. Therefore, in
order to develop more sophisticated body force models insightful knowledge of force distributions on propeller disk is needed when propeller rotates in complicated flow.

On above points of view, an investigation on hydrodynamic characteristics of a marine propeller in oblique flow is made in this study. Only few authors have reported on this aspect until now. Dubbioso et al. (2013; 2014) analyzed performance of a marine propeller in oblique flow by RANS computations. In their work the hydrodynamic forces acting on blades and flow features around the propeller were obtained and discussed in details. Unfortunately the simulations were not validated due to lack of experimental data. Krasilnikov et al. (2009) used an unsteady RANS method to investigate on propeller blade forces for a podded propeller operating in pulling and pushing modes in oblique flow. They found by the analyses that blades of pulling propeller experience comparable amplitudes and load levels at positive and negative heading angles being mainly affected by the cross flow. Shamsi and Ghassemi (2013) studied the effects of oblique angles on characteristics of podded propulsions. A RANS method with Moving Reference Frame (MRF) was used in their study. The predicted hydrodynamic forces showed good agreements with experimental data.

The present work is achieved by using the unsteady RANS solver in an open source Computational Fluid Dynamics (CFD) toolbox named OpenFOAM to simulate the cases of combinations of advance ratios and oblique angles. Seventy combinations are considered. During the simulations, Menter \( k-\omega \) SST model (Menter and Kuntz, 2003) without wall function is used to model turbulence and an approach of sliding grid is applied to compute rotating motions of the propeller. Total force and moment acting on blades, as well as average force distributions in one revolution on propeller disk, are obtained. The computed results are compared with available experimental data and discussed.

OUTLINE OF METHOD

Mathematical equations

A right-handed coordinate system is used to describe the rotating motion of the propeller, where the origin \( o \) locates at the centre of the propeller, \( x \)-axis towards cap as depicted in Fig. 1 and \( z \)-axis vertically downward. Since the rotating motion in three-space is symmetric with respect to \( x \)-axis, the oblique angle \( \alpha \) could be generally defined as the angle from \( x \)-axis to the direction of free-stream flow. It is clear that the range of \( \alpha \) is from 0° to 180°. Here \( \alpha \) is defined in \( o-xy \) plane without loss of generality.

Under the assumption of incompressible Newtonian fluid the flow around the propeller has to satisfy conservation equations of mass (continuity equation) and momentum (RANS equations), which could be written in tensor notation as

\[
\frac{\partial U_i}{\partial x_i} = 0
\]

\[
\frac{\partial (U_i U_j)}{\partial t} + \frac{\partial (U_i U_j)}{\partial x_j} = f_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + \tau_{ij}
\]

where \( x_i = (x, y, z) \) are independent coordinates, similarly \( U_i = (U, V, W) \) are Reynolds-Averaged velocity components, \( \rho \) is pressure, \( \rho \) is water density, \( \nu \) is kinematic viscosity coefficient, \( f_i \) are body force components and \( \tau_{ij} \) is Reynolds stress tensor resulting from the time-averaged procedure to N-S (Navier-Stokes) equations. The Reynolds stress tensor is expressed under Bousinesq’s eddy viscosity hypothesis as

\[
\tau_{ij} = \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} k
\]

where \( \nu_t \) is eddy viscosity, \( \delta_{ij} \) is Kronecker symbol and \( k \) is turbulent kinetic energy. The body force \( f_i \) are zero in present simulations.
The Menter $k$-$\omega$ SST model (Menter, 2003) without wall function is applied to approximate the eddy viscosity in equation (3) where SST is the acronym of Shear-Stress-Transport and $\omega$ is specific dissipation rate. The transport equations are as follows.

$$\frac{\partial(k)}{\partial t} + \frac{\partial(U_i k)}{\partial x_i} = \tilde{P}_k - \beta' k \omega + \frac{\partial}{\partial x_i} \left[ (v + \sigma_k v_i) \frac{\partial k}{\partial x_i} \right]$$

$$\frac{\partial(\omega)}{\partial t} + \frac{\partial(U_i \omega)}{\partial x_i} = \frac{\alpha_P}{v_i} - \beta \omega^2 + \frac{\partial}{\partial x_i} \left[ (v + \sigma_\omega v_i) \frac{\partial \omega}{\partial x_i} \right] + 2(1 - F_1) \sigma_\omega \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}.$$  

