Effect of the winglet on reduction of blade tip vortex from elliptical hydrofoil

S Maeda¹, T Sano¹, M Iino¹, M Farhat², and A Amini²

¹ Mitsubishi Heavy Industries, Ltd., Research & Innovation Center, Hyogo Japan
² Laboratoire de Machines Hydrauliques, Ecole Polytechnique Fédérale de Lausanne, Avenue de Cour 33bis, 1007 Lausanne Switzerland

satoshi78_maeda@mhi.co.jp

Abstract. It is well known that the performance of the turbomachinery (losses, vibrations, noise, and erosion) is often affected by the trailing vortex from the blade tip (Tip Leakage Vortex; TLV) of the impeller or propeller, and sometimes cavitation occurs in the center of the vortex core (Tip Vortex Cavitation; TVC). Many researches on the vortex characteristics of isolated hydrofoil by numerical simulations or experiments have been done so far, however, the research on understanding the mechanism of the vortex control technique is not enough, and it is important to mitigate the effect of TLV or TVC, to enhance the performance of turbomachinery. In this study, numerical simulation and experimental investigation were carried out to clarify the effect of winglet, attached to the elliptical hydrofoil (NACA16020). Five different winglet patterns were tested, changing bending angle and direction. Experiments were carried out using the high-speed cavitation tunnel in Ecole Polytechnique Fédérale de Lausanne (EPFL). As for numerical simulation, it was carried out in the same condition of the experiment. As a result, it was clarified that TVC decreased as increasing the bending angle of winglet, and disappeared in the case of 90 degree bent toward pressure side. And it is clarified that the vortex strength, in other word, vortex core pressure is affected by the interaction of the leading edge vortex and the tip vortex, and in case of pressure side 90 degree bended winglet, both vortices were mutually affected and cancelled the strength of these vortices.

1. Introduction

In hydraulic machinery, such as pumps, water turbines, and marine propellers, tip vortex cavitation (TVC) and tip leakage vortex (TLV) are critical problems and cause noise, vibration, and erosion [1]. It is important, therefore, to mitigate the effect of TVC or TLV, to enhance the performance of the turbomachinery. For many years, many researches to mitigate TVC and TLV have been done. Hongtao et al. [2] shows that winglets installed on marine propeller are mitigated the volume of cavitation, however, the real mechanism is still not clear. Amini et al.[3] shows that when the blade tip of the hydrofoil is bent toward the pressure side, TVC is suppressed and the pressure of the vortex core is larger than the blade tip is not bent, however, it is still not fully understood the accurate mechanism of the reduction of TVC, either.
In addition, it is still difficult to evaluate a method of mitigating TVC and TLV using CFD analysis. Reclari et al. [4] shows that in order to solve tip vortex by CFD analysis, it is not necessary to refine the resolution of the analysis grid in the blade wake region and solve it diligently, but it is essential to capture the separation vortex accurately from the leading edge of the blade.

This paper focuses on the effect of winglet at the blade tip on mitigation of tip vortex obtained from experimental results, and considers the mechanism using CFD analysis. Calculations are performed using the mesh creation method [4]: refinement of the resolution of the area near blade tip surface. Comparing the experimental results [3] with CFD analysis, the generation and control mechanism of tip vortex in both hydrofoils with and without winglet are considered from CFD analysis.

2. Research method
2.1 Experimental setup
2.1.1 The high speed cavitation tunnel

The experiments are carried out in EPFL high-speed cavitation tunnel [5]. The test facility is a closed loop equipped with a squared test section (150x150x750 mm) and a circulating pump. Upstream to the test section, a honeycomb along with a 46:1 converged nozzle is used to maintain the turbulence intensity below 1%. The operating flow parameters are the hydrofoil incidence angle $\alpha$, freestream velocity at the test section inlet $U_\infty$, and the cavitation coefficient $\sigma$ shown as the following equation (1):

$$\sigma = \frac{p_\infty - p_{vap}}{\frac{1}{2} \rho U_\infty^2}$$

Where $p_\infty$ is the pressure at the inlet of the test section, $p_{vap}$ is the vapor pressure of the water at the working temperature and $\rho$ is the water density. $U_\infty$ is derived from pressure difference between both ends of the convergent.

