Journal of
Ship&Ocean Technology
©MARCH 2004

SOTECH Vol. 8, No. 1, pp. 42 ~ 50, 2004

Numerical Simulation for the Rudder in order to Control the Cavitation Phenomena

KyungTae Boo[1], InHang Song[1] and SooChul Shin[1]

[1]Samsung Heavy Industries Co., Ltd. Shipbuilding & Plant Research Institute, Daeduk Science Town, Daejeon, Korea; E-mail: inhaeng.song@samsung.com

Abstract

In these ten years, the cavitation and erosion phenomena in the rudder have been increased for high-speed container ships. The cavitation in the rudder blades which is injurious to rudder efficiency is mainly caused by the main flow with a large angle of attack induced by propellers, and the erosion which occurs as a result of repeated blows by shock wave that cavitation collapse may produce was observed in the gap region of the rudder. However, gap cavitation is not prone to occur in model experiments because of low Reynolds number. So, the viscous effect should be considered for solving the flow of the narrow gap. In order to predict the cavitation phenomena and to improve the performance of the rudder, the analysis of the viscous flow in the rudder gap is positively necessary.

In this study, numerical calculation for the solution of the RANS equation is applied to the two-dimensional flow around the rudder gap including horn part and pintle part. The velocity and pressure field are numerically acquired according to Reynolds number and the case that the round bar is installed in the gap is analyzed. For reduced the acceleration force when fluid flow through the gap, modified rudder shape is proposed, It is shown that pressure drop can be highly restrained numerically and in model experiment, the cavitation bubbles can be reduced.

Keywords: rudder cavitation, cavitation phenomena, CFD

1 Introduction

In these ten years, ships were faster and larger, and the cavitation and erosion phenomena in the rudder frequently have been reported. The cavitation in the rudder blades is mainly caused by the main flow with a large angle of attack induced by propeller, and the erosion which occurs as a result of repeated blows by shock wave that cavitation collapse may produce was observed in the gap region of the rudder (Lambert, 1996). However, gap cavitation was not prone to occur in model experiment because the model size is too small than real scale. So, flow simulation by using CFD is tried for the flow around the rudder. Reynolds number in the real ship is so high that turbulence modeling is not good. However, numerical analysis can be done for the case that Reynolds number is higher than one of the model experiment, and more information can be added. In this study, we predicted the occurrence of cavitation phenomena, and proposed the plan that bubble occurrence could be delayed. And we could explain the difference between model and real ship by inquiring into Reynolds effect.

2 Numerical treatments

2.1 Governing equation and boundary conditions

The calculation is performed for two dimensional viscous flow, and standard k-ε model is used for turbulence (Launder/Spalding, 1974). For boundary condition, at the inlet, a constant velocity distribution is imposed, and pressure condition is used at the exit. And symmetric condition is used for the upper and lower side. Non-dimensional variable are used as follows:

$$C_P = \frac{2(P - P_\infty)}{\rho U_{in}^2}, \quad C_L = \frac{2L}{\rho U_{in}^2 c}, \quad Q = \frac{2 \int_{gap} \rho U dl}{\rho U_{in} c} \tag{1}$$

where P_∞ is the reference pressur and the same value with the exit pressure. L is Lift force and lift coefficient(C_{in}) is dimensionless with inlet velocity(U_{in}) and cord length(c). Q is the flux that pass by gap. And the density of the working fluid is 998.2kg/m^3 and the viscosity is 1.307x10^{-3} N·s/m^2.

2.2 Rudder geometry and grid system

Rudder shape of the general type is shown in Figure 1 is analyzed. In figure, Section I is used NACA0021, and section II is NACA0020. In this study, flow around section I is first calculated. And new section is selected from the acquired results and compared with old section II.

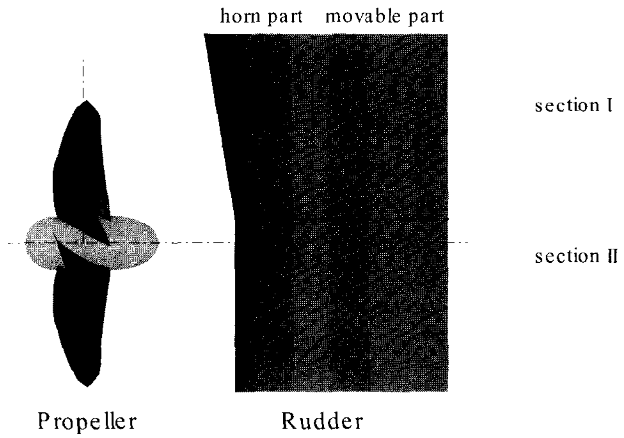
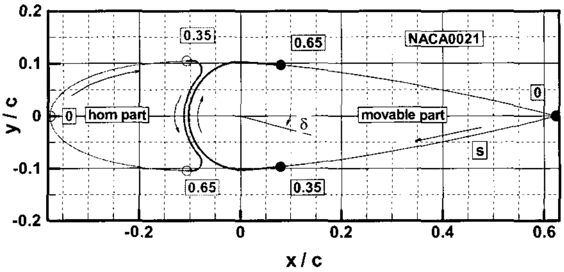
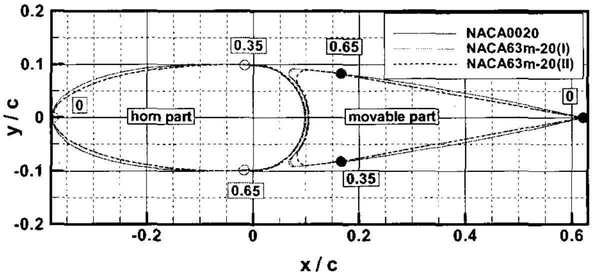


Figure 1: Propeller and rudder configuration

Figure 2 is shown the geometric information of two sections considered for calculation. Where, the girth-length in a clockwise direction is introduced that in the horn part, started from the leading edge and in the movable part, from the trailing edge. Where, c is the cord length.



(a)

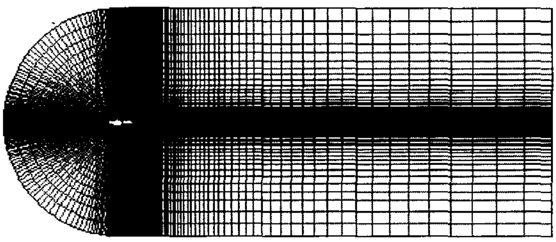


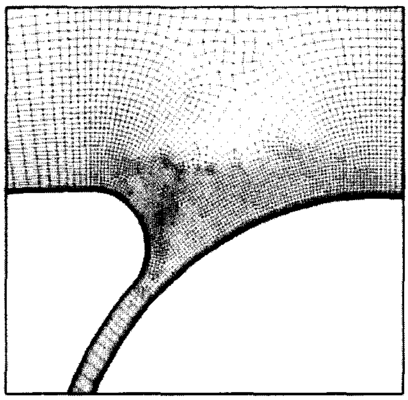
(b)

Figure 2: Rudder section; (a) Section I, (b) Section II

Where, the figure in the box is shown the value of s/s_{MAX}, and s is the girth-length.

The grid system is selected C type system as shown in Figure 3. A hybrid grid system is imposed on the computational domain, which is a combination of structured and unstructured grids. And the domain size is $-5<x/c<17$ and $-4.5<y/c<4.5$. Where, an unstructured grid system is established around the gap. The number of nodes is about 66000 to 68000.





(a) (b)

Figure 3: Grid arrangement and enlarged drawing for the gap;
(a) computational grids (b) unstructured grids established around the gap.

