Investigation on internal flow of draft tube at overload condition in low specific speed Francis turbine

Yuta Tamura$^1$ and Kiyohito Tani$^1$

$^1$Hitachi Mitsubishi Hydro Co., 2-1, Saiwaicho, 3-Chome, Hitachi 317-0073, Japan

tamura.yuta.ws@hm-hydro.com

Abstract. The cavitating vortices causes the unsteady phenomena like the pressure fluctuation, the noise and the vibration in the draft tube at the overload condition which is the far operating point from the design point. Because the full load was normally near the design point, there were few troubles due to cavitating vortices at the full load. Today, however, the design point is sometimes set to lower load to achieve the high efficiency from the partial load to the full load in low specific speed Francis turbines, which have good performance to a change in a discharge. Then, the full load is relatively further from the design point. As the result, the potential for the cavitating vortices at the full load is increased. To control of the unsteady phenomena at the full load, the study focused on the cavitating vortices at the overload condition is important. This paper presents the unsteady behavior of the cavitating vortices at the overload condition with the scaled model of specific speed $N_{Qe}=0.083$. On the experimental approach, the pressure pulsation in the upper draft tube was measured and the unsteady behavior of cavitating vortices was taken movies with a high speed camera. On the numerical approach, Computational Fluid Dynamics (CFD) adopting a two-phase unsteady analysis was carried out. The pressure fluctuation and the velocity distribution of two runners, an original and a newly designed, were compared.

1. Introduction

Hydraulic turbines with good responsiveness to the power demand are adopted at broad operating range. Especially the low specific speed Francis turbines, which have the better performance to a change in a discharge than the high specific speed Francis turbines, are demanded to operate with high efficiency and high reliability at whole power range in high head plants. One of the causes of the unsteady phenomena like pressure fluctuation, power house vibration and noise is the unsteady behavior of the cavitating vortices at the far operating point from the design point. Because passing the partial load is not to be avoided to reach the full load condition from a start, many researches about the cavitating vortices causing the unsteady phenomena in the draft tube at the partial load have been carried out. On the other hand, the full load was near the design point because of the demand of the high efficiency, so there were few troubles due to the cavitating vortices at the full load. Today, however, the design point is sometimes set at lower load to achieve the high efficiency from the partial load to the full load. Then the full load is relatively further from the design point. As the result, the potential for the occurrence of the unsteady phenomena on the full load is increased. To achieve the demand which is the high efficiency and high reliability from the partial load to the full load, it is important to investigate the unsteady phenomena on the overload condition.
As for the overload cavitating vortices, there are only a few reports that have been presented. Koutnik et al. [1] represented the effect of the cavitating vortices under the overload conditions by a simplified one-dimensional mathematical model. They and Alligne et al. [2] successfully simulated the unstable oscillation propagating in the full circuit of the hydraulic power plant with the use of an appropriate value as the parameter of above mentioned mathematical model. Recently, Chen et al. [3, 4] tried to clarify the cause of the unstable oscillation. They discovered the swirl flow from the runner contributes the stabilization of flow field under the overload conditions. The main cause of instability was pointed out to be the diffuser effect in their literatures.

These pieces of research are helpful in understanding the general characteristics of flow in a draft tube. However, to design hydraulic turbines evaluating the internal flow in accordance with the shape of the runner and the draft tube is important. Therefore, detailed computational fluid dynamics (CFD) simulations will be helpful to understand the internal flow and formulate strategies. Furthermore, what is difficult to measure on model test can be measured by CFD simulation. Mossinger et al. [5] estimated the overload condition of the prototype sized Francis turbine by using the CFD simulation and one dimensional approach. To investigate the unsteady phenomena at the overload condition, our research group [6, 7] carried out the unsteady CFD simulation considered the influence of the cavitation, and compared with the model test. As the results, the CFD simulation results showed the good agreement with the model test results on both the partial load and the overload.

As the next step, it is important to evaluate pressure fluctuation of any runner relationally. In the project of the runner replacement that aimed the decrease of the unsteady phenomena caused by the cavitating vortices at the draft tube, the newly designed runner was estimated by the same way as the original runner [8]. In the present work focused on the overload condition, the cavitating vortices and the pressure fluctuation caused on those of both original and new runners were compared by the CFD simulation and the model test. Furthermore, the volume changes of the cavitating vortices on both two runners were compared by the CFD simulation results.