2.1.2 Hydrofoils

In this study, the elliptical hydrofoil has been selected, which has NACA16-020 cross section with a chord length of 60 mm at the root, decreasing elliptically to 0 mm through 90 mm span, as shown Figure 1. This hydrofoil has been defined as the Baseline and bent its tip area to get hydrofoils listed in Table 1.

Figure 1. Sketch of the elliptical NACA 16-020 hydrofoil
Table 1. The list of hydrofoils

| Designation | Description | Drawing |
|-------------|-------------|---------|
| Baseline    | Baseline geometry (NACA16-020) | ![Drawing](Baseline.jpg) |
| 45° Upward  | 10% of the chord length bent to 45° suction side | ![Drawing](45°Upward.jpg) |
| 45° Downward| 10% of the chord length bent to 45° pressure side | ![Drawing](45°Downward.jpg) |
| 90° Upward  | 10% of the chord length bent to 90° suction side | ![Drawing](90°Upward.jpg) |
| 90° Downward| 10% of the chord length bent to 90° pressure side | ![Drawing](90°Downward.jpg) |
| Double 45°  | 10% of the chord length bent to 45° downward & upward | ![Drawing](Double45°.jpg) |

2.1.3 Measurements

The measurement of the hydrodynamic forces on the hydrofoil is performed by a five component load cell. The load cell consists of a deformable H-beam on which the hydrofoil is mounted. The deformation of the H-beam is measured by several strain gauges forming five full bridge circuits. The maximum measurable lift force is 104 N and the precision is 1.5 N and 0.5 N for the lift and drag, respectively [6].

Also, flow visualization [3] using photography and a high-speed camera, were performed to investigate the systematic observation of TVC produced by different shapes of winglets to evaluate the benefit of such a device.

2.2 Computational setup

The computation is done in a rectangular channel, as shown Figure 2.a. Inlet and outlet boundary conditions are constant total pressure and constant velocity, respectively. Inlet turbulence intensity is 1%. SST k-ω is used as turbulence model. The working fluid is water, and single-phase computation without any cavitation models were performed. Cavitation and tip vortex domains are assessed by focusing on the isosurface of vapor pressure and Q criterion. The CFD analysis is performed only in steady state with ANSYS CFX 19.3. The mesh is made of an unstructured grid, and 25 boundary layer meshes are inserted on the blade surface.
Especially, the blade tip mesh is sized so that the minimum $Y_+$ is below 1, and the other region are sized at the rate shown in Figure 2.b.

![Figure 2. a. Dimension of the computational domain. b. The surface mesh on the baseline hydrofoil, the numbers in this figure show the dimensionless number obtained by $Y_+$ (blade tip mesh size is $Y_+=$1).](image)

3. Results

3.1 Validation of CFD analysis

In this section, the validity of the CFD analysis is evaluated by comparing the experimental results with the calculation results. First, the relationship between lift and drag coefficients on the incidence angle of blade is compared. Figure 3 shows the results of the experiment in which the incidence angle was changed from $-25^\circ$ to $+25^\circ$ when the upstream velocity is 10 m/s and non-cavitating condition, and the CFD analysis in which the incidence angle are 10°, 12°, and 14° under the same conditions as the experiment. Lift and drag coefficients are defined by the following equations (2):

$$
C_L = \frac{L}{\frac{1}{2} \rho U_\infty^2 S_{ref}} \quad C_D = \frac{D}{\frac{1}{2} \rho U_\infty^2 S_{ref}}
$$

(2)

Where $L$ and $D$ are the lift and drag forces and $S_{ref}$ is a reference surface defined as the projected area of each hydrofoil.
The CFD results are in good agreement with the experimental results, but there is a tendency to underestimate the drag coefficient at positive incidence. This reason is under investigation. In addition, in both experiments and CFD analysis, there is almost no difference in lift and drag coefficient due to the difference in winglet shape. This shows that the performance of the blade does not change even if the winglet is installed.

