SUMMARY

Cavitating flow around a Clark Y 11.7% hydrofoil in water tunnel is studied both experimentally and numerically in order to acquire the knowledge about the cavitating flow field and its relation to hydrofoil performances. In the experiment, time-averaged pressure and velocity distributions around the hydrofoil are measured in cavitating conditions to evaluate the cavitation performance of hydrofoil. In addition, the unsteady cavitating flows are visually observed using high-speed video camera. Numerical simulation of cavitating flow is also carried out for the same test apparatus (but two-dimensional). The angle of attack of 8 degrees is chosen for the present investigation.

It is confirmed experimentally that, as the cavitation number is decreased, the estimated lift slightly increases from that in the non-cavitating condition just before the sudden breakdown, which has been reported by the past literatures. However, this is not observed in numerical simulations, and the critical cavitation number, at which the breakdown starts to occur, is seen to be different between the experiment and the numerical simulation, revealing the insufficient prediction ability of present numerical simulation. In the experiment, the oscillating sheet cavity with cloud cavities shed from its trailing edge seems to sustain the low pressure downstream of the partial cavity, which keeps the lift unchanged from that in non-cavitating conditions. On the other hand, the simulated sheet cavity with short length is rather stable, and pressure behind the cavity increases. As a result, the lift starts to decrease with the formation of short partial cavity in the numerical simulation.

INTRODUCTION

The recent rapid progress of computer science including hardware technology as well as numerical simulation technology has enabled us to simulate the cavitating flow and achieve the qualitative, and in some extent, the quantitative agreements with real flow from basic ones such as cavitating hydrofoils and high-speed projectiles to practical ones such as cavitating pumps and hydroturbines. However, such cavitation CFD (Computational Fluid Dynamics) still often fails to predict the cavitation performance in some cases. For example, The industry-university collaborative research project on cavitation CFD organized by Turbomachinery Society of Japan has carried out benchmark tests with several CFD codes for the cavitating flow around two types of hydrofoils, NACA0015 and Clark Y 11.7% (see Kato [1]). It has been shown that none of the cavitation models can correctly predict the sudden breakdown of the lift coefficient as observed in experiments, partially due to the under-prediction of the cavity length.

Since, for the development of more reliable cavitation CFD solvers, it is necessary to provide the sufficient data on the unsteady cavitating flow structures, we have activated the experimental investigations of cavitating Clark Y 11.7% hydrofoil (Watanabe et al. [2]) partly under the above-mentioned project. In the present paper, the hydrofoil performance in the cavitating conditions and its relation to the unsteady cavitating behaviors and flow fields are experimentally and numerically investigated.

EXPERIMENTAL APPARATUS

Present experiments are done using a cavitation tunnel in Kyushu University (Watanabe et al. [3]). This tunnel has a rectangular test section with the width (height) of 200mm and the span of 81.5mm. Test obstacles can be set horizontally at the center on one of the side walls with arbitrary angle of attack. In the present study, a two-dimensional Clark Y 11.7% hydrofoil with the chord length of \(C=100\) mm and the span of \(B=81.0\) mm as shown in Fig. 1 is tested. The tunnel height to the chord length ratio is 200 mm/100 mm \(=2.0\), indicating a substantial blockage effect of tunnel walls. The top, bottom and one of side walls of the test section consist of a transparent acrylic resin for visual observations. The cavitating flows are filmed from the top and the side simultaneously using two high-speed cameras.
The pressure measurements around Clark Y hydrofoil have been done for several cavitation numbers in our past study [2], using the model fitted with totally 25 pressure tap holes (1 at the leading edge, 18 and 6 respectively on the suction and pressure surfaces) as shown in Fig. 1 (a). The locations of taps are, on the suction surface, 3, 6, 10, 15, 20, 25, 30, 35, 40, 45, 50, 55, 60, 65, 70, 75, 80 and 85% of chord length from the leading edge, while 10, 20, 30, 40, 50 and 60% on the pressure surface. The hole sizes are 0.5 mm for all the taps, which are connected through transparent polyurethane tube and hand-operated scanning valves to the piezo-resistive absolute pressure transducers (GE Measurement & Control Solutions, UNIK5000). The accuracy of the transducer itself is 0.2 kPa, and the overall error estimation is at most 1kPa. The accuracy of the reference location (200 mm upstream from the mid-chord), \( \rho \) the density of water.

In every experiment, the amount of dissolved oxygen is checked to be less than 2 ppm before and after the experiments to keep the similar conditions in terms of water quality.