### 2. Specification of hydraulic plant and way of model test and simulation

A Francis turbine scaled model of specific speed $N_{QE}=0.083$ was an investigation object. The specification was shown in table 1 and the sectional view of model turbine was shown in figure 1. The turbine contained 18 guide vanes, a runner and a bent draft tube. The monitoring points of the pressure fluctuation were placed on the upper draft tube. The original runner was composed with 15 blades and outlet diameter was 318.1mm. The new runner was a type of splitter runner composed with 13 long blades and 13 short blades, and the outlet diameter was 300.1mm. The purpose of designing the new runner was the improvement of the efficiency and the reduction of the pressure fluctuation at the both of the full and partial load. The upper draft tube was made of the clear acrylic glass, so the unsteady behavior of cavitating vortices at the Thoma number corresponding to the suction height of the prototype power plant could be monitored by using a high speed camera.

| Table 1. Specification of model. |
|-----------------------------------|

| Reference Diameter | D (mm) | 318.1 | 300.1 |
|--------------------|--------|-------|-------|
| Speed Factor $n_{ED}$ | (-) | 0.2034 |
| Discharge Factor $Q_{ED}$ | (-) | 0.1650 |
| Specific Speed $N_{QE}$ | (-) | 0.083 |
| Thoma number $\sigma$ | (-) | 0.0445 |
The simulations were carried out with the use of the commercial CFD software ANSYS CFX 16.1. The effects of turbulence were taken into account by using the shear stress transport (SST) turbulence model. The cavitating vortices at the draft tube depended on the velocity distribution from runner outlet, and the internal flow of runner was influenced by the guide vane outlet flow. So, the simulation contained all guide vanes, all runner blades, and a draft tube in the computational domain in figure 2. The grid for the runner part rotated for the duration of the simulation. The boundaries between the rotating part and stationary parts were connected by using a general grid interface (GGI). The evaporation and condensation process was described by using a simplified Rayleigh-Plesset equation. A two-phase homogeneous model was adopted to calculate the mixture of gas and liquid phases. In the present research, the uncertainties caused by the numerical models were evaluated by the mutual confirmation between the numerical and experimental results. To control the Thoma number, the draft tube outlet boundary in the simulation was given by the constant static pressure, and to investigate discharge fluctuation as time progress the guide vane inlet boundary was given by the constant total pressure.

3. Turbine performance and pressure fluctuation on model test
The turbine efficiency and the pressure fluctuation of the original and the new runner on the model test were shown in figure 3. Here, efficiency was normalized by the best efficiency of the original runner, pressure fluctuation was normalized by the net head of the model test and discharge was normalized by the discharge at the best efficiency point of the original runner.

The efficiency of the new runner was higher than the original one on the whole measuring points. Especially, the efficiency of the new runner was about 1% higher at $Q/Q_{BEP}=1.0$ and about 4% higher at $Q/Q_{BEP}=0.6$ than the original one. Regarding the pressure fluctuation, the value of new runner was half of that of the original one at $Q/Q_{BEP}=1.2$. And furthermore, the maximum pressure fluctuation at the partial load of the new runner was over 1% smaller than that of the original, and the discharge of the maximum pressure of the new runner became small compared by the original.
4. Comparison of pressure fluctuation between CFD and model test

The comparison of the pressure fluctuation between the CFD simulation and the model test at \( Q/Q_{BEP} = 1.2 \), which is the overload condition were shown in figure 4. Here, the pressure fluctuation was defined as the pulsation of the static pressure as advance of time at the monitoring points in the upper draft tube showed in figure 1. Pressure in both the CFD simulation and the model test were normalized by the net head of the model test. Time \( t_n \) was normalized by using the rotational speed of runner \( n \) as follows

\[
    t_n = \frac{t \cdot n}{60}
\]

This normalization converted the times or periods into equivalent numbers to the runner rotation.

