Numerical investigation of tip clearance effects on the performance of ducted propeller

Ding Yongle, Song Baowei and Wang Peng

School of Marine Science and Technology, Northwestern Polytechnical University, Xi’an, China

Received 3 February 2015; Revised 7 April 2015; Accepted 22 April 2015

ABSTRACT: Tip clearance loss is a limitation of the improvement of turbomachine performance. Previous studies show the Tip clearance loss is generated by the leakage flow through the tip clearance, and is roughly linearly proportional to the gap size. This study investigates the tip clearance effects on the performance of ducted propeller. The investigation was carried out by solving the Navier-Stokes equations with the commercial Computational Fluid Dynamic (CFD) code CFX14.5. These simulations were carried out to determine the underlying mechanisms of the tip clearance effects. The calculations were performed at three different chosen advance ratios. Simulation results showed that the tip loss slope was not linearly at high advance due to the reversed pressure at the leading edge. Three type of vortical structures were observed in the tip clearance at different clearance size.

KEYWORDS: Tip clearance; Ducted propeller; Computational fluid dynamic (CFD).

INTRODUCTION

Ducted propeller has been widely used in vessels and Autonomous Underwater Vehicles (AUVs) for decades. The duct propeller consists of an annular wing duct and a propeller. There are mainly two kinds of duct: the accelerating and the decelerating duct based on different kind of wing section. Since the accelerating duct is the most widely used, we only discuss this kind of duct in this paper.

A lot of experimental and theoretical researches have been done on ducted propellers (Kerwin et al., 1987; Abdel-Maksoud and Heinke, 2002; Hoekstra, 2006; Hsiao and Pauley, 1999; Hughes et al., 1992). One of the primary challenges for ducted propeller is the accurate prediction of hydrodynamic characteristics. The main limitations for the experimental and numerical prediction are the complex geometry and other technical problems (Park et al., 2005; Morgut and Nobile, 2012; Berchiche and Janson, 2008; Peng et al., 2013). An important factor influencing the accurate prediction of ducted propeller performance is the tip clearance or tip gap. The blade tip clearance between a blade and the duct has been the major source of many unfavourable flow phenomena. Complicated vortical structures are generated by the mixing of the two flows: the flow from the pressure side to the suction side through the clearance and the main flow and the injection of the flow into the boundary layers on the blade tip and the duct (You et al., 2007). These vortices often lead to noise, vibration, erosion and thrust/torque loss.
Quantitative studies on tip clearance of turbomachinery have been presented (Xiao et al., 2001; McCarter et al., 2001; Booth, 1985; Yaras and Sjolander, 1992; Yaras et al., 1992). Most of these issues are focused on tip clearance flow in turbines, compressors and pumps. A basic understanding of the effect of the tip clearance is that the tip clearance generates complicated vortical structures which can result in performance decrease and rotating instabilities. In previous researches, the tip clearance loss grows roughly linearly with tip gap height. An proper approach to describe the tip clearance loss is the tip loss slope, i.e. the change in performance loss for a given change of Gap to Span Ratio (GSR) (Chima, 1998; Moon et al., 2002). Some researches have been developed emphasis on loss mechanisms of tip clearance. A series works on tip clearance loss have been done by Donghyun You and Meng Wang using large-eddy simulation (You et al., 2002; 2003; 2004; 2006).

Different with turbine and compressor, ducted propeller works in incompressible flow. Consequently, the heat transfer is neglected in numerical prediction. Ducted propeller works at a low advance ratio or high advance ratio occasionally. At high-advance ratios, ducted propeller generates complicated vortical structure which have effect on the blade tip loading. In turn, the blade tip pressure distribution could reverse at off-design points, which can change the vortical structure at the blade tip.

This paper aims to investigate the influence of tip clearance on the performance of ducted propeller. Considering the working condition of ducted propeller, the tip clearance effects are presented and compared at three different chosen advance ratios. These simulations are carried by solving the incompressible Navier-Stokes equations with a Multiple Reference Frame (MRF) method.

DUCTED PROPELLER MODEL

In this paper, calculations were carried out with a worldwide employed ducted propeller (the Ka4-70 propeller in 19A duct), which was presented by MARIN (Maritime Research Institute Netherland) (Oosterveld, 1970). The ducted propeller had a diameter of $D = 240$ mm and a pitch-diameter ratio of $P/D = 1$. The 19A duct was modified with a round trailing edge. The tip clearance of the propeller was set to 1 mm (about 0.42 percent of the propeller diameter $D$), which was the uniform clearance in the experiment research. Several difference models with tip clearance from 0 to 3 mm were used to investigate the effect on the performance of the ducted propeller.

NUMERICAL METHOD

The numerical simulations presented in this paper were carried out with the Ansys CFX14.5 (Ansys, 2012). The RANS solver with an Multiple Reference Frame (MRF) approach was used to predict the performance of the ducted propeller. The hydrodynamic equations were solved with the node-centered finite volume method. Considering that the propeller working in uniform flow, numerical predictions were carried out with single passage with rotational periodicity interface. Cavitation was not considered in all the simulations. All simulations were carried out in the water at 25°C with a density of 997.0 kg/m³ and a dynamic viscosity coefficient of $0.893 \times 10^{-3}$ kg/(m·s). The advance speed is set to $V_A = 4$ m/s. The Reynolds number was computed based on the chord length at 0.75 span and the water speed with a result of $5.3 \times 10^5$.

Domains and meshes

The overall computational domain involves two parts as show in Fig. 1, the stationary domain (a cylinder) and the rotating domain which are part of a cylinder. The stationary domain should be large enough to avoid that the far field boundary conditions affect the prediction of the flow near the propeller. The rotating domain is a single passage containing one blade. The dimensions of both domains are given in Table 1.

Once the computational domains have been defined, boundary-fitted grid is required for the flow solver. As Morgut and Nobile (2012) performed, both the hex-structured meshes and the hybrid-unstructured meshes could guarantee with similar levels of accuracy. Nevertheless, the hex-structured meshes showed better performances for a detailed investigation of the local flow field. In this paper, the stationary domain was discretized with hex-structured meshes. For less grid generation efforts, hybrid-unstructured meshes had been employed on the rotating domain. Hex-structured meshes were used in predicting the details flow near the propeller and the duct. To present the detail of the flow near wall, boundary layers elements were employed.
on the blade and the duct. The first element height of boundary layers was 0.05 mm, with a maximum Y-plus of 5.58, which is appropriate for the wall function for the flow inside the duct. Moreover, grids near the blade tip and in the clearance were refined with a finer mesh region, as showed in Fig. 2.

Table 1 Dimensions of the domains.

<table>
<thead>
<tr>
<th>Locations</th>
<th>Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1</td>
<td>4D</td>
</tr>
<tr>
<td>S2</td>
<td>5D</td>
</tr>
<tr>
<td>R</td>
<td>3D</td>
</tr>
<tr>
<td>L</td>
<td>0.4D</td>
</tr>
</tbody>
</table>

The boundary conditions were defined as follows. On the inlet boundary a velocity inlet with turbulence intensity of 1% were set. On the outlet boundary a static pressure outlet with a reference pressure of 0 Pa was defined. On the periodic boundary the rotational periodicity was applied. On all solid surfaces the no slip boundary condition was defined. The free slip wall was applied on the cylinder face. Two sliding interfaces (the front face and the back face on the streamwise of the rotating domain) were defined between the stationary and the rotating domain, through which the flow properties are transmitted.

Verification and validation

For evaluating the effect of the clearance, the non-dimensional thrust and torque coefficients are given: the duct thrust coefficient $K_{TN}$, and the torque coefficient $K_{Q}$:

$$K_{TN} = \frac{T_N}{\rho n^2 D^3}, \quad K_T = \frac{T}{\rho n^2 D^3}, \quad K_Q = \frac{Q}{\rho n^2 D^3}$$

(1)

The ducted propeller efficiency is defined as:

$$\eta = \frac{J}{2\pi} \frac{K_T}{K_Q}$$

(2)
\[ T_c = \text{the duct thrust}, \quad J = \frac{V_A}{nD} = \text{the advance ratio and} \quad \rho, n \quad \text{and} \quad D \quad \text{represent the density of water, the Rotation Speed (RPS) and the propeller diameter respectively. The total thrust of the ducted propeller is} \quad T = T_c + T_b, \quad \text{where} \quad T_b \quad \text{is the blade thrust. The clearance size is represented as Gap to Span Ratio (GSR) which is defined as:}

\[ \text{GSR} = \frac{\text{Clearance size}}{D} \times 100 \quad (3) \]

The verification investigation of mesh dependency was carried out by evaluating the thrust coefficients with different mesh densities. Three levels of meshes having the same node distribution with difference node numbers were chosen for the simulations. As Table 2 shows, it indicates that the medium and the fine mesh have the same results substantially compared with the coarse mesh. Considering the computational efficiency and the accuracy, the medium level mesh was chosen for the simulations finally.

**Table 2 Results of mesh dependency (J = 0.6, GSR = 0.5, VA = 4 m/s).**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Node number</th>
<th>( K_T )</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Stationary domain</td>
<td>Rotating domain</td>
</tr>
<tr>
<td>Coarse</td>
<td>1465120</td>
<td>546536</td>
</tr>
<tr>
<td>Medium</td>
<td>1465120</td>
<td>1491752</td>
</tr>
<tr>
<td>Fine</td>
<td>1465120</td>
<td>1896048</td>
</tr>
</tbody>
</table>

As Peng et al. (2013) suggested, eddy viscosity turbulence models are slightly better than Reynolds stress models in the prediction of tip vortex flow and the thrust and torque coefficients. Therefore, Standard \( k - \varepsilon \), Realizable \( k - \varepsilon \) and two-equation Shear-Stress Transport (SST) \( k - \omega \) turbulence models were evaluated to verify the effect of turbulence model on the simulations. The relative percentage errors \( \Delta K_T \) and \( \Delta K_q \) were defined as Morgan and Nobile (2012) presented. As can be seen in Table 3, the results predicted by three turbulence models are at a similar level of accuracy. Two-equation SST \( k - \omega \) turbulence was chosen for the simulations for the minimum \( \Delta K_T \).

\[ \Delta K_T(\%) = \frac{K_T^{\text{CFD}} - K_T^{\text{EXP}}}{K_T^{\text{EXP}}} \times 100 \quad \Delta K_q(\%) = \frac{K_q^{\text{CFD}} - K_q^{\text{EXP}}}{K_q^{\text{EXP}}} \times 100 \quad (4) \]

**Table 3 Influence of turbulence models (J = 0.6, GSR = 0.5, VA = 4 m/s).**

<table>
<thead>
<tr>
<th>Model</th>
<th>( \Delta K_T )</th>
<th>( \Delta K_q )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard ( k - \varepsilon )</td>
<td>-3.4</td>
<td>-2.2</td>
</tr>
<tr>
<td>Realizable ( k - \varepsilon )</td>
<td>2.6</td>
<td>-3.5</td>
</tr>
<tr>
<td>Two-equation SST ( k - \omega )</td>
<td>2.1</td>
<td>2.0</td>
</tr>
</tbody>
</table>

The results of simulations which are represented by the thrust coefficient, the torque coefficient and the propeller efficiency are compared with the experiment data. The maximum relative errors of thrust coefficient and the torque coefficient are 4.45% and 3.17% appear at \( J = 0.7 \) and \( J = 0.4 \) respectively. As presented in Fig. 3, the computed results of propeller open-water characteristics have a good agreement with the experimental data (Oosterveld, 1970).
In Fig. 3, $K_T$, $K_Q$, $K_{TN}$ and $\eta$ are the thrust coefficient, the torque coefficient, the duct thrust coefficient and the efficiency.

RESULTS AND DISCUSSION

To investigate the effect of tip clearance on ducted propeller performance, several models (Ka4-70 in 19A duct) with different tip clearance had been selected. In this paper, tip clearance was changed by expanding the diameter of the duct for maintaining the propeller blades. As the flow field around the propeller shows different character, three levels of advance ratio were chosen to show the tip clearance effect. This ducted propeller reaches the maximum efficiency at about $J = 0.6$. Therefore, we choose $J = 0.3$ as the low advance ratio, $J = 0.6$ as the middle advance ratio and $J = 0.8$ as the high advance ratio. Some of the computed results will be presented and discussed follow.

Effects on performance of ducted propeller

The flow leaking through the tip clearance causes the change of efficiency. The change of the efficiency with tip clearance size was observed roughly linearly at low advance ration. The level of tip clearance effect is evaluated by the leakage loss slope. Fig. 4 shows the efficiency varies with tip clearance size at different advance ratio. In general, efficiency decreases while the clearance expands. The changing trends is approximately linearly at $J = 0.3$ and $J = 0.6$. Fig. 4(c) shows that efficiency decreases firstly then increases at $J = 0.8$. This implies that the leakage loss is not always roughly linearly at all working condition. It is influenced by the advance ratio. In other words, the leakage loss is related to the ratio of two components (rational velocity and advancing velocity) of the absolute velocity near the clearance.
For detailed investigation, thrust coefficients and torque coefficients at $J = 0.8$ are given in Fig. 5. We found that thrust and torque coefficients decrease as the gap expands from 0.1 to 0.5 GSR and then increase as the gap expands. The duct thrust curve in Fig. 5(c) can be treated as two lines with slopes as $-6.228 \times 10^{-3}$ and $-5.965 \times 10^{-4}$ respectively. All the three coefficient curves reach the turning point at a GSR near 0.5.

**Effects on duct and blade loading**

Fig. 6 shows the pressure coefficient distribution on the inner duct of ducted propellers with different GSR. The pressure coefficient is defined as:

$$C_p = \frac{p - p_\infty}{0.5 \rho U_\infty^2}$$

The distribution of pressure coefficient is strongly influenced by the gap size. Low $C_p$ area expands when GSR decreases. Area 2 is mainly influenced by vortex between two blades at the suction side. Tip vortex causes the decrease of pressure at area 1. Area 3 is near the leading edge tip of the blade. Because the local attack angle at the blade tip is negative at this working condition, pressure at area 3 reverses. Tip vortex breaks down after area 4. It is obvious that pressure on the inner duct is influenced less by the expanding of the gap. Combining with Fig. 5(b) we can conclude that slope of the fore curve is mainly caused by the tip vortex effect on area 1 and 4, the slope after is mainly influenced by flow separation at area 3 and vortex between blades at area 2.
Fig. 7 shows the pressure coefficient distribution on blade suction side. The distribution of the pressure is almost the same except for the little difference at the blade tip near the leading edge. It is obvious that the influence of gap size mainly concentrate on the blade tip.

Fig. 8 shows the blade tip loading at a constant span of 0.98 of different gap. Blade tip loading of GSR below 0.5 is illustrated in Fig. 8(a). In Fig. 8(a), it is obvious that blade tip loading is strongly influenced by the gap size. As the GSR increases from 0.1 to 0.5, the reverse point transfers from 0.19 to 0.27 streamwise. Because of the existing of reversed pressure area at leading edge, the tip vortex origination is pushing back to the trailing edge. When GSR is below 0.5, blade tip loading near the leading edge decreases while loading near the trailing edge increases with the expanding of the gap size. Blade tip loading of GSR above 0.5 is illustrated in Fig. 8(b). It demonstrates that blade tip loading has no relation with the gap size. The reversed points locate at about 0.1 streamwise. Pressure coefficient reaches the maximum at about 0.9 streamwise.