### Pressure distribution measurements

The pressure measurements around Clark Y hydrofoil have been done for several cavitation numbers in our past study [2], using the model fitted with totally 25 pressure tap holes (1 at the leading edge, 18 and 6 respectively on the suction and pressure surfaces) as shown in Fig. 1 (a). The locations of taps are, on the suction surface, 3, 6, 10, 15, 20, 25, 30, 35, 40, 45, 50, 55, 60, 65, 70, 75, 80 and 85% of chord length from the leading edge, while 10, 20, 30, 40, 50 and 60% on the pressure surface. The hole sizes are 0.5 mm for all the taps, which are connected through transparent polyurethane tube and hand-operated scanning valves to the piezo-resistive absolute pressure transducers (GE Measurement & Control Solutions, UNIK5000). The accuracy of the transducer itself is 0.2 kPa, and the overall error estimation is at most 1kPa taking account of the observed resolved gas column in the polyurethane tube. The time-averaged data are used to calculate the following pressure coefficient distribution \( C_p(x) \) from the measured pressure \( p(x) \).

\[
C_p(x) = \frac{p(x) - p_{ref}}{p U^2/2} \quad (2)
\]

### Velocity measurements

We investigate the velocity field around the hydrofoil by one-dimensional Laser Doppler Velocimetry (LDV) with spherical nylon particles (mean diameter 4.1\( \mu m \) and specific weight 1.02) for seeding particles. As shown in Fig. 1 (b), measurements are done at 11 points along the coordinate normal to the hydrofoil surface (minimum distance from the wall is 1.5mm), at every 10% locations from the leading edge along the chord on the suction surface. Only the velocity component tangent to the hydrofoil surface is measured in a non-cavitating condition as well as at cavitation number of \( \sigma=1.6 \), where the partial cavity with cloud cavity shedding is observed. The upstream free-stream velocity and the downstream velocity distributions are also measured at 100mm upstream and 83mm downstream from the mid-chord of hydrofoil respectively, for several cavitation numbers.

### NUMERICAL SIMULATION

A numerical simulation is carried out for tested Clark Y hydrofoil, using an open source software, OpenFOAM, which implements several cavitation models; we use interPhaseChangeFoam, which is an incompressible Navier-Stokes solver with homogeneous cavitation model considering the phase change between liquid and vapor phases. For the vapor production/destruction due to evaporation/condensation, Kunz model [4] [5] is employed, which solves the following transport equation of liquid phase volume fraction \( \alpha_l \).

\[
\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l \mathbf{U}) = \frac{1}{\rho_l} \left( \dot{m}_l + \dot{m}_v \right) \quad (3)
\]

\[
\dot{m}_l = C_{prod} \rho_l \alpha_l \min[0, \rho - \rho_v] \quad (4)
\]

\[
\dot{m}_v = C_{prod} \rho_v \alpha_l^2 (1 - \alpha_l) / \tau_v \quad (5)
\]

where \( x \) and \( \mathbf{u} \) is Cartesian coordinates and velocity component respectively, \( t \) and \( t_v = C/U \) are time and characteristic convection time, and \( \rho_l \) and \( \rho_v \) are liquid phase and mixture density. The empirical constants for production and destruction of liquid phase are given as \( C_{prod}=C_{prod}=1000 \). The RNG-\( k-\varepsilon \) turbulence model is employed, which gives a nice agreement of non-cavitating pressure distribution with the experiment, as shown later.

The computational domain considered here is a two-dimensional one, but the chord length and the tunnel width are the same for the experimental ones (Fig. 3). Inlet and outlet boundaries locate at 5C upstream and downstream from the mid-chord of hydrofoil. As boundary conditions, the velocity is fixed at the inlet, and the static pressure is fixed at the outlet. The non-slip flow condition is applied on the hydrofoil surface, while the free-slip flow condition on the tunnel wall boundaries.
to reduce the number of nodes. The total number of nodes in the whole computational domain is 82,816.

RESULTS AND DISCUSSIONS

Pressure distribution around hydrofoil

Figure 4 shows the measured pressure coefficient \(C_p\) distributions in various cavitation numbers. We can see a strong suction peak near the leading edge in the non-cavitating condition of \(\sigma=3.01\). At \(\sigma=1.92\), the short partial cavity with its leading edge showing the fingering pattern is observed. We can see the flat distribution of pressure coefficient \(C_p\) with the value of about -1.9 near the leading edge on the suction side, meaning that this flat \(C_p\) area with \(C_p= -\sigma\) is a cavitating region. Lower the cavitation number is, the region of the flat \(C_p\) distribution on the suction side becomes wider, indicating the development of cavitation. At \(\sigma=1.44\), the leading edge of cavity becomes uniform, and the unsteadiness of cavity becomes remarkable due to the formation of large scale cloud cavity from the trailing edge of the cavity. At \(\sigma\) less than 1.28, substantial cavity volume oscillation becomes predominant, inducing large pressure fluctuations upstream and downstream of the hydrofoil. Finally at \(\sigma=0.46\), the cavity is already stable super-cavitation.

By integrating the \(C_p\) distribution along the hydrofoil surface, we can estimate the pressure forces as follows.

\[
\frac{F_P}{\rho U^2 BC / 2} = \left(C_D, C_L\right) = \frac{1}{C_p} \int C_p \, d\sigma
\]  

where \(F_P\) denotes the pressure force, and \(C_D\) and \(C_L\) the drag and lift coefficients due to the pressure force. Numerical integration has been done with the linear interpolation of the pressure coefficient \(C_p\) shown in Fig. 4, and the estimated values of \(C_L\) is plotted against the cavitation number \(\sigma\) with red circles in Fig. 5. In this figure, the lift coefficient directly measured by the force balance by Numachi [6] is also plotted with blue triangles. The tunnel height to chord length ratio is 190mm/70mm=2.71 in Numachi’s experiments, which is larger than 2.0 of the present experiments. Then, the blockage effect is expected to be more significant in the present study.

By comparing the present results with the Numachi’s experiments, we can see a qualitative agreement; as the cavitation number is decreased, the estimated lift slightly increases from that in the non-cavitating condition just before the sudden breakdown. The present study underestimates the lift coefficient; The stagnation point near the leading edge locates on the pressure side, but it cannot be recognized in the pressure distribution shown in Fig. 4, due to the sparse measurement points, which results in the underestimation of the lift coefficient. The breakdown cavitation number is larger in the present study. This is partly due to the stronger blockage effect in the present study.

Unsteadiness of sheet cavity

Figure 6 summarizes the time-averaged sheet cavity length and its fluctuation range plotted against the cavitation number \(\sigma\). As \(\sigma\) is decreased, the cavity becomes longer and starts to oscillate. For \(\sigma=1.3\) (with angle of attack of \(\alpha=8^\circ\)), which corresponds to the time-averaged cavity length less than 50% chord and the maximum length less than 100%, so-called “partial cavity oscillation” (Watanabe et al., [7]) is observed with the formation of the cloud cavity from the trailing edge of sheet cavity. For \(\sigma<1.3\), another instability, “transitional cavity oscillation” is observed, where the cavity significantly oscillates from cavitation-free state to super cavitation with several cloud cavity shedding during one cycle of oscillation.

Figure 7 shows the time evolution of the sheet cavity length and the time-averaged pressure coefficient \(C_p\) distribution around the hydrofoil for the cavitation number of \(\sigma =1.65\), where the lift coefficient increase is observed as shown in Fig. 5. The locations of trailing edge of sheet cavity with the time-averaged, maximum and minimum cavity length are indicated by broken lines. In the former region of sheet cavity (approximately \(x/C<0.3\), the pressure taps there are covered by
the cavity during the most part of oscillation cycle, and the time-averaged pressure is almost equal to the vapor pressure ($C_p = -\nu$). On the other hand, in the rear region of the sheet cavity, the pressure taps are exposed from the cavity during some part of the oscillation cycle, and then the pressure gradually increases in this region.

Figure 8 shows the typical example pictures of sheet cavity from side views during one cycle of partial cavity oscillation for $\sigma = 1.64$. The sheet cavity elongates from $x/C=0.2$ to $x/C=0.6$ during $t=0.375-0.3925s$. After that, the cloud cavity is formed and convected downstream, and then the sheet cavity starts to elongate again. The partial cavity oscillation is maintained by such a process in this condition.

Recalling the cavitation performance of hydrofoil shown in Fig. 5, we can notice that the lift coefficient $C_L$ at $\sigma = 1.92$ is almost the same as that in the non-cavitating condition, whereas the slight increase in $C_L$ can be seen at $\sigma = 1.64$ and 1.44. At $\sigma = 1.92$, the fluctuation of sheet cavity is not very significant, and the pressure downstream of sheet cavity rapidly increases from the vapor pressure as shown by red closed circles in Fig. 4. On the other hand, at $\sigma = 1.65$ and 1.44, the fluctuation of sheet cavity length becomes significant, and the pressure recovery downstream of the sheet cavity is more gradual than that at $\sigma = 1.92$, contributing the slight increase of $C_L$ at $\sigma = 1.65$ and 1.44. It seems that the unsteady behaviors of sheet and cloud cavities are very important factors for the cavitation performance of hydrofoil especially around the breakdown cavitation number.

