NUMERICAL AND EXPERIMENTAL INVESTIGATIONS ON UNSTEADY CAVITATION OF A HIGH-SKEW CP PROPELLER

Jianqiang Wang¹,², Xuemei Feng¹,²
¹Marine Design & Research Institute of China,
²Laboratory of Science and Technology on Waterjet propulsion.
South Xizang Road 1688, Shanghai, China
e-mail: wjq369889@163.com

ABSTRACT: This paper presents results of the numerical and experimental investigations of the cavitation characteristics of a high-shear CP propeller in the usual design and off-design conditions. The computations are carried out with RANS and LES solutions using a cavitation model based on a two-phase mixture formulation. The computations of unsteady non-uniform inflow conditions at two different pitch settings are carried out. Compared with experimental observations, the computational results of RANS solutions closely reproduce the main features of cavity patterns under the design pitch condition. The cavity pattern obtained by RANS model is a little smaller than the experimental result. LES model gives a better cavity pattern at the tip area. Under the off-design condition the face tip-vortex cavitation are observed in the experiment. The results obtained by RANS and LES model show an acceptable cavity pattern in comparison with the experiment.

Key words: Cavitation, High-shear CP propeller, Numerical simulation, LES

1. INTRODUCTION

In this paper, the numerical and experimental investigations of the cavitation characteristics of a high-shear CP propeller in the usual design and off-design conditions are presented. Controllable pitch propellers (CPP) can meet the requirement of multiple conditions of the surface ship and improve the propulsion performance and ship maneuverability under different working conditions. Except for the hydrodynamic performance, the cavitating behaviour and resulting radiated noise are the major concern of the propeller design, especially CPP design. Today the standard design tool typically including potential flow solvers on the basis of lifting-line, lifting-surface theory and panel method, are able to predict the proper pressure distribution on the blade, but with clear theoretical limitation regarding cavitation dynamics. The RANS (Reynolds-Averaged Navier-Stokes) method are applied more and more widely. However, some general features such as cavity extent or shedding behaviour are still questionable due to the statistical character of RANS in its way of modelling turbulence. It has been motivating the study of LES, Large Eddy Simulation, which has the ability to treat highly unsteady problems [1].

In case of CP propellers, some additional open points still exist if “very off-design conditions” are considered; as, for example, the hydrodynamic characteristics at reduced pitches. In these conditions, the correct prediction of the flow field and pressure distribution can prove to be a significant challenge for potential flow based methods and may require the use of more sophisticated tools, like RANS and LES solvers [2].

The performance of different solvers in simulating cavitating flow has been summarized in a recent workshop on cavitation and propeller performance, supporting the above discussion on merits of potential flow solvers, RANS, and LES. Thus, RANS is expected to be able to predict reasonably accurately cavity extent, but regarding the cavitation dynamics, results
presented in [3] indicate that scale resolving methods, like DES or LES, are needed.

In this work the cavitating flow around a high-screw CP propeller at different pitch conditions is simulated with RANS and LES computational methods to demonstrate the capability of different simulation tools for this complex flow case. The experiment was carried out in the cavitation tunnel at SSSRI (Shanghai Ship and Shipping Research Institute). The numerical simulations are set up with as exact set-up of experiment as reasonably possible.

The structure of the paper is as follows: Propeller characteristics are reported in Section 2; while in Section 3 the computational configuration and the experiment settings are presented. A detailed analysis of simulated results of two pitch conditions are presented in Section 4. Conclusions are summarized in Section 5.

2. PROPELLER CHARACTERISTICS

The propeller utilised for present study is a high-screw CPP, which is operated at constant RPM in correspondence to very different ship speeds by means of blade pitch angle variation. The main characteristics are reported in Table 1, where \( D \) is the propeller model diameter, \( \text{EAR} \) is the propeller expanded area ratio, \( \text{D}_h \) is the hub diameter and \( Z \) is the blade number.

<table>
<thead>
<tr>
<th>Table 1 Propeller main characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>( Z )</td>
</tr>
<tr>
<td>( D )</td>
</tr>
<tr>
<td>( \text{EAR} )</td>
</tr>
<tr>
<td>( \text{P/D} )</td>
</tr>
<tr>
<td>( \text{D}_h/D )</td>
</tr>
</tbody>
</table>

Figure 1 Model of CP propeller

3. COMPUTATIONAL CONFIGURATION

The cavitation experiment of the high-screw marine CP propeller in the wake under two pitch conditions (A: design pitch \( \text{P/D}=1.15 \) and B: off-design pitch \( \text{P/D}=0.80 \)) was carried out in the cavitation tunnel. The test condition of two different pitch is \( \sigma_n=1.299 \) and 1.291, where \( \sigma_n \) is the cavitation number, defined as \( \sigma_n = (P_0 - P_v) / (0.5 \rho n^2 D^2) \), where \( P_0 \) is the pressure at the centre of the tunnel, \( P_v \) is the vapour pressure, as shown in Table 2.
Table 2 Cavitating condition of the cavitation tunnel test

<table>
<thead>
<tr>
<th>Condition</th>
<th>P/D</th>
<th>( n_\text{m}(\text{r/s}) )</th>
<th>( \sigma_n )</th>
<th>( K_T )</th>
<th>( P_0 ) (kPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1.15</td>
<td>22</td>
<td>1.299</td>
<td>0.2358</td>
<td>21.079</td>
</tr>
<tr>
<td>B</td>
<td>0.80</td>
<td>22</td>
<td>1.291</td>
<td>0.1326</td>
<td>21.093</td>
</tr>
</tbody>
</table>

The cavitating flow cases in different pitch conditions are implemented and simulated using commercial RANS and LES solvers (FLUENT). The computational domain is divided into two parts: a stationary and rotating zone. The cylindrical stationary zone is discretized with a structured mesh. The rotating zone is composed of tetrahedrals with prism layer of hexahedrals around the blade and hub. See Figure 2 and the sliding mesh method is applied. The grids try to mimic the experimental as much as possible by including the actual size of the cavitation tunnel and the real geometry of the propeller and hub. For the condition A, three meshes are used to investigate the cavitation by RANS and LES methods. The details of the meshes are shown in Table 3 and Figure 2.

Table 3 Three meshes for condition A

<table>
<thead>
<tr>
<th>Mesh Type</th>
<th>Mesh Size</th>
<th>( Y^+ )</th>
<th>Boundary layer</th>
</tr>
</thead>
<tbody>
<tr>
<td>GridA</td>
<td>8.0 million</td>
<td>30-70</td>
<td>without</td>
</tr>
<tr>
<td>GridB</td>
<td>9.6 million</td>
<td>20-60</td>
<td>without</td>
</tr>
<tr>
<td>GridC</td>
<td>14.3 million</td>
<td>5</td>
<td>with</td>
</tr>
</tbody>
</table>

Figure 2 Computational domain for condition A.

(a) stationary zone; (b) rotating zone; (c) Grid A; (d) Grid B; (e) Grid C.

For condition B, a similar mesh set is used to discretize the fluid domain. The total mesh size is 8.9 million cells and \( y^+ = 25-70 \) on the major part of the blade.

The numerical simulations of the CP propeller are configured to be as close to the
experimental set-up as possible. The investigation of the cavitation behavior is based on RANS and LES solvers by Zwart-Gerber-Belamri cavitation model, using a Volume-of-Fluid implementation to present two phases of liquid and vapor. The rotational speed is fixed with the experimental set-up, the inflow speed and pressure of the tunnel are adjusted to match the cavitation number $\sigma_n$ and the thrust coefficient $K_T$.