2.3 Section I

For the study of 2D turbulent flow around the horn rudder with variation of 'α', angle of attack, and 'δ', rotating angle of movable part, the calculation is performed about several types of fluid when the Reynolds number is 1.6×10^6, and pressure distribution on the section wall is compared. Figure 4 is presented obtained results by several numerical methods. The results of PANEL and EULER equation are predicting excessive pressure drop in the entrance and exit of the rudder gap, so the solution of the RANS equation is most realistic. Because the flow through the narrow gap in the rudder is much effected by viscosity.

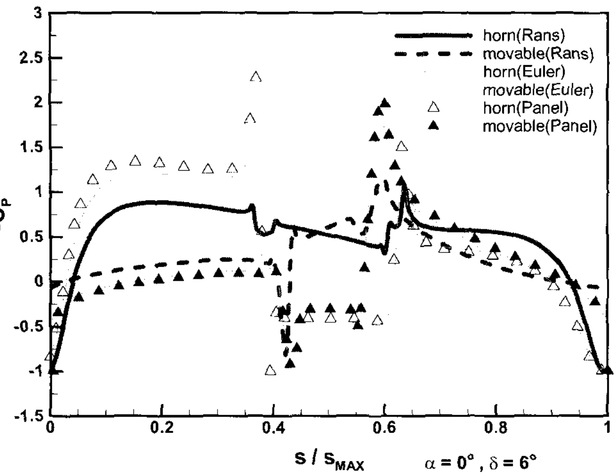


Figure 4: Pressure distributions (Re = $1.6x10^6$, NACA0021 section)

Streamline and pressure distribution obtained by RANS equation in case that the movable part is rotated by 6 degree is shown in Figure 5. At the lower front part of the movable part, stagnation point occurred. And, the possibility of cavitation would be higher at the upper front of the movable part because the pressure is the lowest.

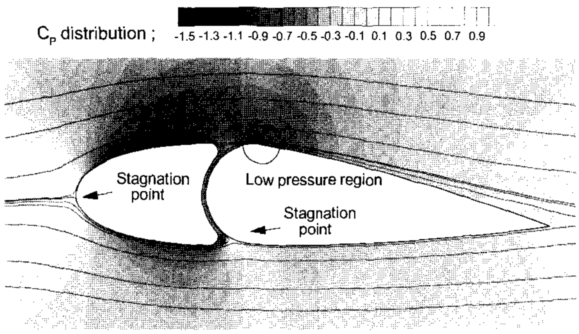


Figure 5: Obtained results by analyzing the RANS equation

Table 1: Lift coefficients and mass flux (Re = $1.6x10^6$, NACA0021 section)

α, δ	0°, 0°	0°, 6°	8°, 0°	8°, 6°	-8°, 6°
C_L	0	0.21439	0.45983	0.53306	-0.2038
Q	0	0.01262	0.01470	0.01663	-0.0028

Upper table indicates the flux and lift coefficients of equation (1). Where, flow that pass by gap shall be increase in proportion to lift. And when angle of attack is 8^0 and δ is 6^0, the flux and the lift coefficient are the highest. If the lift is high, flow that pass by gap is naturally fast. And flow shall be accelerated when the flow go out the gap in rudder in proportion to flux. Therefore, when lift force is larger, flux is increase, and pressure shall be the lowest in exit of the gap because of the flow acceleration for the turning. To restrain the cavitation occurring around the gap, adding an addition in the gap is tested in order that the flow resistance is increase, and flux is decrease.

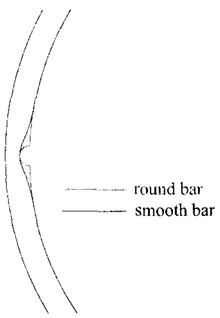


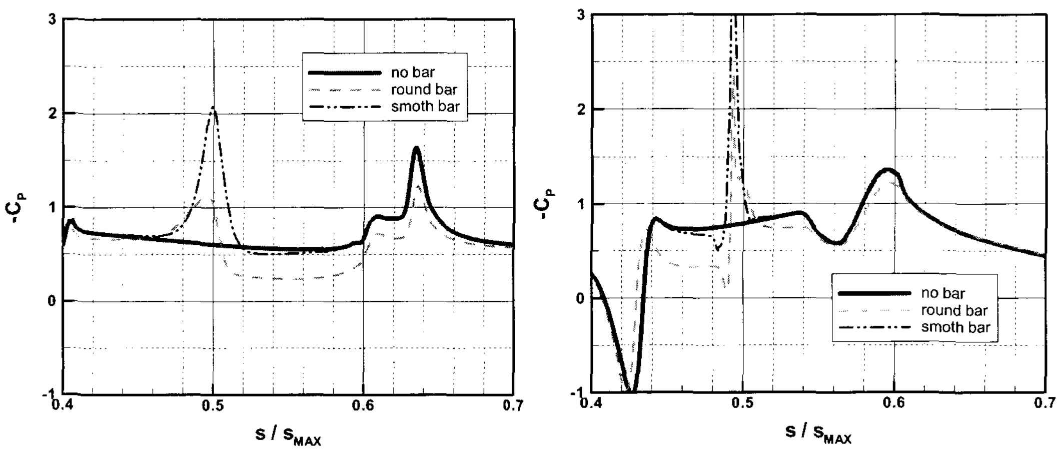
Figure 6: Rudder gap shape with addition

In this study, a round bar and a smooth bar are used with an addition. The geometry in each case is indicated on Figure 6, and the calculation performs in case that the Reynolds number is 6.4×10^6.

When the smooth bar is used, it is not almost decrease the flux to the gap in Table 2. So, the use of smooth bar can be done very well without. However, the use of round bar is shown better results, so round bar function as a flow resistance.

Table 2: Lift coefficients and mass flux ($\alpha=0^\circ$ $\delta=6^\circ$, Re = 6.4×10^6, NACA0021 section)

Bar	No	Round	Smooth
C_L	0.24458	0.24367	0.23416
Q	0.01766	0.01253	0.01752



(a) (b)

Figure 7: Pressure distribution in the gap: (a) horn (b) movable part

Upper conclusion is confirmed in Figure 7. In cases that there is no bar and round bar, the difference of the pressure distribution is little exist in the horn and movable part. However, in case that round bar is installed, dimensionless pressure is increase about 0.4 and 0.15 in each horn and movable part. But, it shall be noticed that the pressure in the bar (; s/s_{MAX}=0.5) is very low. So, the addition is effective to control cavitation occurrence in the gap exit, however shall be possible for an exchange of the one. And to get good results with an addition, optimizing the size of the bar and the geometry of the gap is more needed

2.4 Section II

In this case, flow is analyzed into the section shown in Figure 2 (b). Modified section has a thin leading edge for reducing the drag force and hollow section for increasing the lift force than NACA0020 section (Söding, 2000). Where, NACA63m-20(II) is that the curvature of the gap exit become enlarge for reducing the acceleration force in Figure 8. Because the pressure variation is linear with the reciprocal of curvature as shown in equation (2)

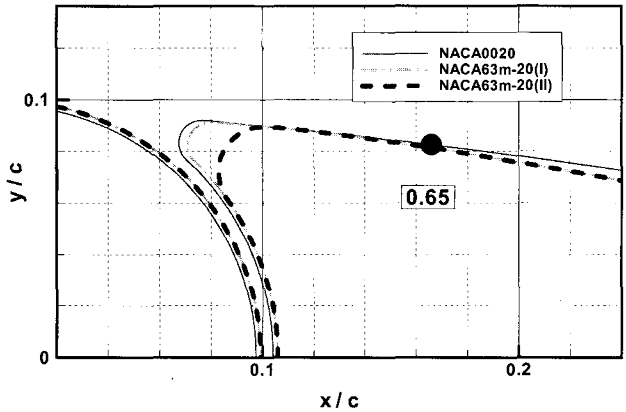


Figure 8: An enlarged figure of the rudder section.

$$\Delta P \propto m\frac{V^2}{R} \tag{2}$$

Where, m is mass and R is the curvature term.

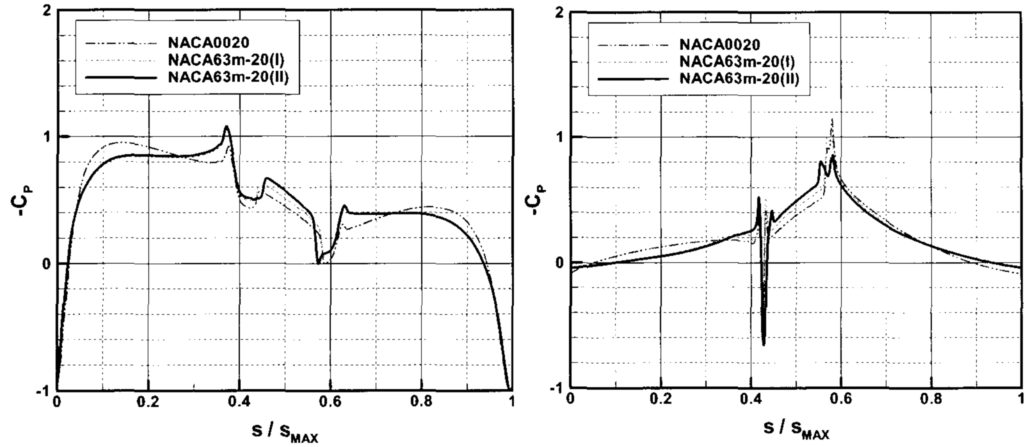
However, the area of the entrance becomes increase in Figure 8.

Table 3: Lift coefficients and mass flux (α=0° δ=6°, Re = 1.6×10^6)

Section	0020	63m-20(I)	63m-20(II)
C_L	0.296698	0.308372	0.316186
Q	0.008237	0.008465	0.008712

In the Table 3, flux is the largest in case of NACA63m-20(II) that the area of the entrance is the widest, though the variation of lift is needed more research to explain.

However, the pressure variation becomes reduce in Figure 7(b). It is the very reverse to the case that the use of addition reduce the flux in the gap and the pressure variation. Although flux becomes increase, the pressure does not fall because of the increasing of the curvature. So, increasing of the gap curvature is very effective method that pressure drop is reduced about 25% in Figure 7(b).



(a) (b)

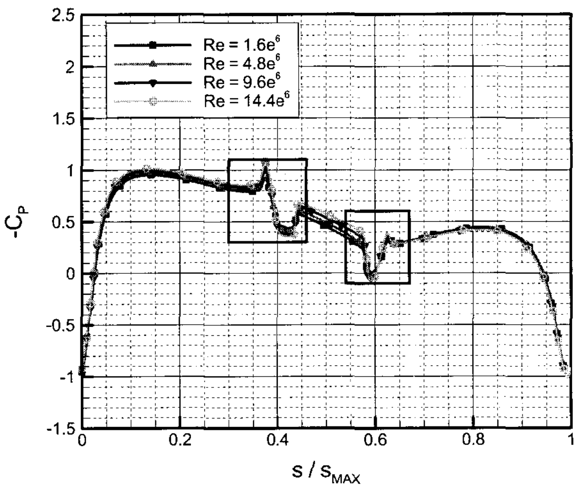
Figure 9: Pressure distribution in the gap: (a) horn (b) movable part

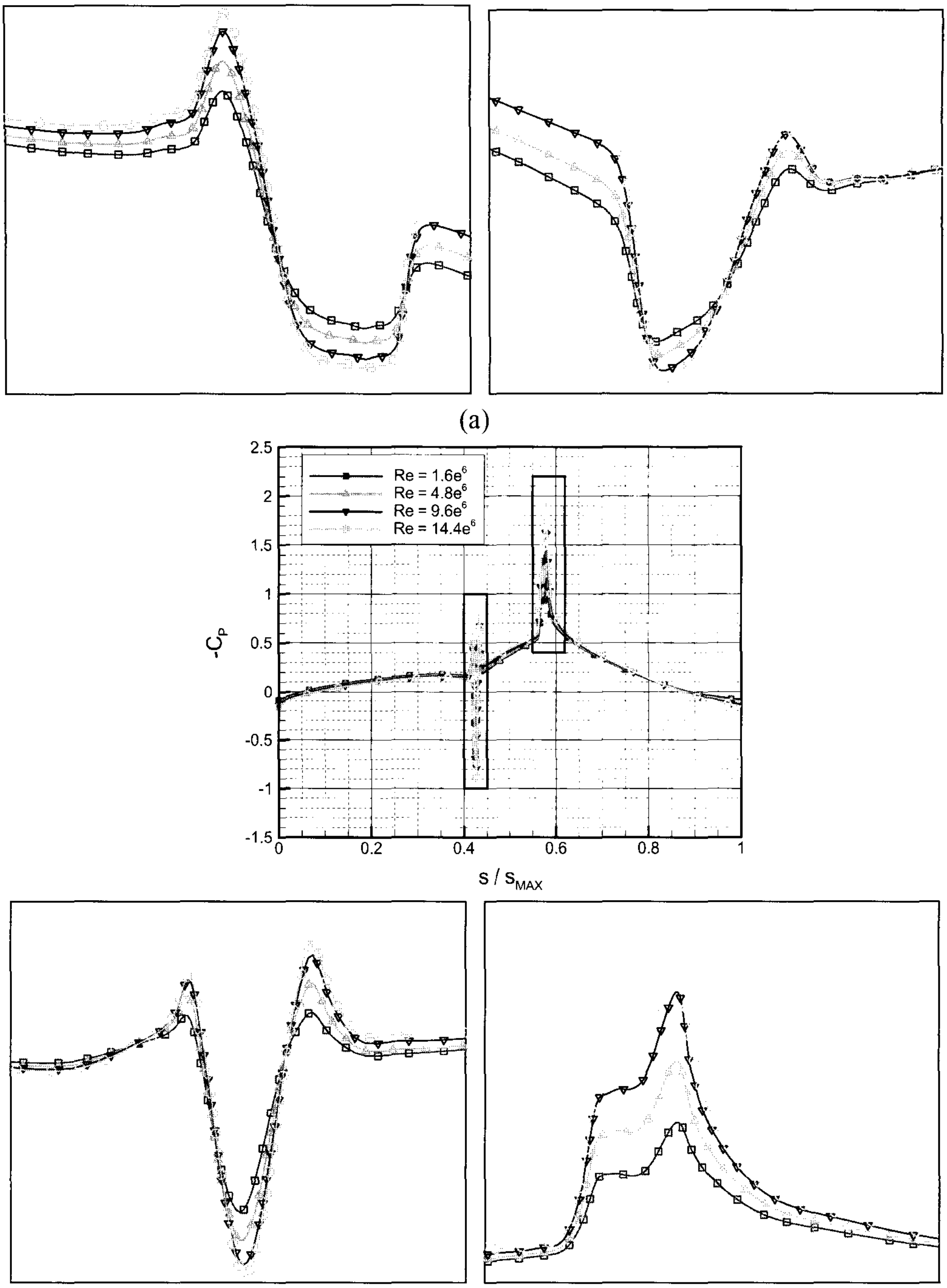
2.5 Reynolds effect

The fluid around the gap of the horn rudder would be strongly affected by changing of Reynolds number because of viscosity. The cavitation phenomena around the gap would be affected because of it. In this study, when $\alpha=0^o$ and $\delta=6^o$, the calculation is performed in cases that Reynolds number is 4.8×10^6, 9.6×10^6, 14.4×10^6 in section II.