Here the functions $F_1$ and $F_2$ are defined by

$$F_1 = \tanh \left\{ \min \left[ \max \left( \frac{\sqrt{k}}{\beta' \omega y_w}, \frac{500 \nu}{y_w^* \omega} \right), \frac{4 \rho \sigma_{\omega k}^2}{C_{D_{\omega \omega} y_w^*}} \right] \right\}^4,$$

$$F_2 = \tanh \left\{ \max \left( \frac{2 \sqrt{k}}{\beta' \omega y_w}, \frac{500 \nu}{y_w^* \omega} \right)^2 \right\}$$

with

$$C_{D_{\omega \omega}} = \max \left( 2 \rho \sigma_{\omega k} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-20} \right)$$

and $y_w$ the distance to wall. The turbulent eddy viscosity is defined by

$$v_i = \frac{a_{ik}}{\max(a_{ik} , SF_i)},$$

where
\[ S = \sqrt{2S_y S_y}, \quad S_y = \frac{1}{2} \left( \frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right). \]

\( S_y \) is the strain rate tensor and \( a_1 = 0.31 \) is a constant. And the production of turbulent kinetic energy is

\[ P_t = \tau_y \frac{\partial U_i}{\partial x_j} \to \bar{P}_t = \min(P_t, 10 \cdot \beta^2 k \omega) \]

All constants are computed by a blend from the corresponding constants via \( c = c_1 F_1 + c_2 (1 - F_1) \). The values of model constants are specified to the default values given in OpenFOAM, which are: \( \beta^* = 0.09 \), \( \sigma_{\alpha_1} = 0.553 \), \( \beta_1 = 0.075 \), \( \alpha_2 = 0.44 \), \( \beta_2 = 0.0828 \), \( \sigma_{\alpha_1} = 0.85 \), \( \sigma_{\omega_1} = 0.5 \), \( \sigma_{\omega_2} = 1 \), \( \sigma_{\omega_2} = 0.856 \).

**Numerical method**

The RANS solver in OpenFOAM (2014) is based on a finite volume technique which permits use of arbitrary polyhedral grids including hexahedron, tetrahedron, prism, and so on. A suite of basic discretization schemes and solution algorithms are available. In present applications, a second order upwind difference scheme and a central difference scheme are selected to approximate convective terms and diffusive terms respectively. Turbulence is discretized with a second order upwind difference. The PIMPLE algorithm which merges PISO (Pressure Implicit with Splitting of Operators) and SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithms is employed to couple the mass and momentum conversation equations. The systems of linear equation resulting from the discretized equations are solved by using the iterative solvers, Gauss-Seidel or generalized geometric multi-grid (GAMG).

**Sliding grid approach**

The capability in dealing with sliding grid is available in OpenFOAM. This technique is quite useful for simulations involving rotating parts e.g. rotating propeller. With the sliding grid approach the computational domain used may be divided into two parts: an outer static part and an inner dynamic part. The inner part is cylinder-shaped and could rotate relatively to the outer part. The two parts are connected by an interpolation scheme by which flow information is transformed between relative sliding interfaces. Fig. 2 shows partial view of an unstructured grid in a cut plane. As seen the inner part includes the geometry of a propeller and grid elements in it rotate during simulations. The red circle in the figure represents sliding interfaces on which the interpolation scheme is imposed via setting a boundary condition named Arbitrary Mesh Interface (AMI) in OpenFOAM.

![Fig. 2 Sliding grid in a cut plane.](image)
Table 1 Main particulars of the propeller.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter</td>
<td>8.3 m</td>
</tr>
<tr>
<td>Pitch ratio (mean)</td>
<td>1.03</td>
</tr>
<tr>
<td>Expanded area ratio</td>
<td>0.8034</td>
</tr>
<tr>
<td>Number of blade</td>
<td>6</td>
</tr>
<tr>
<td>Hub ratio</td>
<td>0.18</td>
</tr>
<tr>
<td>Direction of rotation (looking upstream)</td>
<td>Right-handed</td>
</tr>
</tbody>
</table>

GEOMETRY, GRID, BOUNDARY CONDITION AND GRID, TIME-STEP DEPENDENCY STUDIES

A propeller having 6 blades is taken as an example. The geometry of the propeller is shown in Fig. 3 and main particulars at full scale are listed in Table 1. A model with scale 39.885 has been tested in the cavitation tunnel at TUB (Technical University of Berlin, Germany).

According to the experiences from the 2nd International Symposium on Marine Propulsions, a rational computational domain is chosen for the computations, which is limited in a box with \(-7.0D < x < 3D, -5.0D < y < 5.0D\) and \(-5.0D < z < 5.0D\). Fig. 4 shows the computational domain and boundary conditions (BC). BCs are summarized as follows.

- **Inlet BC**: at boundaries \(x = 3D, z = \pm 5D\) and \(y = -5D\), the free-stream values of velocity, \(k\) and \(\omega\) are specified, while zero gradient is given to pressure. The free-stream values of \(k\) and \(\omega\) could be estimated by following formulas.

\[
k_m = \frac{3}{2} \left( I_{um} \right)^2, \quad \omega_m = \frac{k_m^{1/2}}{C_\mu^{1/4} I_t}
\]

where \(I_t\) in range of \(1\% \sim 10\%\) is the turbulent intensity as a fraction of mean flow velocity \(u_m\), \(C_\mu = 0.09\) is a constant and \(I_t\) is the turbulent length scale often close to the cell size.

- **Outlet BC**: at boundaries \(x = -7D\) and \(y = 5D\), specify pressure and normal gradients of flow velocity, \(k\) and \(\omega\) to 0.
- **Wall BC**: on blades, cap and shaft no-slip condition is set, i.e. flow velocity and gradients of pressure, \(k\) and \(\omega\) are specified to 0.
- **AMI BC**: set AMI BC at the interfaces connecting the static and dynamic parts.
Due to the complicated configuration of the propeller a hybrid grid is employed. An unstructured grid is generated for the inner part, while multi-block structured grid for the outer part. The surface grid on blades consists of triangular elements and three mesh layers are attached at blade surfaces. The grid in the inner part is composed of triangular prisms in the three mesh layers and tetrahedrons in remaining space. The advantage of the mesh layers is that it is convenient to adjust spacing of first grid point to wall, but avoid strong increase of cell number. However, a special attention should be paid to the dimensionless distance to wall ($y^+$) when generating grid since no wall function is used in this study. According to amount of pre-computations it has been found that specifying the spacing of first grid point to wall to 0.1 mm could get an appropriate $y^+$ (around 1) in present simulations. Fig. 6 shows surface grids on a blade of a coarse, medium and fine grid, which has roughly 1.5 million, 2.6 million and 5.2 million cells respectively. The grid in a mid-cut plane of the coarse grid is shown in Fig. 5.

The simulation accuracy not only depends on grid resolution but also the length of time step used. In particular for cases of considering dynamic grid, time-step length would also affect significantly on numerical stability. For present cases it has been found that performing 50 time steps in one propeller revolution could ensure numerical stability and become instable using 25 time steps. During the simulations in each time step it applies 10 outer loop iterations, and in each iteration pressure is corrected twice. Note that if using smaller length of time step less number of iterations and pressure correction may also ensure accuracy.

Grid and time-step dependency studies are usually necessary for reliable numerical results. For this purpose, computations are conducted on the coarse, medium and fine grid respectively for a selected case of $n=50$ rps, $u_0=4.162$ m/s and $\alpha=0^\circ$ which correspond to $J=0.4$ and a Reynolds number $R_e=1.24\times10^6$, where $n$ is revolution rate, $u_0$ flow velocity, and $J$ advance ratio. The present Reynolds number is defined by $MWR \times D \sqrt{(0.7\pi n D)^2 + u_0^2} / \nu$ where $MWR=0.2565$ is mean blade width ratio. In addition, for time-step dependency study computations are carried out on the medium grid for the same case, but performing different Number of time Step (NS) in one propeller revolution. The computations for grid dependency study perform 400 time steps in one propeller revolution, and for the time-step dependency study it keeps everything unchanged but only varies NS which is specified to 50, 100, 200, 400 and 800 here.
Table 2 Summary of thrust and torque obtained from the grid and time-step dependency studies for the case of $J = 0.4$ and $\alpha = 0°$.

<table>
<thead>
<tr>
<th>Grid</th>
<th>NS</th>
<th>Thrust [N]</th>
<th>Torque [Nm]</th>
<th>Err (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$T_{pre}$</td>
<td>$T_{vis}$</td>
<td>$T_{tot}$</td>
</tr>
<tr>
<td>Grid dependency study</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Coarse</td>
<td>400</td>
<td>1756.34</td>
<td>-13.67</td>
<td>1742.67</td>
</tr>
<tr>
<td>Medium</td>
<td>400</td>
<td>1773.02</td>
<td>-15.36</td>
<td>1757.66</td>
</tr>
<tr>
<td>Fine</td>
<td>400</td>
<td>1783.76</td>
<td>-17.08</td>
<td>1766.68</td>
</tr>
<tr>
<td>Time-step dependency study</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Medium</td>
<td>50</td>
<td>1570.71</td>
<td>-15.89</td>
<td>1554.82</td>
</tr>
<tr>
<td>Medium</td>
<td>100</td>
<td>1717.47</td>
<td>-15.42</td>
<td>1702.05</td>
</tr>
<tr>
<td>Medium</td>
<td>200</td>
<td>1759.71</td>
<td>-15.34</td>
<td>1744.37</td>
</tr>
<tr>
<td>Medium</td>
<td>400</td>
<td>1773.02</td>
<td>-15.36</td>
<td>1757.66</td>
</tr>
<tr>
<td>Medium</td>
<td>800</td>
<td>1777.20</td>
<td>-15.41</td>
<td>1761.79</td>
</tr>
</tbody>
</table>

For all above computations, the resulting maximum Courant numbers are between 13 and 60, but mean values are all less than 0.1. It seems the present length of time step and cell size can maintain computational stability. The computed thrust and torque acting on blades are summarized in Table 2. It is seen from the grid dependency study that viscous thrust and torque ($T_{vis}$ and $Q_{vis}$) are more greatly influenced by grid revolution than pressure thrust and torque ($T_{pre}$ and $Q_{pre}$). There is still a discrepancy of about 10% between the computed $T_{vis}$ based on the medium and fine grids, whereas only 0.6% between $T_{pre}$. Fortunately the viscous thrust and torque account for a quite small proportion (less than 1%) of the total thrust and torque ($T_{tot}$ and $Q_{tot}$), which result in that the total thrust and torque are not sensitive to grid revolution.

In the time-step dependency study, it is found that as increase of NS the pressure thrust and torque change smaller and smaller, while the vicious thrust and torque are nearly unchanged. When NS reaches 200, the changes of total thrust and torque are less than 1%.

In order to illustrate clearly, the thrust convergence curves obtained from the computations of grid and time-step dependency studies are shown in Fig. 7. It is obviously concluded that all computations reach steady-state after 3 propeller revolutions.
RESULT AND DISCUSSIONS

Seventy cases of combinations of $J$ and $\alpha$ are considered, where $J$ is varied from 0.1 to 1.0 with an interval of 0.1 and $\alpha$ from 0° to 30° with an interval of 5°. For all computations the magnitude of inflow velocity remains unchanged (here 4.162 m/s) and propeller revolution rate is determined for desired $J$s according to $J=U_0/(nD)$. The Reynolds numbers of these cases are between $0.537 \times 10^6$ and $1.24 \times 10^6$. In consideration of large amount of cases, all computations are carried out on the medium grid and perform 200 time steps in one propeller revolution. Based on the grid and time-step dependency studies in previous section, it could be expected that the discrepancy between the present numerical results and the results obtained by using the fine grid and larger number of time step should be within 2%.

![Fig. 8 Sketch of an area-weighted interpolation scheme. The shaded trapezoid represents an element on propeller disk and triangles are the projection of surface grid elements of blade on the disk.](image)

In order to obtain average force distributions on propeller disk in one propeller revolution for all cases, each computation performs 4 revolutions and the pressure force and viscous stress acting on grid elements on one blade are written out every two time steps in the fourth revolution. This means there are 100 outputs for each computation. Then for each case the 100 force distributions on the blade are transformed onto propeller disk via an area-weighted interpolation scheme as sketched in Fig. 8, and averaged in the revolution. Note that only the force on one blade is outputted but not on all blades, as each blade experiences same loads at same phase in the rotating motion. Thus the average force distribution what we want exactly is determined by simply multiplying the number of blade with the average force distribution obtained from one blade.

Fig. 9 shows computed hydrodynamic force and moment versus $J$ for a range of oblique angles. The force and moment are made dimensionless by

$$T'_i = \frac{T_i}{\rho n^2 D^4}, \quad Q'_i = \frac{Q_i}{\rho n^2 D^5} \quad (i = x, y, z).$$

The propeller efficient is expressed by

$$\eta = \frac{J}{2\pi} \frac{T'_x}{-Q'_x}.$$

As seen in the figure, with the increase of advance ratio the thrust ($T'_x$) decreases but the side force ($T'_y$) and vertical force ($T'_z$) become larger. For the moments, the torque ($Q'_x$) decreases as well and ($Q'_y$) increases, but ($Q'_z$) increases first and then decreases. It is also noted that the influences of oblique angle on force and moment become larger and larger when advance ratio increases, and with the decrease of advance ratio the influences reduce to zero gradually. This is quite understandable and could be explained by making use of velocity vectors for a section of a blade shown in Fig. 10. In the figure $\pi Dn$ represents circumferential velocity. It is no doubt that for a fixed revolution rate the level of the oblique angle impact on
loads acting on the blade section is determined by the value of $|\alpha_1 - \alpha_2|$, where $\alpha_1$ and $\alpha_2$ are effective angles of attack. Larger $|\alpha_1 - \alpha_2|$ means larger influences, and if $|\alpha_1 - \alpha_2|$ is zero no influence occurs. On the other hand, in present considerations large advance ratio corresponds to small revolution rate, since the free-stream velocity $u_0$ keeps constant and $J = u_0/(nD)$. If oblique angle is fixed and revolution rate goes to infinity, both advance ratio and $|\alpha_1 - \alpha_2|$ tend to zero, which means influences of oblique angle on thrust and torque become smaller when advance ratio decreases. In contrast, if revolution rate tends to zero, advance ratio tends to infinity and $|\alpha_1 - \alpha_2|$ increases, resulting in larger influences when advance ratio increases. From another point of view, if advance ratio is changed through the free-stream velocity but not the propeller revolution rate, advance ratio could be zero corresponding to zero velocity of free stream. There is of course no influence of oblique angle on thrust and torque when advance ratio is zero.

Fig. 9 Computed force and moment versus $J$ for different $\alpha$s.
With above analyses it could be also to explain the patterns of force distributions on propeller disk. Fig. 11 shows the force distributions of the three components of force on the disk for the cases of $J = 0.7$ and $\alpha = 0^\circ, 10^\circ, 20^\circ, 30^\circ$. It is observed that when oblique angle is zero, the axial force distribution is axisymmetric, while for the other two components there exists a pair of region on the disk, in which the distribution pattern is identical but with opposite sign. When oblique angle is not zero, the force distributions are no longer symmetric, and this trend is more obvious as oblique angle increases. It is interesting to find that for the case of $J = 0.7$, $\alpha = 20^\circ$, it appears a region of negative thrust in axial direction and this trend is becoming more and more obvious with the increase of $\alpha$. The appearance of negative thrust is due to that the blade section experiences negative effective angle of attack.

![Fig. 11 Distributions of the three components of force on propeller disk at $J = 0.7$ and $\alpha = 0^\circ, 10^\circ, 20^\circ, 30^\circ$ (from top to down).]
Fig. 12 Pressure distributions on pressure side (left) and suction side (right) at $J = 0.7$, $\alpha = 0^\circ$ (up) and $J = 0.7$, $\alpha = 30^\circ$ (down).

Fig. 12 shows contour plots of pressure distribution on pressure side (left) and suction side (right) for two cases of $J = 0.7$, $\alpha = 0^\circ$ (up) and $J = 0.7$, $\alpha = 30^\circ$ (down). The pressure is non-dimensionalized by $(p - p_h)/(0.5 \rho u_0^2)$ where $p_h = 0$ is reference pressure. The pressure distributions on blades could give us a better understanding of the average force distributions on the disk. For the case of $J = 0.7$, $\alpha = 30^\circ$, the blade located at the position of phase angle 150° defined in $o-yz$ plane experiences largest loads than other blades on pressure and suction sides, which indicates the blade experiences the largest effective angle of attack. This is accordance with the average thrust distribution of this case shown in Fig. 11.

Fig. 13 Comparison of iso-surfaces of $Q = 5000$ $s^{-2}$ between the cases of $J = 0.7$, $\alpha = 0^\circ$ (a) and $J = 0.7$, $\alpha = 30^\circ$ (b).

Fig. 13 shows a comparison of iso-surfaces of $Q = 5000$ $s^{-2}$ between the cases of $J = 0.7$, $\alpha = 0^\circ$ (up) and $J = 0.7$, $\alpha = 30^\circ$ (down). $Q$ is a scalar to visualize turbulent fields. It is defined by

$$Q = -\frac{1}{2}(S_y S_y - \Omega_y \Omega_y)$$

where

$$S_y = \frac{1}{2}(U_y - U)$$
$$\Omega_y = \frac{1}{2}(U - U_y)$$
As seen, the flow around the propeller becomes not axisymmetric in oblique flow.

Current simulations are validated through comparisons between computed results and experimental data. The computed and experimental open water curves are plotted in Fig. 14 for a range of oblique angles. It is noted that when advance ratio is larger than 0.45 the RANS results show a generally good agreement with experimental data. However, when advance ratio is smaller than 0.45 the experimental $T'$ and $Q'$ become smaller as decrease of advance ratio, while the computed $T'$ and $Q'$ show opposite characteristic. This is because when advance ratio is smaller than 0.45 (corresponding to $n = 44.44$ rps) cavitation becomes severe observed in experiments, which causes a thrust loss strongly. However in present study the cavitation is not considered. Although at $J = 0.6$ and $\alpha = 25^\circ$ a weak cavitation occurs at tips located at upstream as shown in Fig. 15, it seems the weak cavitation has no significant impacts on thrust and torque acting on blades since the computed results still show good agreements with experimental data. The comparisons also show that at small oblique angles the RANS results agree excellently with experimental data under no or weak cavitation condition, while at large oblique angles the discrepancy becomes larger. The largest error is found to be around 15%. The large discrepancy at large oblique angle may be caused by the effects of tunnel
wall, because the section size of the tunnel is $0.7 \, m \times 0.7 \, m$, and at $\alpha = 25^\circ$ the distance between propeller center and the closest wall is around $2.5D$, which may be not far enough anymore.

Fig. 15 Cavitation observed in experiment at $J=0.6$ and $\alpha=25^\circ$.

Fig. 16 shows the computed and experimental $T'_x$, $T'_y$, $T'_z$ and $Q'_z$ versus oblique angle for $J = 0.5$ to 1.0. It is seen that $T'_y$ and $T'_z$ are relatively quite small with respect to $T'_x$. There are obvious differences in $T'_y$ and $T'_z$ between simulations and experiments, especially for $T'_z$. This is probably due to cavitation.

Fig. 16 Computed and experimental $T'_x$, $T'_y$, $T'_z$ and $Q'_z$ versus oblique angle for selected advance ratios.
CONCLUSIONS

This study has used the unsteady RANS solver in OpenFOAM to investigate hydrodynamic performance of a propeller in oblique flow. A sliding grid approach is applied to compute the rotating motion. 70 combinations of advance ratios and oblique angles are considered. The current simulations are validated via comparisons between computed hydrodynamic force, moment and available experimental data. The computed results show generally good agreement with experimental data under no or weak cavitation condition. The cavitation could be considered in future work, which would be more interesting. Generally the present results could be improved by using high revolution grids and large number of time steps in one propeller revolution, for example more than 400. In addition, the averaged force distributions on propeller disk in one revolution are obtained by using an area-weighted interpolation scheme, which is a great foundation to develop better body force models and could give us a better understanding of hydrodynamic performance of the propeller in oblique flow.

ACKNOWLEDGMENTS

This work is financially supported by China Scholarship Council (CSC). The author would like to thank Professor Dr.-Ing. Andrés Cura Hochbaum and Dipl.-Ing. Johannes Beutel for helpful discussions and offering experimental data.

REFERENCES


OpenCFD Ltd., 2014. OpenFOAM user guide of version 2.3.0. Bracknell, United Kingdom: OpenCFD Ltd.

