Cavitation CFD Prediction for NACA0015 Hydrofoil Flow Considering Boundary Layer Characteristics

[1]Motohiko Nohmi*; [1]Tomoki Tsuneda; [1]Byungjin An; [1]Takayuki Suzuki; [2]Satoshi Watanabe; [2]Shin-ichi Tsuda

[1]EBARA Corporation, Fujisawa-shi, Kanagawa, Japan; [2]Kyushu University, Motooka Nishi-ku, Fukuoka, Japan

Abstract

The accuracy of cavitation CFD is not sufficient for the flow around very simple geometries such as single hydrofoil. The lift force and the cavity length are underestimated particularly for the high angle of attack cases. One of the possible reasons of the discrepancy is the singular characteristic of sheet cavitation in the boundary layers that is not taken into account in the recent cavitation models. It was observed that the sheet cavitation is not initiated in the laminar boundary layers. The inception and growth of the sheet cavitation occurs at the reattachment point of the laminar boundary layer separation or at the transition point of the laminar boundary layer to the turbulent boundary layer. These characteristics were reported in many literatures however they are not well modeled for cavitation CFD. In this paper the laminar boundary layer zone is identified for NACA0015 hydrofoil from the high speed video observation. Then, cavitation CFD is carried out by deactivating the cavitation model at the estimated laminar boundary layer zone. This modified cavitation CFD results are discussed and compared to conventional cavitation CFD results and measurement results.

Keywords: cavitation CFD; NACA0015 hydrofoil; boundary layer

Introduction

Cavitation CFD prediction functions are built in many commercial codes currently, and are frequently used in the development and design of turbomachinery such as pumps, hydraulic turbines and the ships propellers. On the other hand, whether the analysis accuracy of them is sufficient or not remains a matter of debate. In Japan, the effect of the cavitation coefficient on the lift force of a hydrofoil was analyzed with various kinds of cavitation CFD codes and the prediction performance was evaluated in the "Industry-University Collaborative Research Project on Numerical Predictions of Cavitating Flows in Hydraulic Machinery" conducted in Turbomachinery Society of Japan from 2009 to 2011. As a result, it was found that none of these codes gave sufficient analysis accuracy, and the reliability of cavitation CFD came into question.(1) Responding to this situation, several groups began studies after the project to track down the cause and discuss improvement measures.(2-5) As part of them, this study also conducts an analysis of the NACA0015 hydrofoil that was analyzed in the above project, considering the characteristics of cavitation in a boundary layer that was not considered sufficiently in the existing analysis, and evaluates its effect on analysis accuracy.

Target of the Analysis

The target of the analysis is the flow around the NACA0015 hydrofoil installed in a cavitation tunnel. The result of an experiment conducted in Marine Propeller Cavitation Tunnel at The University of Tokyo is used as the verification data(1). The analytical domain is determined according to this experiment. The shape of the cross section of the measuring part in the tunnel is 600 mm in length and 150 mm in width. The chord length of the NACA0015 hydrofoil is 150 mm and the width 150 mm. In this study the analysis is conducted under the conditions of the inlet flow velocity = 8 m/s (constant) and the angles of attack = 6° and 8°. The analytical domain ranges up to 5C upstream from the center of the chord and down to 5C downstream from it, where C is the length of the chord. For details of other analysis conditions, see the reference (1). The analysis code used in this study is a commercial cord, ANSYS-FLUENT. In a two-dimensional steady analysis, the number of grids is about 90,000, the turbulent model is SST-k-ω model, and the cavitation model is the Zwart-Gerber-Belami model(6).

In this study, the analysis result is arranged using the lift coefficient, C_l, with the cavitation coefficient, σ, and the static pressure coefficient, C_p, defined below:

*Corresponding Author, Motohiko Nohmi: nohmi.motohiko@ebara.com

$$\sigma = \frac{p_\infty - p_V}{\frac{1}{2}\rho_l u_\infty^2} \quad (1) \qquad C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_l u_\infty^2} \quad (2) \qquad C_l = \frac{F_l}{\frac{1}{2}S\rho_l u_\infty^2} \quad (3)$$

where p is static pressure, p_∞ and u_∞ are the average static pressure of flow and the flow velocity at the inlet of the analytical domain respectively, p_V is saturated vapor pressure, ρ_l is the density of single-phase water, F_l is lift force, and S is the area of blade.

Characteristics of Cavitation in a Boundary Layer

The scale of the microscopic behavior of bubbles those form cavitation is extremely small compared to the representative dimensions of pump impellers, hydraulic turbine runners, and the blades of a ship propellers. It is therefore difficult to directly analyze the overall flow of fluid machinery using grids with a size that can track or capture the surface phenomena of cavitation bubbles. Thus, most of the current cavitation CFD use a so-called cavitation model, in which grids that can resolve the flow around the blades of fluid machinery are used, and cavitation phenomena with a sub-grid scale is modelled using simplified equations. Many kinds of cavitation models have been proposed. Here, as a representative example, the model by Chen and Heister shown in Eq. (4) is introduced which was proposed in an early stage and therefore simple, and whose physical meaning is easy to understand.(7) The Zwart-Gerber-Belami model mentioned above is one of the models developed based on the model by Chen and Heister.

$$\frac{D\rho}{Dt} = c(p - p_V) \quad (4)$$

where ρ is the local average density of the mixed fluid of gas (vapor phase) and liquid phase, and p is the static pressure defined locally in the mixed fluid. In this equation it is assumed that when there is a difference between the local static pressure and the saturated vapor pressure in a mixed fluid of water and vapor, the progress of phase change depends on the magnitude of the difference, with 'c' being the empirical constant that determines the change rate. If the local static pressure equals to the saturated vapor pressure, phase change does not progress anymore, resulting in an equilibrium state with a constant mixed-fluid density. In other words, it is assumed in this model that a water-vapor system has a nature of proceeding to an equilibrium state. This is an easy to understand concept, but it has to be noted that there are many cases familiar to us where an equilibrium state is not easily reached such as supercooling and supersaturation.

As a result of a literature research in this study, it became apparent that many cases were reported in actual cavitation where an equilibrium state was not reached in a short time and a nonequilibrium state continued. For example, it was reported by Huang and Peterson that the occurrence of cavitation on a hydrofoil was limited to the location of separation-reattachment of a laminar boundary layer or the location of transit from a laminar boundary layer to a turbulent boundary layer. Therefore, they point out that the characteristics of Reynolds numbers and boundary layers differ among blades with different sizes even if their shape is identical, and as a result, the positions of the occurrence of cavitation also greatly differ among them, leading to the difference between small-scale model tests and actual blade tests.(8) Kuiper reported a similar result, showing that cavitation did not occur when a laminar boundary layer remained on a hydrofoil even if the absolute static pressure was negative, meaning that a tensile force existed in water.(9) Kato et al. briefly summarized in his book the occurrence conditions and types of cavitation as a flowchart, which showed a result similar to the above ones.(10) From the above results, it is anticipated that a cavitation analysis of hydrofoils that does not appropriately consider the characteristics of cavitation in a boundary layer does not give sufficient accuracy. Phase change of liquid water does not occur in a laminar boundary layer, and cavitation occurs only after fluctuations by turbulent flow transition. It resembles the bumping phenomena occurring by stimulation after a supersaturation (superheated) state is reached by gentle heating or rapid heating of still water.

On the basis of the above discussion, an important subject of this study is to predict with high accuracy the separation-reattachment of a laminar boundary layer on a hydrofoil or the transition from a laminar boundary layer to a turbulent boundary layer. An option to achieve this is to conduct a high-accuracy three-dimensional turbulent analysis such as LES and identify, using that result, the location where a laminar boundary layer disappears. However, whether LES should be adopted or not is a continuous matter at present because an LES analysis requires considerable computational resources/loads, and to what degree the result of a single-phase LES analysis can be applied to the occurrence of cavitation is not clear in consideration of the possibility that cavitation itself may change the characteristics of a boundary layer. In this study, therefore, we decided to obtain it from the image of the experimental result as a simpler and more reliable method. By observing the high speed motion pictures of the occurrence of cavitation on the NACA0015 blade at angls of attack of 6° and 8°, it was found that the front edge location of sheet cavitation did not change very much when the cavitation coefficient changed. It was also found that the front edge location of sheet cavitation was not influenced very much by the unsteady fluctuation of cavitation on the blade. The front edge location of sheet cavitation was measured based on the image, and it was 7.8 % of the chord length downstream from the forward edge of the blade for the angle 6°, while 5.5 % of the chord length downstream for the angle 8°. In this study, using these values, and on the assumption that a laminar boundary layer is kept from the stagnation point at the leading edge of the hydrofoil blade to the location of these values although this is considerably an artificial assumption, the cavitation model is deactivated upstream of these points and it is applied downstream of them. In the next chapter, the case this method is adopted is compared with the case the cavitation model in a standard manner is used.