Figure 8 is obtained pressure distribution. In figure, when Reynolds number is increase about nine times, pressure variation is about 70% together. Therefore Reynolds effect exists, and we estimate that pressure variation will be very large and cavitation be well occur in case of real ship that flow has a high Reynolds number.

In general, physical value is little change regardless of the increasing of Reynolds number. However, in case that there is excessive vortex like as cavitation flow, the strength of the vortex is increase with Reynolds number, together. If the strength is increase, the pressure variation is increase, too.





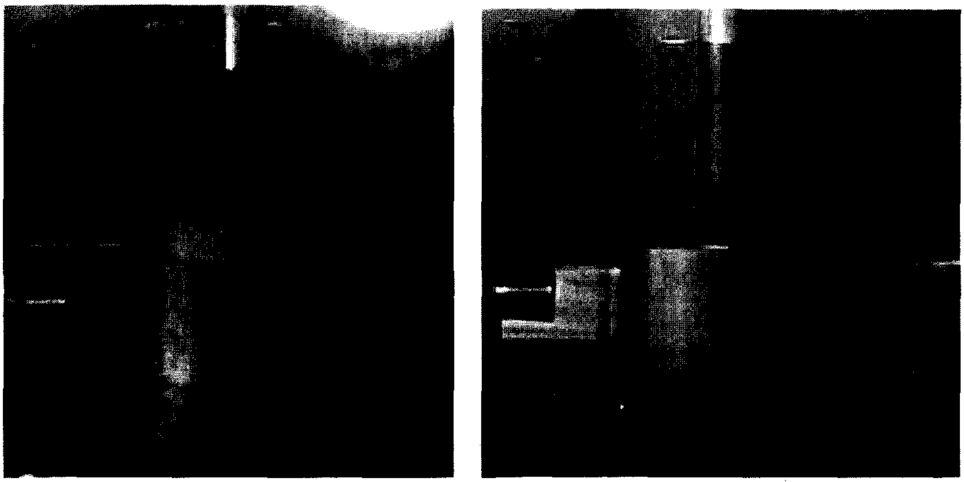
(b)

Figure 10: Pressure distribution according to Reynolds number: (a) horn (b) movable part

3 Results

In this study, numerical calculation for the solution of the RANS equation is applied to the two-dimensional flow around the gap of the rudder section, and investigated several planes

that bubble occurrence could be delayed.
Numerically, Cavitation phenomena were first predicted at the upper front of the movable part. And, pressure variation that is raised the cavitation possibility was in proportion to lift and flux that pass by the gap inside.
For reducing the flux, the installing of the round bar in the gap inside was very effective to control the cavitation. The round bar could be reduced the flux as a flow resistance and as a result, pressure variation was reduced. But, smooth bar did not cause any effect as a flow resistance. However, bars should be possible for an exchange with the new one.
Enlarging of the curvature in the gap exit increased the flux in the gap, but pressure variation was decrease on the contrary. Because the acceleration force is linear with flux and the reciprocal of the curvature as shown in equation (2). For the good effect, optimizing the size of the bar ad the geometry of the gap was needed.
We also explained that pressure variation would be larger in case that flow has a high Reynolds number, so cavitation would be well occur in real ship case.



(a) (b)

Figure 11: Cavitation visualization test (STBD 8^0): (a) Original (b) Modified rudder

References

Lambert, D. 1996. Cavitation-Minimizing Its Destructive Effects in Control Valves Control Engineering, Don-Donnelley Publishing Corp.

Peric, M. 1985. A Finite Volume Method for the Prediction of Three-Dimensional Fluid Flow in Complex Ducts, Ph. D. Thesis, Univ. of London.

Söding, H. 2000. Limits of Potential Theory in Rudder Flow Predictions, Twenty-Second Symposium on Naval Hydrodynamics, 622-637.

Launder, B.E. and D.B. Spalding 1974. The Numerical Computation of Turbulent Flows, Comp. Meth. Appl. Mech. and Eng., **3**, 269-289.