The pressure fluctuation of the original runner in the CFD simulation was described the very low-frequency (\( t_n = 10 \text{~to~} 20 \)) and large amplitude and was similar to that of the model test. Besides, regarding the new runner, the pressure fluctuation in the CFD simulation also seemed to agree with that in that of the model test, that is, there are not big oscillation obtained in the original runner and there were small and short oscillation. In the model test of both runners, the peak of the maximum value of short oscillation was bigger than that of the CFD simulation.

5. Behaviour of cavitating vortices on overload condition

The comparison of the behaviour of the cavitating vortices at \( Q/Q_{BEP} = 1.2 \) of the both runners between the model test and the CFD simulation were shown in figure 5. Here, 4 representative moments were focused on. The cavity shapes of the CFD simulation were visualized by using an isosurface of the constant vapor volume fraction \( \gamma = 0.5 \). And the photographs of the cavity in the model test were obtained by the using high speed camera. The circumferential velocity \( (V_r) \) distribution at the runner outlet, was also shown in figure 5. High velocity against the runner rotational direction was evaluated by the positive value. To compare the connection of the cavity shape with the pressure fluctuation, the pressure fluctuations of the both runners on the CFD simulation shown in figure 4 were also plotted in figure 5.
The cavitating vortices of the original runner at $t_n=25.0$ formed bell-shape. At the same time, the pressure was low. The cavity was shrinking to upstream while the pressure was increasing ($t_n=28.3$~$31.7$). And at the minimum cavity size, the highest pressure was indicated ($t_n=31.7$). Then, the cavity was expanding to downstream while the pressure was decreasing. The cavity of the model test also formed bell shape at $t_n=25.0$, and it was similar that the cavity was shrinking while the pressure was increasing and expanding while the pressure was decreasing.

On the other hand, the cavity shape of the new runner in the CFD simulation was also bell shaped at $t_n=25.0$, but the cavity looked shorter in the flow direction than the original. And then, the cavity size was constant as the time progress. Therefore, the pressure of the new runner did not almost change. The behavior of the cavity on the model test was similar to that of the CFD simulation.

With respect to the velocity distributions, at the outlets of both the runners, the direction of the circumferential flow was against runner rotation and additionally there were two differences observed between them.

First, the velocity distribution of the original runner changed periodically. Especially the high velocity region at the area in middle of outer circle and center point spread during $t_n=28.3$~$33.3$. At the same time, the cavity was shrinking and elongating and then pressure fluctuated drastically. On the other hand, the velocity of the new runner was almost stable compared with that of the original runner.

Second, in the original runner the sectional distribution of the circumferential velocity was asymmetry at $t_n=31.7$, the cavity volume was minimum and short. The bottom surface of the cavity was so wavy that these changes of the velocity distribution might correlate with cavitating vortices. On the other hand, on the new runner the velocity distribution was nearly axisymmetric at all the time.
Figure 5. Behavior of cavitating vortices at $Q/Q_{BEP} = 1.2$. 

Original Runner

New Runner
The averaged axial (meridian) and circumferential velocity profiles at the runner outlet at the 4 moments which were same as figure 5 were shown in figure 6. Here, the radius was normalized by the outlet diameter of the new runner and velocities were normalized by using the peripheral velocity at the new runner outlet (the section was the same as $V_t$ section shown in figure 5). Positive value of circumferential velocity meant the opposite direction against the runner rotation, and negative value of axial velocity meant same direction as the main flow.

On the new runner, the circumferential velocity at $R^*=0.28$~0.52 was lower in spite of the axial velocity which was higher than the original. Because of the smaller circumferential flow, the cavity at the upper draft tube did not develop as large as the original runner. As the result, the pressure fluctuation of the new runner decreased compared with that of the original one. In addition, the efficiency of the new runner at the overload condition was higher than that of the original runner because of the swirl loss caused by the swirl flow at the draft tube was decreased.

Both circumferential and axial velocity profiles changed as the time got progressed on the original runner were larger than those on the new runner. Especially, those changes on $R^*=0$~0.3 were remarkable. On the remarkable area which was the middle of the flow passage, a part of the axial velocity showed a positive value (i.e. reverse flow was occurring) and the reverse flow changed as time progressed.