**Velocity distribution**

Figure 9 shows the measured results of (a) upstream free-stream velocity at various cavitation numbers $\sigma$, (b) velocity distributions downstream of hydrofoil at $\sigma = 2.2$, 1.6 and 1.1, and (c) velocity distributions around suction surface of hydrofoil at $\sigma = 2.2$ and 1.6. The averaged velocities (closed symbols) with the standard deviations (open symbols) are shown in the figure. Please note that the averaged velocities are not time-averaged ones, but are calculated by averaging velocities of randomly flowing particles. Then, the velocity are somewhat biased especially near the sheet cavity and inside the cavitating wake. We can notice that the upstream free-stream velocity is almost constant with small standard deviation for all the examined values of $\sigma$, meaning that the free-stream velocity is almost steady despite of the unsteadiness of cavitation. In non-cavitating condition ($\sigma = 2.2$, in Fig. 9 (c)), the development of boundary layer toward the trailing edge can be clearly seen, but the velocity distribution around the hydrofoil is almost steady. At $\sigma = 1.6$, the velocity is almost steady except near the cavity surface for $x/C<0.3$, where pressure is almost constant with vapor pressure as shown in Fig. 7. On the other hand, the averaged velocities near the hydrofoil surface downstream ($x/C>0.4$) are remarkably decreased compared to those in the non-cavitating conditions, indicating the existing large wake from the partial cavity. The large standard deviations are observed even far from the hydrofoil surface, which are probably caused by periodical induced velocities due to vortical cloud cavity shedding. This large wake still remains downstream of hydrofoil (Fig. 9 (b)), even though the cloud cavities have already disappeared during their convolutions.

**Numerical simulation**

The lift and drag forces obtained by CFD are plotted against the cavitation number in the former figure of Fig. 5. The time averaged, maximum and minimum partial cavity lengths from CFD are also plotted in Fig. 6. We can see qualitative agreements of evolutions of lift and drag forces and the cavity lengths against the decrease of cavitation number between the experimental and numerical results. However, the breakdown of lift force starts to occur at larger cavitation number in CFD than in the experiments, despite that the cavity lengths are underestimated in CFD.
Figure 10 shows the comparison of time-averaged pressure distributions between CFD and experiments at three different cavitation numbers $\sigma$. A quantitative agreement is clearly observed in non-cavitating condition. At $\sigma=1.63$ (1.64), which is just before the lift breakdown in the experiment, low pressure region downstream of closure location of sheet cavity is observed only in the experiment, perhaps due to the differences of unsteady motions of cavitation, as shown later. Since the lift force is mainly determined by the pressure distribution around the hydrofoil, this low pressure region sustains the lift before the breakdown in the experiment, whereas the lift decreases even with short sheet cavity at this cavitation number in CFD.

Figure 11 shows (a) time evolutions of partial cavity length, lift coefficient, and drag coefficient, and (b) typical example pictures of sheet cavity (contour map of vapor void fraction), for $\sigma = 1.63$. Since the periodic motion of sheet cavity is
observed, data for one cycle of oscillation are plotted in the figures. Although the sheet cavity length remarkably oscillates, the formation of cloud cavity is not at all observed in CFD, which is very different from the experimental observation shown in Fig. 8. This seems to be a cause of the discrepancy of pressure distribution in the closure region of sheet cavity, where the violent cloud cavity shedding is observed only in the experiment.

Figure 12 shows the comparisons of velocity distributions around and downstream of hydrofoil between experimental and numerical results at around $\sigma=1.6$. It is clear that, near the closure region of sheet cavity ($x/C=0.4-0.5$), the standard deviations of velocities above the sheet cavity are much larger in the experiment than those in CFD, which is due to the formation of cloud cavity around there in the experiment. Toward downstream from there, the standard deviations in the experiment gradually decay. The wake of sheet cavity is thinner in the experiment than that in CFD, which probably makes a difference in pressure distributions among them. Then, it is conjectured that the dynamics of the closure region of sheet cavity should be more appropriately taken into account for the development of more reliable CFD simulation; more detailed information are expected toward it.

CONCLUDING REMARKS

In the present study, the measurements of pressure and velocity distributions around a Clark Y 11.7% hydrofoil were carried out as well as the high-speed video observations of cavitation. The relations among the lift and drag forces, flow fields, and the unsteady behaviors of cavitation were discussed. In addition, the cavitation CFD based on the existing homogeneous cavitation model, which could not quantitatively reproduce the lift breakdown characteristics of hydrofoil, was carried out, and the difference of flow fields from the experimental ones were investigated.

From the measured data, the unsteady motion of sheet cavity associated with the periodic shedding of vortical cloud cavity seemed to sustain the low pressure downstream of closure region of sheet cavity, just before the lift breakdown of hydrofoil. This could not be reproduced by one of the standard cavitation CFD done in the present study, resulting in the occurrence of earlier lift breakdown (larger critical cavitation number) in CFD. Perhaps the dynamics of the closure region of sheet cavity should be more appropriately taken into account for the development of more reliable CFD simulation.

ACKNOWLEDGMENTS

This study is partly supported by KAKENHI (21686019). The part of this study has been carried out under the industry-university collaborative research project on cavitation CFD (Project leader, Prof. C. Kato of The University of Tokyo.)

REFERENCES