Tip vortex structure

Quantitative studies of flow in turbine, compressor and pump have been done to understand the mechanisms of tip leakage loss. You et al. (2007) found that the tip vortex structure is formed by three parts: the tip-separation vortex, the tip-leakage vortex and the induced vortex. The tip-leakage vortex is caused by the pressure different between the pressure and the suction side. The tip-separation vortex is formed due to flow separation underneath the blade tip. The induced vortex is generated by the tip-leakage vortex. Fig. 9 shows the vortex core in steady state using the $\lambda_2$ vortex-identification (Jeong and Hussain, 1995). Compared with Fig. 9(a), tip vortex in Fig. 8(b) has a different structure. Tip leak-age vortex is separated into two parts by the reversed pressures point as show in Fig. 8. In the case $J = 0.8$, tip-leakage vortex near the leading edge originates from the suction side at about 3% to 10% of the chord, and extends toward the pressure side.
The tip vortex structure is affected not only by the blade loading but also the gap size. Fig. 10 shows vortex line on Y-Z plane (X = 0) in the tip clearance of different GSR. In the case GSR=0.1, as shown in Fig. 10(a), a main large vortex is formed at the beginning of the clearance near the inner duct. Several shedding vortex is found after the main vortex. These vortices are formed because of the injection of the fluid through the clearance into the boundary layer on the inner duct. Fig. 10(c) shows planar vortex line of GSR = 0.5 and 1 respectively. In the case GSR=1, the main vortex is near the blade tip face. This vortex is formed mainly due to the flow separation at the tip. The interaction of the main flow through the clearance with the boundary layer of the blade also contributes to the vortex near the blade tip face.

As the advance ratio increases to 0.8, the attack angle at the blade tip reduces to negative. Inversed pressure appears near the leading edge at the blade tip. A different vortex structure was generated by this inversed pressure area shown in Fig. 9(b). This vortex structure can be separated to two parts by the reversed point. As the pressure downstream rose, the vortex breakdown causing the loss of energy. The first energy loss was before the reversed point as shown in Fig. 9(b). The second energy loss begins from the reversed point to downstream. As shown in Fig. 10(a), the clearance was occupied by the tip separation vortex when the clearance was small enough. The shedding vortex breakdown appeared in the clearance at high advance ratio at small GSR. When the GSR was up to 1, the vortex breakdown was not that much in the clearance as the GSR 0.1. This can be the explanation of the broken line in Figs. 4 and 5.

**Effects on turbulence kinetic energy**

Fig. 11 shows the Mass Circle Average (MCA) turbulence kinetic energy on the inlet to outlet line which is defined as

$$
\bar{k} = \frac{1}{m} \int k_p \vec{V} \cdot \vec{n} dA
$$

(6)
where

\[ \dot{m} = \int_A \rho \hat{V} \cdot \hat{n} \, dA \]  

(7)

The blade boundary layer separation and the clearance leakage flow generate strong turbulence kinetic energy. The clearance shows no effect on the turbulence kinetic energy from the inlet to about 50% streamwise. The turbulence kinetic energy increases with the expanding of the clearance after 50% streamwise. It is obvious that the flow through the larger clearance generates more turbulence kinetic energy. The influence of the clearance on turbulence kinetic energy mainly locates after 50% streamwise.

**Fig. 11 Mass circle average (MCA) turbulence kinetic energy from the inlet to outlet.**

**CONCLUSIONS**

A ducted propeller with different size of tip clearance has been presented to investigate the influence of the tip clearance to ducted propeller. This analysis was carried out with RANS method. Several validation studies have been done to detect the influence of the grid and the turbulence model on simulations. The influences of the clearance on ducted propeller are reflected in five aspects mainly.

1) The thrust and torque coefficient decrease linearly with the clearance expanding at low advance ratio. While at high advance ratio, these coefficients decrease with the clearance expanding to a GSR of 0.5 then increase with its expanding. At high advance ratio, the main cause of the nonlinear of loss slope is the reversed pressure at leading edge.

2) The clearance size has little effect on the pressure distribution on the blade below 0.98 spanwise.

3) Pressure distribution on the inner duct and blade tip was influence strongly by the tip clearance below a GSR of 0.5. The mainly influence area locates at the suction side and near the leading edge.

4) The vortex structure in the clearance changes from a shedding vortex to one big separation vortex with the expanding of the clearance. This vortex structure and the blade tip loading affect each other before the trailing edge.

5) The bigger clearance generates more turbulence kinetic energy. The influence on the turbulence kinetic energy mainly located after 50% streamwise.

The negative attack angle at high advance ratio caused the inversed pressure area near the leading edge. The vortex structure was influenced strongly by the inversed pressure area. The shedding vortex breakdown loss more energy can be considered as the explanation of two section slope at high advance ratio.

**ACKNOWLEDGEMENT**

This research was supported by the Fundamental Research Funds for the Central Universities (Grant NO.3102014JCQ 01007).
REFERENCES