![Figure 6. Runner outlet velocity profile at Q/Q_{BEP}=1.2.](image)

To examine the correlation between the reverse flow and the cavity elongating and shrinking at the upper draft tube, the changes of both the cavity volume, flow rate of the reverse flow and main flow with respect to time were plotted and shown in figure 7. In this work, the flow rate of the reverse flow $Q_r$ was the value integrated by using the positive value of $V_m^*$ shown in figure 6 and the flow rate of the main flow $Q_m$ was the value integrated by using the negative value of $V_m^*$. These values were defined as the proportions to the flow rate $Q_{ave}$ which was the value integrated by using $V_m^*$ at all region and averaged all the time. The cavity volume $V_C$ was defined as follows.

$$V_C = \sum (\gamma \cdot V_m)$$  \hspace{1cm} (2)

Here, $\gamma$ was the vapor volume fraction of each mesh, and $V_m$ was the volume of the mesh. The cavity volume was normalized by the maximum value of the cavity volume of the original runner.
The volume of the cavitating vortices of the original runner was frequently changed as the time progress. At the moment of the minimum cavity volume, the volume reduced less than 10% of the maximum volume. The frequency of the cavity volume fluctuation was about 15 times runner rotation and was same as that of the pressure fluctuation seen in figure 5. On the other hand, the volume of the new runner showed little changed. Besides, the volume of the new runner was half of that of the original runner at the moment of the maximum cavity volume.

In the original runner, at \( t_n = 25.0 \), the low pressure at the upper draft tube because of the larger cavity and the high pressure at the draft tube due to the diffuser and bent effect formed the adverse pressure gradient at the draft tube. The pressure gradient caused the reverse flow. While \( t_n = 25.0 \sim 31.7 \), the cavity was shrinking because the main flow rate decreased. As the cavity volume became small and then the pressure at the upper draft tube became high, the adverse pressure gradient might weaken and the reverse flow decreased. As the result, the main flow rate increased. While \( t_n = 31.7 \sim 33.3 \), the main flow rate increased then the cavity volume increased. While the cavity volume was increasing, the reverse flow was increasing again. On the original runner, this cycle was repeated.

In the case of the new runner, the reverse flow rate was lower than the maximum value of the original runner because the circumferential velocity was low. And the cavity volume also did not develop as large as that of the original. So the cavity volume did not change.

![Graphs showing cavity volume, reverse flow, and main flow](image-url)

**Figure 7.** Flow rate and volume of cavity at \( Q/Q_{BEP} = 1.2 \).
6. Conclusion
In this work focused on the cavitating vortices at the overload condition, the cavitating vortices and the pressure fluctuation caused on those of both the original and the new runner were compared with by the CFD simulation and the model test. Furthermore, the volume changes of the cavitating vortices of both the two runners were compared with by the CFD simulation results.

As a result, the behavior of the cavitating vortices and pressure fluctuation seen in the CFD simulation was similar to those of the model test. That is, the cavity of the original runner was shrinking when the pressure was increasing and expanding while the pressure was decreasing. The cavity volume change is interrelated with the reverse flow occurred at the upper draft tube. On the other hand, the pressure of the new runner did not change much because of the constant cavity volume with respect to time. As the time progressed, the change in cavity volume was little when compared with original runner. Furthermore, because the circumferential velocity and swirl loss at the new runner outlet was lower than the original, the efficiency of the new runner at the overload condition is higher than that of the original runner.

In the next steps, the authors intend to compare the velocity distribution between the model test and the CFD simulation at not only the overload condition but also at the best efficiency point and the partial load condition.

Acknowledgements
To evaluate the performance of the original and new runners in this research, Shikoku Electric Power Company, Incorporated contributed evaluation data to us. We are deeply grateful to them.

References
[1] Koutnik, J., Nicolet, C., Schohl, G. A. and Avellan, F., 2006, “Overload Surge Event in a Pumped-Storage Power Plant", Proc. 23rd IAHR Symp. Hydraulic Machinery & Systems, Yokohama, Japan, Paper No. F135.
[2] Alligné, S., Maruzewski, P., Dinh, T., Wang, B., Fedorov, A., Iosfin, J. and Avellan, F., 2010, “Prediction of a Francis Turbine Prototype Full Load Instability from Investigations on the Reduced Scale Model", IOP Conf. Ser.: Earth Environ. Sci., 12, 012025.
[3] Chen, C., