Influence of Hydrodynamic Interaction between Ducted Propellers and Struts on Performance of Azimuth Thrusters

Isao Funeno

Kawasaki Heavy Industries, Kobe, Japan.

ABSTRACT
Since strong hydrodynamic interaction between ducted propellers and struts of azimuth thrusters is very complicate, it is not so easy to design optimally the ducted azimuth thrusters in spite of the importance. Both the shape and the location of struts affect performance of the ducted propellers such as inflow into propellers, increase of strut drag due to suction of propellers, steering moment of the units and others, which bear a strong relationship of performance tradeoff. In this paper, first in order to improve bollard thrust of the ducted azimuth thrusters, the interaction was discussed by using the Lagally’s theorem theoretically and the CFD (Computational Fluid Dynamics) technique numerically. Particularly effects of the strut cross-sections, relative position between propellers and struts were discussed. Next, in free running condition the strut shape should be considered to avoid large-scale separation flow, which causes harmful propeller cavitation. The issues were studied for changing the both shapes of propellers and struts using the CFD computations and the cavitation observations. Finally, the concerns about increase of the unit steering moments and the unit installation onto vessels due to larger distance between ducted propellers and struts were discussed in order to design the optimal units.

Keywords
Ducted azimuth thruster, Propeller, CFD, Cavitation, Steering moment.

1 INTRODUCTION
Ducted azimuth thrusters are very useful propulsion units on providing those advanced maneuverability and higher bollard thrust to such support vessels – tugboats, platform supply vessels, cable layers, drill ships and others. However, it is very difficult to design optimally ducted azimuth thrusters from both hydrodynamic performance and mechanical viewpoints. As one of the reasons, the ducted azimuth thrusters are very complicated and consisted of many components such as propeller, duct, gear housing, strut and others as shown in Figure 1. In the former SMP the author presented the difficulty to obtain optimal design of the ducted azimuth thrusters, in which discussed mainly hydrodynamic interaction between the duct and the gear housing (Funeno, 2009).

On the other hand, it is also very important to consider hydrodynamic interaction between the ducted propeller and the strut of the azimuth thruster. Then, we have to study hydrodynamically optimal shape of the struts within mechanical constraints. Of course the thrusters should provide excellent propulsive performance in bollard conditions as well as free-running conditions. However, there are little references about the hydrodynamic interaction.

In this paper the author presents basic concept and investigation using CFD in detail in order to obtain larger bollard thrust and higher propulsive efficiency of the ducted azimuth thrusters. Further cavitation erosion risk in free-running conditions due to low water speed region, wake behind the strut and the countermeasures are discussed. Additionally, optimal location between the ducted propeller and the strut to avoid excessive steering moment of the ducted azimuth thrusters and those installations onto vessels are discussed and finally conclusions are described.
2 INFLUENCE OF STRUT IN BOLLARD PULL CONDITIONS

2.1 Basic concept in bollard pull condition

Considering hydrodynamic interaction between a strut incorporating L-drive or Z-drive of power transmission system of an azimuth thruster and a rotating propeller behind the strut, the interaction can be almost regarded as an analogy with that between a ship hull and a rotating propeller behind the stern, in brief an issue of thrust deduction fraction in propulsion tests. The basic mechanism concept of thrust deduction can be explained simply using Lagally’s theorem (Milne-Thomson, 2011). As with the concept, on the assumption of bollard pull condition of the azimuth thruster, consider simply the force exerted on a circular cylinder by a sink. Taking Figure 2 with the sink of the strength of ‘m’ at A and the circular cylinder with the radius of ‘a’ at O on the x-axis, then the complex potential \( w(z) \) is described in the following Equation (1) with the complex variable \( z \).

\[
w(z) = m\log(z) - m\log(z - f) - m\log(z - f')
\]

Where \( f = a' / f = OB \) and \( f = OA \). Therefore the cylinder is attracted towards the sink with the following horizontal force \( F_x \),

\[
F_x = \frac{2\pi \rho m^2 a^2}{f(1 - a^2)} \quad (2)
\]

In that case, conceptually \( m, a \) and \( f \) can correspond with the propeller thrust loading, the curvature radius of a strut rear end and the distance between the propeller and the strut respectively. Next consider how these factors contribute to \( F_x \). Figure 3 shows the change of nondimensional force \( F_x / F_{xo} \) with regards to these nondimensional factors for easy to understand the effect of the factors on the force. Where, \( F_{xo} \) is \( F_x \) at \( m = a = 1 \) and \( f = 2 \), both \( a / a_o \) and \( m / m_o \) are ratio to each unit quantity and \( F_{xo} \) is ratio to twofold quantity. From Figure 3, it is inferred by analogy that the force exerted on the strut by the rotating propeller, the induced drag on the strut can be increased steeply by the increased propeller thrust loading, the increased curvature radius of the strut rear end and the decreased distance between the propeller and the strut.