The inlet of the computational domain is set with the non-uniform wake. To reduce the grid dissipation, the wake acting on the velocity inlet should be as close to the propeller disk as possible. However, if the velocity inlet is too close to the propeller disk, it is hard to maintain the mesh quality and the propeller would generate induced velocity to the velocity inlet. In this paper, we choose 1.0D as the distance to the propeller disk. A research on the computational domain is carried out to make sure the velocity inlet is not influenced by the induced velocity of the propeller rotation. An extended computational domain, where the distance from the velocity inlet to the propeller disk is 1.5D is used to simulate the velocity distribution of the 1.0D plane in non-cavitating condition. The sliding mesh is applied and the other settings are the same with the cavitating condition. The computational domain increases the distance from the inlet to the propeller disk compared with the cavitating computational domain, see Figure 3.

![Computational domain](image)

**Figure 3 computational domain of non-cavitating condition**

The velocity distribution in the inlet (1.5D) and the 1.0D plane are shown in Fig 4. The velocity distribution of the two planes are very similar, thus the inlet (1.0D plane) of the cavitating computational domain is hardly influenced by the induced velocity of the propeller rotating. We can also see that there is a remarkable area of reduced axial speed between $340^\circ$-$0^\circ$-$50^\circ$.

![Velocity distribution](image)

**Figure 4 velocity distribution in the inlet (1.5D plane) and 1.0D plane**

left--inlet(1.5D plane); middle--1.0D plane; right--definition of wake angle(seen from downstream)

4. **NUMERICAL RESULTS**

The investigation of the cavitation behavior in the condition A and B is based on RANS and LES solvers by Zwart-Gerber-Belamri cavitation model. The transient RANS simulation
is conducted with k-omega SST model. The RANS computation is started with Frame Motion mode to achieve a stabilized pressure field and then is changed to Mesh motion mode to get the final results.

4.1 Condition A

The developments of the cavity during the high wake area of experimental and RANS results are illustrated in Figure 5. The cavity of the blade is influenced largely by the wake distribution. In the simulation visualization, the cavity interface is indicated by the iso-surface of the vapor volume fraction \( \alpha_v = 0.5 \). From the first and second columns, we can find tip vortex, back sheet cavitation and hub cavitation for the experimental results of condition A. At 340° position a small sheet cavity is observed between \( r/R = 0.6 \) and 0.7 in the leading edge of the blade back. As the propeller is rotating, the sheet cavity develop into a bigger one and a slender tip vortex cavity appears in the leading edge. At 20° position, the former back sheet cavity disappears gradually and the cavity in the leading edge originates from \( r/R = 0.8 \); the tip vortex cavity keeps developing downstream. Strangely a slender sheet cavity is observed between \( r/R = 0.7 \) and 0.9. At 30°-40° position, the starting point of the cavity in the leading edge shifts slowly downstream to 0.9R. The sheet cavity in the middle of the blade back vanishes little by little. At 50° position, the originating point changes to 0.95R and the area of the tip vortex reduces and sheds off the tip. At all positions there exists stable root cavity in the blade back.

The third, fourth and fifth columns are the RANS results of Grid A, B and C respectively. The location of the sheet cavity coincides well with the experimental frame. At 340°-0°, the sheet cavity does not appear absolutely in the leading edge for Grid A and B RANS results. Grid C results give a better prediction of the cavity than others. However, grid C results over-predict the sheet cavity and gives an exaggerated area of sheet cavity in low-radius region except 40° and 50° position. Three grid results are not good at capturing the tip vortex cavity. I assume the main reason is that the mesh dimension of the blade tip is not fine enough to define the tip vortex cavity directly.
Figure 5 development of the cavity during one blade passage of RANS results

first column---hand drawing cavitation pattern; second column---experimental cavitation pattern; third column---simulation results of Grid A; fourth column---simulation results of Grid B; fifth column---simulation results of Grid C; vapor volume fraction $\alpha_v=0.5$ (green).

Hub vortex cavity is captured by grid B and C RANS results. Obviously Grid C has a better pattern compared with grid B, see Figure 6. After shedding from the hub, the hub vortex cavity collapses very quickly. The grid in stationary domain is thin and has a big mesh dimension. Once hub vortex cavity enter the stationary domain, it vanishes rapidly.

Figure 6 Hub vortex cavity of experimental and RANS results.