Results and Discussion

Fig. 1 shows the comparison between the experimental value of distribution of the static pressure coefficient, C_P, on the blade under the no-cavitation condition, and the calculated value by an incompressible analysis not considering cavitation for angls of attack of 6° and 8°. As seen in Fig. 1, there is close agreement between the experimental value and the calculated value. The validity of the incompressible analysis, which is the start point of the cavitation analysis, was ascertained.

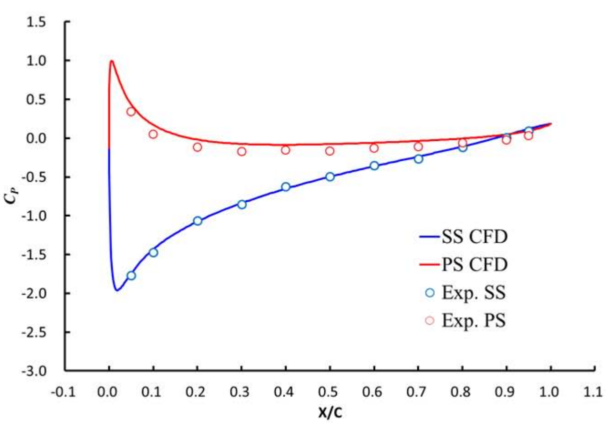

Figure 1(a) Angle of Attack 6 deg

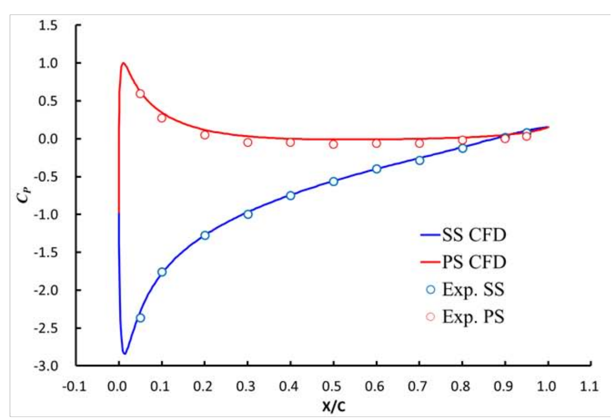

Figure 1 (b) Angle of Attack 8 deg

Figure 1 Static Pressure Distribution without Cavitation

Fig. 2 shows the σ-C_l diagram for angles of attack of 6° and 8°. For 6°, when a laminar boundary layer is considered, the point from which the lift force begins to decrease moves toward the lower side of σ, that is, the direction coming close to the experimental result, compared to the calculation result using the standard cavitation model. However, the trend that the lift force increases and then decrease in the low-σ region is not predicted in the calculation. For 8°, when a laminar boundary layer is considered, the point from which the lift force begins to decrease moves toward the lower side of σ compared to the calculation result using the standard cavitation model. However, the trend that the lift force does not decrease until σ is about 1.2, which is seen in the experimental result, is not predicted. The trend that the lift force decreases as σ decreases is common in two types of calculation results, which is a considerable difference compared to the experimental result.

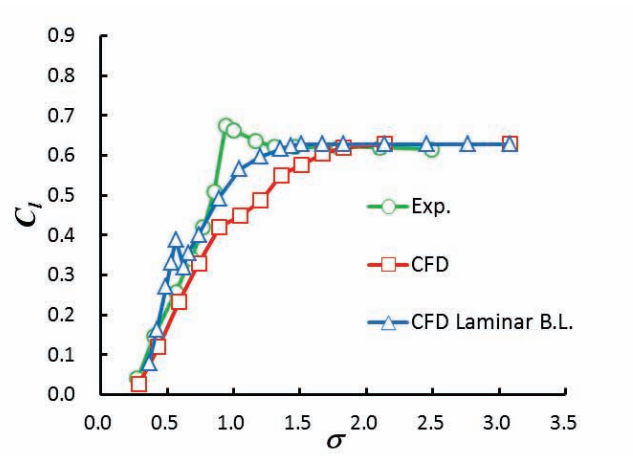

Figure 2(a) Angle of Attack 6 deg

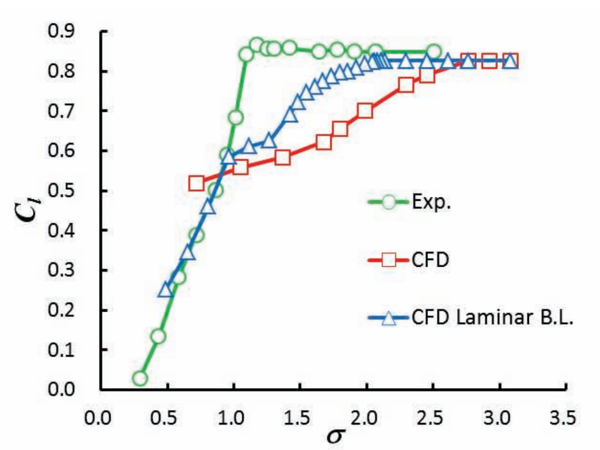

Figure2 (b) Angle of Attack 8 deg

Figure 2 Cavitation Coefficient and Lift Coefficient Characteristics

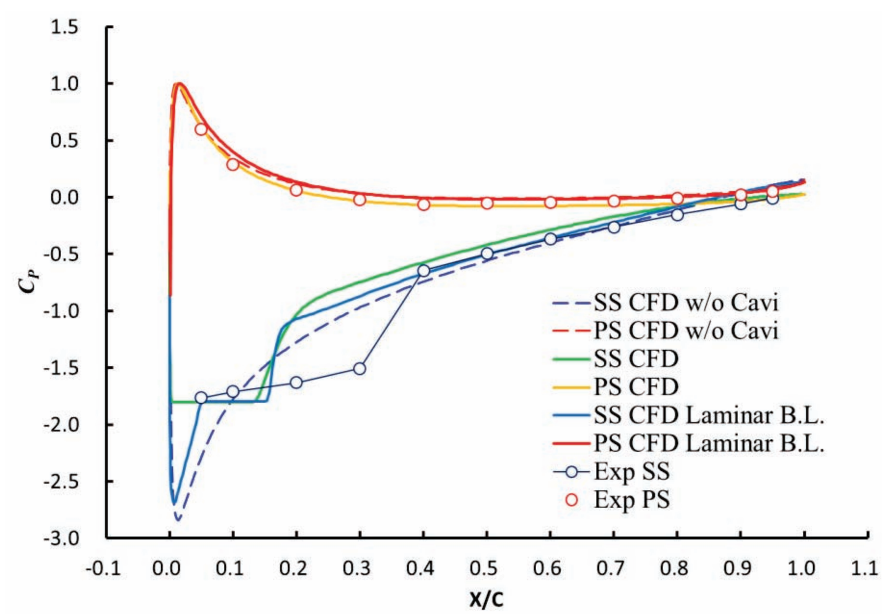

Figure 3 Static Pressure Distribution at σ=1.8

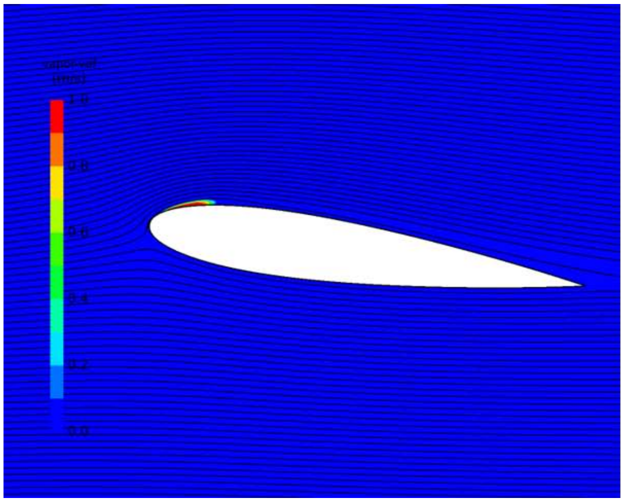

Figure 4(a) Conventional Cavi. Model

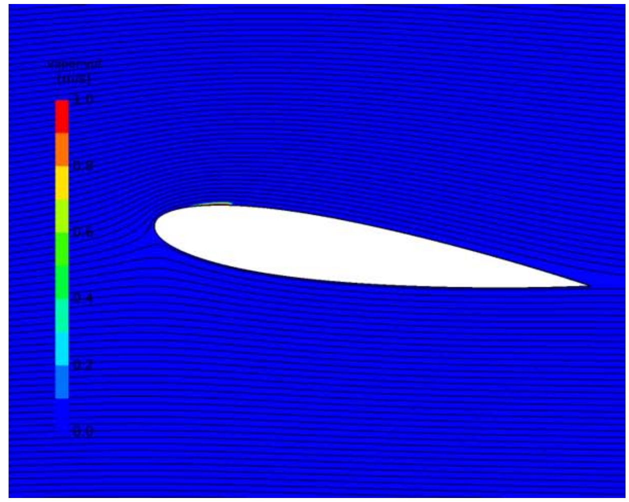

Figure 4(b) Cavi. Model with Laminar B.L.

Figure 4 Stream Lines and Void Fraction Contour at σ=1.8

Fig. 3 shows the comparison of the calculation result and the experimental result for the static pressure distribution on a blade surface for σ = 1.8. Fig. 4 shows the stream lines around the blade and the void fraction distribution. It is seen in Fig. 3 that there is good agreement between the experimental result and the calculation result for the static pressure distribution on the pressure surface even when cavitation exists. When a laminar boundary layer is considered, the peak of the lowest static pressure upstream from the point the laminar boundary layer ends becomes lower than the saturated vapor pressure, but is higher than the peak of negative pressure of the analysis result without cavitation. On the other hand, when the standard cavitation model is used, the saturated vapor pressure is nearly equal to the lower limit of the static pressure. The validity of the pressure being lower than the saturated vapor pressure cannot be verified as there is no static pressure measuring point at the relevant position. In the experiment, it is not easy to obtain reliable data because absolute negative pressure cannot be measured easily and a pressure measuring hole itself may become a cavitation generation source. In Fig. 3, the length of cavitation is predicted to be smaller compared to the experiment. This tendency is also pointed out in a past study.(1) It is also seen in Fig.4 that the position of the front edge of sheet cavitation moves downstream in the analysis considering a laminar boundary layer compared to the standard model. In both results the stream line is separated from the front edge of sheet cavitation.

Conclusion

As a cavitation fluid analysis of the NACA0015 hydrofoil, calculation was conducted considering the nature that cavitation did not occur in a laminar boundary layer. Although some improvement was seen in the calculation result, the prediction accuracy was not sufficient yet. Overall improvement considering cavitation models, turbulent models, computational schemes, analysis grids, three-dimensionality, unsteadiness, etc., is needed.

Acknowledgments

This analysis was made with huge efforts by Mr. Naoki Kodama. We would like to thank his contribution.

References

[1] Kato C., (2011). *"Industry-University Collaborative Project on Numerical Predictions of Cavitating Flows in Hydraulic Machinery -Part 1: Benchmark test on cavitating hydrofoils-"*, Proc. AJK2011-FED,, AJK2011-06084.

[2] Tsuda, Tani and Yamanishi, (2012). *"A Construction of a Multi-Process Cavitation Model for Cold Water (Implementation of Inception/Collapse and Coalescence/Break-up)"* (in Japanese), Transactions of the JSME (B) Vol. 78, No. 790 (2012-6), pp. 1165-1180.

[3] Suzuki and Kato, (2013). "Essential Problems Associated with Numerical Simulation of Cavitating Flows Based on Homogeneous Models" (in Japanese), Transactions of the JSME (B) Vol. 79, No. 808 (2013-12), pp. 2743-2758.

[4] Watanabe S., Tsuru W., Yamamoto Y. and Tsuda S, (2016). *"Cavitation simulation using a new simple homogeneous cavitation model"*, ISROMAC 2016.

[5] Iga, Matsuura and Tateishi, (2016). *"Consideration of Numerical Model of Cavitation in Hydraulic Oil Flow"* (in Japanese), Symposium on Cavitation (18th).

[6] Zwart P. J., Gerber A. G. and Belamri T., (2004). "A Two-Phase Flow Model for Predicting Cavitation Dynamics", Proc. ICMF Fifth International Conference on Multiphase Flow, see the following website:
http://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node343.htm

[7] Chen Y., Heister S. D. (1995). *"Two-phase modeling of cavitated flows"*, Computers and Fluids 24, 7, pp. 799–809.

[8] Huang T. T., Peterson F.B., (1976). *"Influence of Viscous Effects on Model/Full-Scale Cavitation Scaling"*, Journal of Ship Research, Vol. 20, No. 4, pp. 215-223.

[9] Kuiper G., (2010). *"Cavitation in Ship Propulsion"*, Chapter-7, pp. 47-79.
https://ocw.tudelft.nl/wp-content/uploads/Chapter-7_02.pdf

[10] Hiroharu Kato (Editor),(2016). "Shinban Cavitation Kiso To Saishin No Shinpo ; Foundations of Cavitation and Recent Progress (New Edition) (in Japanese), Morikita Publishing, pp. 76-79.