In more details, consider the pressure distribution on the surface of the circular cylinder exerted by the sink. We can obtain simply the pressure distribution using the velocity potential derived from the complex potential, Equation (1) and Bernoulli’s theorem. The pressure is nondimensionalized by the following equation,

\[
C_p = \frac{\Delta p}{\frac{1}{2} \rho (m^2)}
\]

Where \( \Delta p \): differential pressure. Further the drag component of pressure, \( C_{px} \) is defined as,

\[
C_{px} = C_p \cos \alpha
\]

Figure 4 and 5 show distribution of the pressure and the drag component with the circumferential position, \( \alpha \) on the circular cylinder by parametrically changing the distance \( f/a \) respectively. From these figures, it is found that the peaks of negative pressure and drag component are shifting from about 40 degree in circumferential toward 0 degree, the rear end of circular cylinder with decreasing the distance \( f/a \). Hence it is very important to consider the optimal cross-sectional shape in the vicinity of the rear end of strut intensively so that its curvature radius should be as small as possible to take account of both the diameter and the horizontal location of the vertical drive shaft inside the strut.

In addition, approaching the propeller toward the strut rear end makes more efficiently the propeller thrust increased due to low induced water speed behind the strut, the wake gain. However, the strut drag could be increased considerably due to the increased propeller thrust loading.
as mentioned in Equation (2) and then the unit thrust could be decreased totally.

2.2 Discussion with CFD Results

More specifically, the above mentioned discussion should be verified from flow simulation using the advanced viscous CFD (Computational Fluid Dynamics) technique based on the unstructured grid method so as to deal with complicate geometries such as the ducted azimuth thrusters. The computational method was almost the same with that described in (Funeno, 2009). But the versatile commercial CFD software, STAR-CCM+ by CD-adapco was used for these computations. As it was before, all the flow simulations in bollard pull conditions were computed in the advance coefficient $f/a=0.05$ approximately.

The flow simulations were carried out for the two representative cross-section of strut with both the identical propeller and the ducts almost similar to MARIN No.19A duct. Further for each case the distances between the ducted propellers and the strut rear ends were changed from the original ones. Figure 6 shows the CFD results of pressure distribution around the thrusters on the horizontal cutting plane at $0.7R$ above the propeller shaft centers. In these cases the propeller wash direction is from left to right. The pressure is nondimensionalized by the following pressure coefficient, $C_{pn}$,

$$C_{pn} = \frac{(p-p_o)}{\frac{1}{2} \rho \omega^2 D^2}$$  \hspace{1cm} (5)

Where, $p$: static pressure, $p_o$: standard pressure, $\rho$: water density, $n$: propeller revolution, $D$: propeller diameter, $R$: propeller radius. The both strut sections are called “Type A” and “Type B”, which have relatively larger and smaller curvature radius of the strut rear ends respectively.

Figure 6-(a) shows the pressure distribution in case of the section “Type A” close to the ducted propeller. The intensive negative pressures, which cause the induced drag of struts, occur in both side regions at about one-quarter strut length far from the strut rear end. Figure 6-(b) shows that in case of the same section far from the ducted propeller. The negative pressures around the strut rear end are considerably weakened by distancing the ducted propeller from the strut rear end. Next Figure 6-(c) shows that in case of the section “Type B” close to the ducted propeller. The negative pressure region on the both side around the strut rear end are relatively weakened to those of Type A due to the smaller curvature radius of the strut rear end than that of Type A. Finally Figure 6-(d) shows that in case of the same section far from the ducted propeller. The negative pressure regions are most weakened in all the cases.