Moreover, the aspect of TVC is compared. In order to visualize TVC region in single-phase flow CFD analysis, the region where Q-criterion [7], which is the second invariant of the velocity gradient tensor, and the saturated vapor pressure are shown on the isosurface. After that, the isosurface is colored by the velocity helicity [8], which is an index indicating the vortex strength. Figure 4 shows the comparison of the results with those of the experiments. As a result, both the experimental and CFD results show that the length of the vortex decreases as the bending angle of winglet increases. In particular, TVC is significantly suppressed in 90° Downward hydrofoil bent toward the pressure side. In this way, it has been confirmed that almost the same tendency as the experimental results are obtained by CFD results.

![Figure 3. Lift coefficient (left) and drag coefficient (right) vs incidence angle, upstream velocity: 10m/s.](image-url)
3.2 The behavior of the blade tip vortex

3.2.1 Origin of the tip vortex

In this section, focusing on the generation process of the tip vortex, especially Baseline and 90° Downward, which are especially different depending on the presence of the winglets, are compared and considered. First, the streamlines passing through the region where the distance from the fine part of the blade tip mesh is within 1 mm are noted. The streamlines are colored by flow velocity and vorticity, and the streamlines on the blade surface are also showed by thin black lines, as shown in Figure 5.

**Figure 4.** The pictures during the experiments and visualization (The isosurface of Q-criterion and vapor pressure colored by velocity helicity) of TVC region by CFD (Incidence angle=12° and Flow velocity=10m/s).
Figure 5. **a.** Streamlines passing through the region near the fine part of the blade tip mesh colored by normalized flow velocity, which is divided by the main stream velocity and surface streamlines on the blade (Left : Baseline, Right : 90°Downward) **b.** Streamlines colored by vorticity and others are the same as 6.a (Incidence angle=12° and Flow velocity=10m/s).

Since the incidence angle is large, flow separation at leading edge occurs. Because of this, separation vortex from the pressure side to the suction side is rolled up to the origin of the tip vortex. It has been considered that strong TVC is generated by the synthesis of the leading-edge vortex and the tip vortex generated by the leakage of the circulation of the hydrofoil in Baseline. On the other hand, in 90° Downward, although the separation vortex from the leading edge had almost no effect, because the winglet bent at 90 degrees does not change incidence regardless of the angle of attack, it is considered that the leakage vortex is less likely to flow into the suction side due to the winglet shape, and the tip vortex is mitigated. Figure 6 shows surface streamlines on the hydrofoil and isosurface of Q-criterion at the same time. Focusing on the case of 90° Downward, it can be seen that the flow into the leading edge of the winglet is pulled up toward the trailing edge to the root of the winglet. In addition, since the vortex region can be confirmed even at the point where the pulled-up streamlines flow out, it is presumed that a part of the vortex region, which is originally supposed to be the tip vortex when without winglet, is discharged as this vortex region. In order to consider the phenomenon, Figure 7 shows the secondary flow vector (project velocity vector tangential to a section) and z-direction (page upward) flow velocity contour in the streamwise cross section just upstream of the origin of the tip vortex. In Baseline, the flow from the suction
surface to the pressure surface can be observed near the blade tip, and it can be said that this flow contributes to the generation of the tip vortices. On the other hand, in 90° Downward, the flow is bent in the direction from the blade tip of the leading edge to the winglet root of the trailing edge. This is why the streamline shown in Figure 6 is pulled up. The reason why this secondary flow occurs is still under investigation. This phenomenon cannot be confirmed in 45° Downward, it has been considered that the shape of the curved surface of the winglet has an optimum shape.

Figure 6. Surface streamlines on the Baseline (left), 45° Downward (center), and 90° Downward (right) and isosurface of Q-criterion (Incidence angle=12° and Flow velocity=10m/s).

Figure 7. Velocity vector projected in the tangential direction of the streamwise cross section (as shown the green plane) and z-direction (page upward) flow velocity, which is divided by the main stream velocity, contour in Baseline (left) and 90° Downward (right) (Incidence angle=12° and Flow velocity=10m/s).

3.2.2 The tip vortex growth

In this section, the development and damping of the tip vortex is discussed. First, the flow velocity contour in the spanwise cross section of the tip of Baseline and 90° Downward are shown in Figure 8 (The light blue region of the blade tip is the isosurface of the Q-criterion). In Baseline, since the tip vortex tube
flows along the main flow, the mainstream region exists in the area between the vortex tube and the boundary layer when the vortex tube is separated from the wall surface of the blade tip. The vortex tube is stretched due to this effect, and the vorticity grows. This is considered to be the effect of the extension term (right-hand first term in Equation (3)) of the transport equation of vorticity. [8]

\[
\frac{D\vec{\omega}}{Dt} = (\vec{\omega} \cdot \nabla)\vec{U} + \nu \nabla^2 \vec{\omega} + \nabla \times \vec{f}
\]  

(3)

Where \( \omega \) is vorticity, \( U \) is velocity, \( \nu \) is kinematic viscosity, and \( f \) is external force.

In addition, focusing on the flow just after the trailing edge of the blade, Figure 9 shows the flow velocity contour on the surface and the isosurface of the streamline and Q-criterion passing near the blade tip. In Baseline, it can be confirmed that the flow velocity decreases in the vortex core region. Also, it is found that the region around the vortex core is the mainstream region. It can be considered that the main flow around the vortex core causes the tip vortex to be stretched under the influence of the extension term of the transport equation of vorticity, and prevents the damping of the vorticity. On the other hand, in 90° Downward, since the vortex core is affected by the wake because it extends along the extension line of the wake, it is considered that the vortex core is less likely to be extended and thus less likely to grow.

**Figure 8.** The normalized flow velocity, which is divided by the main stream velocity, contour in the spanwise cross section of the tip of Baseline (left) and 90° Downward (right), and the light blue region of the blade tip is the isosurface of the Q-criterion.

**Figure 9.** Visualization of tip vortex regions and normalized flow velocity, which is divided by the mainstream velocity, contour on the streamwise cross section just after the trailing edge of the blade in Baseline (left) and 90° Downward (right).
4. Conclusion

In this paper, CFD analysis has been used to investigate the reduction mechanism of blade tip vortex. The findings are shown below.

1. CFD analysis using the mesh refined on the blade tip has been carried out, and it has been confirmed that TVC is significantly mitigated in 90° Downward as the experimental results.

2. It was clarified that in 90° Downward, leakage vortex from the leading edge is less likely to flow from the pressure surface to the suction surface due to the winglet, and blade tip vortices are less likely to occur.

3. It was clarified that secondary flows are generated from the winglet tip to the winglet root, and the streamline is bent, so that the blade tip vortices are suppressed by the amount of the vortex that is discharged from the area where the pulled-up streamlines flow out. The generation mechanism of the secondary flow will be continuously investigated in the future.

4. It was clarified that after the tip vortex was generated in 90° Downward, the influence of the mainstream around the vortex tube makes it difficult to receive the effect of stretching the vortex core, and thus the vortex tube also became difficult to grow.

References

[1] Brennen, “Cavitation and Bubble Dynamics”, Oxford Engineering Science series 44, pp.215-231, 1995.

[2] G Hongtao et al., "Effect of various winglets on the performance of marine propeller", Applied Ocean Research, 2019.

[3] A Amini, M Reclari, T Sano, M Iino, and M Farfat, “Suppressing tip vortex cavitation by winglets”, Experiments in Fluids, 2019.

[4] M Reclari, T Sano, M Iino, M Drayer, A Amini, and M Farhat, “Roll-up region of tip vortex: numerical investigation”, in 29th IAHR Symposium on Hydraulic Machinery and Systems, Kyoto, 2018.

[5] M. Dreyer, J. Decaix, C. Münch-Alligné and M. Farhat, “Mind the gap: a new insight into the tip leakage vortex using stereo-PIV,” Experiments in Fluids, vol. 55, no. 11, pp. 1-13, 2014.

[6] Dupont P, “Etude de la dynamique d’une poche de cavitation partielle en vue de la prédiction de l’érosion dans les turbomachines hydrauliques”, EPFL/LMH, 1993.

[7] V Kolar and J Sistek, “Corotational and Compressibility Aspects Leading to a Modification of the Vortex-Identification Q-Criterion”, AIAA Journal, Vol. 53, No. 8, 2015

[8] S Kida and S Yanase, “Turbulent Dynamics”, Asakura Publishing Co., Ltd., pp. 13-18, 311-315, 1999.