(a). Experimental result; (b) Grid B result (c) Grid C results; $\alpha_v=0.5$ (green)
Grid B is chosen to compare RANS with LES solvers. Grid C is more suitable for this, but LES results cost too much CPU time. This work is under way but not finalized. The cavity patterns of RANS and LES results are shown in Figure 7. Apparently, LES results are better than RANS ones in the aspect of the cavity positions and areas. But LES results are still not good enough to capture tip vortex cavity compared with the experimental patterns. It seems that the grid is the major factor of tip vortex cavity.
Figure 7 development of the cavity during one blade passage of RANS and LES results for Grid B

first column—hand drawing cavitation pattern; second column—simulation results of RANS;
third column—simulation results of LES; vapor volume fraction $\alpha_v=0.5$ (green).

The sheet cavity are related to the pressure distribution and the vortical structures, the former plays more role. However for tip vortex cavity, the vortical structures are crucial. 40° position is chosen to investigate the second invariant of the vorticity $\lambda_2 = \|\nabla v\| - \|\nabla \times v\|$[4], pressure and cavity. There are evident vortical structures (black circled area) in blade 1 and 2, that lead to the tip vortex cavity, see Figure 8 (a). Vortical structures can also be seen in the root of blade 1. The low pressure in Figure 8 (b) results in sheet cavity.

Figure 8 LES predicted iso-surface of the second invariant of the vorticity and pressure.

(a) the second invariant of the vorticity $\lambda_2 = \|\nabla v\| - \|\nabla \times v\|$ at selected value (red) and $\alpha_v=0.5$ (green);
(b) pressure distribution.

4.2 Condition B

Based on the experience of condition A, similar settings are adopted for condition B. RANS and LES solvers are used to simulate the cavity flow. The experimental results at 0°, 20° are shown in Figure 9.
In the experiment of condition B, there are face sheet cavity and back root cavity. RANS and LES results can be seen in Figure 10. The RANS, LES cavity patterns are almost the same and the face cavity is very small, so is the back root cavity. There is no cavity in blade 1 of RANS and LES results, which is different from the experimental results. The face sheet cavity can be seen around 0.65R of the leading edge on blade 2,3,4,5.

Analyze the second invariant of the vorticity $\lambda_2 = \|\nabla \psi\| - \|\nabla \times \mathbf{v}\|$, pressure and cavity of RANS and LES solvers, see Figure 11. Vortical structures (black circled area) and low pressure lead to the face cavity. LES patterns are obviously better than RANS patterns. Blade
1 is located in the wake peak. There are remarkable low pressure area, so there is no face cavity.

Figure 11 LES predicted iso-surface of the second invariant of the vorticity, cavity and pressure.

the second invariant of the vorticity \( \lambda_2 = \| \nabla v \| - \| \nabla \times v \| \) at selected value (yellow) and \( \alpha = 0.5 \) (green)

(a) RANS; (b) LES.

5. CONCLUSIONS

Numerical simulations of cavitation flow in the wake under two pitch conditions (A: P/D=1.15 and B: P/D=0.80) are investigated. For condition A, The three kinds of meshes and two cavitation models are used in the process of the simulation. The computations were carried out with RANS and LES equations using a cavitation model based on Zwart-Gerber-Belamri model. The cavitation pattern obtained matches roughly with the experimental results, except for the tip vortex cavity, hub vortex cavity and the shape of the sheet cavity in suction surface. The better cavity pattern can be obtain by the refined mesh. Comparing the results of RANS with the ones of LES, the results of LES can not only present the better pattern of the cavitation, but also show more flow details. The patterns of the tip vortex cavity in the above numerical simulation are all not very consistent with the ones of experimental results, which indicate that the off-blade meshes are not refined enough to obtain the off-blade cavity.

Based on the experience of condition A, the regimes of RANS and LES of Zwart-Gerber-Belamri cavitation model are applied to investigate the cavitation results of condition B. In the wake peak, the sheet cavity on pressure surface can be obtained very well. However, in the high wake area, no sheet cavity on pressure surface can be seen. The pattern of root cavity is identified with the experimental results.

REFERENCES