With the theoretical discussions and the above discussed CFD results, the influence of the strut on performance of ducted azimuth thrusters in bollard pull conditions are

![Figure 5 Distribution of drag component of pressure with $f/a$.](image)

![Figure 6 Pressure distributions on horizontal section at 0.7R above.](image)
revealed well. Therefore, in order to make the strut drag decreased, it is essential to provide the smaller curvature radius around the strut rear end as well as the longer distance between the strut rear end and the ducted propeller as possible.

3 CAVITATION EROSION RISK IN FREE-RUNNING CONDITIONS

In free-running conditions, strut geometries should be designed so as to produce low strut drag and to avoid large-scale separated flow behind the strut, because the wake behind the strut flowing into the ducted propeller has a strong association with propulsive performance, propeller cavitation, vibration and noise. Especially in this chapter, influence of the strut geometries on cavitation erosion risk over propeller blades in free-running conditions is discussed using the CFD technique considering cavitation effect and cavitation observation tests at a depressurized towing tank.

3.1 Unit Models to Investigate

In order to investigate the influence of wake behind the strut on propeller cavitation, the following ducted azimuth thruster models were computed. Figure 7 shows the 3D-outlines of three representative thruster units, Figure 7-(a) for the “Unit A” having the strut with tentatively blunt flange connected to the gear-housing below, the level supports and the low-skewed propeller, Figure 7-(b) for the “Unit B” having the same geometries with Unit A except the high-skewed propeller and Figure 7-(c) for the “Unit C” having no flange, but smaller curvature radius of the strut rear end section than that of Unit A & B, the proper strut root fillet radius, the angled supports and the same high-skewed propeller that its propeller center is shifted backward somewhat apart from the original position. Figure 8-(a) and (b) show propeller projections of “Propeller A”, the low-skewed propeller and “Propeller B”, the high-skewed propeller respectively.

3.2 Computational Method

In order to conduct flow simulation around the ducted azimuth thrusters in cavitating conditions, the cavitation flow model was introduced in concert with the momentum and mass transfer equations. In the CFD flow simulation, so-called multi-phase flow model was utilized, which was based on the modeling fluid density by phase change of water. But the actual phase change of water is very complicated phenomena that the cavitation bubbles combine into one sheet cavitation, or conversely the sheet cavitations collapse to cloud cavitations or bubble cavitations due to instability of flow. In this paper, in order to comprehend the phenomena from a macro-viewpoint, as a simplified model, the two-phase flow model was utilized with the Volume of Fluid, the VOFS method in conjunction with the cavitation model suggested by Schnerr and Sauer (2001). Further all the unsteady computations in model scale were conducted using the sliding mesh technique in open water condition with the k-omega/SST turbulence model and the wall function up to proper convergences, stable variation of representative characteristics. The time increment, \(\Delta t\) was comparable to 1.0 degree per time step in propeller rotation direction for all the computation cases. The total number of cells was about 12 million for each unit model.

As the representative free running condition of work vessels, the operating point was set to the advance coefficient, \(J = 0.61\) and the cavitation number based on propeller shaft center, \(\sigma_c = 1.91\) for all the computations. According to those non-dimensional factors of the operating condition, the inflow speed at inlet boundary, the propeller revolution and the static pressure were adjusted properly.

3.3 Cavitation Observations

In order to comprehend cavitation phenomena on blades of the ducted propellers behind the strut and to obtain validation data for the CFD computations, the cavitation observation tests were conducted at the Depressurized Wave Basin of MARIN in the Netherlands. Figure 9 shows, for example the model for Unit A, which the thruster housing and duct model were made of transparent PMMA (Perspex) to make the cavitation observations easy and eliminating undesired shadows. The cavitation observation tests were carried out in open water conditions to various combinations of both \(J\) and \(\sigma_c\). During the cavitation tests, the suction side cavitations were observed by means of two high-speed video cameras from port and starboard side.
3.4 Results and Discussions

Some results of sheet cavitation distribution on suction side for Unit A, Unit B and Unit C are shown in Figure 10 changing circumferential position of the key blade, $\theta$ from 0 (top) to 4, 5, 6 and 7 degree. The sheet cavitations are represented with red iso-surfaces of 0.5 in two-phase flow void fraction, $\alpha$. From Figure 10-(a) for Unit A, it can be seen that the sheet cavitation developed at $\theta=0$ degree is isolating gradually toward both blade-tip and blade-root with advancing the key blade. It is quite possible that the isolated sheet cavitation metamorphoses into cloud cavitations or bubble cavitations which could cause cavitation erosion risk on the blade surfaces around the root to mid-span (van Terwisga, et al, 2009).

Next, in the case of conversion of the propeller only from the low-skewed Propeller A to the high-skewed Propeller B not changing the thruster housing, from Figure 10-(b) for Unit B, it can be seen that diminishing pattern of the sheet cavitation is almost the same with that of Unit A, but the isolated sheet cavitation reduces partially due to the higher-skewed blades than those of Propeller A. However, it is still possible to remain the cavitation erosion risk, even though the service life against cavitation erosion damage can be extended.

On the other hand, almost the similar behaviors of every cavitation above-mentioned were observed through the high-speed camera video in the cavitation observation tests. For example, the photographs shown in Figure 11 are snapshots picked out from the video for Unit A and Unit B. From the both photographs, it can be seen that there are the isolated sheet cavitations a little bit far from the blade leading edge from 0.5R to 0.6R in radial direction. Further the isolated sheet cavitation of Unit B is...
somewhat smaller than that of Unit A. These isolated cavitations could be varied to cloud cavitations or bubble cavitations which would produce serious erosion. Also these observed cavitation phenomena correspond reasonably well with those simulated by the CFD computations.

Furthermore, from both the computed and observed cavitation data, it is obvious that very slow speed region flowing out from the tentatively blunt flange of the thruster housing causes the isolated sheet cavitation and should be trimmed fairly. Accordingly the Unit C was designed so as to have especially streamlined thruster housing. As shown in Figure 10-(c), it can be seen that there is no isolated sheet cavitation and consequently it could be no cavitation erosion risk. It is though that the causes of the isolated sheet cavitation are many factors. In order to identify the main cause, the flow field between the propellers and the strut rear ends should be investigated in more detail. Figure 12 shows the computed contour of axial component of flow velocity and distribution of flow vectors at the front of propellers and the bluish colored zones are very slow speed regions. It can be seen that the slow speed region is reduced fairly from Unit A to Unit C. Namely the low wake area behind the strut of Unit A spreads widely and flows into around the propeller blade roots, as a result the sheet cavitation continues to develop further and finally splits into two parts and changes to the harmful cloud cavitation. But the development of cavitation is avoided due to the reduction of wake by the streamlined thruster housing and moving the propeller backward from the rear end of strut.

4 STEERING MOMENT OF UNITS
As mentioned before, in order to reduce the strut drag in bollard pull conditions and the cavitation erosion risk in free running conditions, it is suggested to increase the relative distance between the ducted propeller and the strut as one of the countermeasures. But the measure could cause both increase of the steering moment of thruster units which could need more high-powered steering motors and the installation costs of the units onto vessels which for example, would construct hulls with larger holes to fit the units from the upper deck. Therefore it would be even more essential to design the geometries of strut optimally than to increase the relative distance between the ducted propeller and the strut in order to avoid the undesired increases in cost.

5 CONCLUSIONS
The influence of hydrodynamic interaction between the ducted propellers and the struts of the ducted azimuth thrusters on those performances in bollard pull conditions have been studied theoretically and CFD-numerical analytically. Then the cavitation erosion risk on the propeller blades of the ducted azimuth thrusters in free-
running conditions has also investigated CFD-numerical analytically and experimentally.
Consequently the following concepts should be considered for the optimal design of ducted azimuth thrusters in the same approach as optimal design for both propeller and stern shape of vessels, especially obtaining better thrust deduction factors,

1. In order to reduce the strut drag in bollard pull conditions, conceptually using Lagally’s theorem and the CFD computations, the smaller curvature radius of rear end of strut cross section and the larger distance between the ducted propellers and the strut rear ends should be taken into account as possible.

2. In order to avoid the cavitation erosion risk in free running conditions, first the thruster housings and struts should be streamlined properly without extremely blunt connected flanges if possible, next the more high-skewed propellers and the propeller positions farther from the strut rear ends should be considered from the CFD computation taking account of cavitation and the cavitation observation tests.

However, it should be noted for the arrangement of ducted propellers, since the farther distances between the ducted propellers and the strut rear end could cause the undesired increase of steering moment of the azimuth thruster units and its installation costs.

REFERENCES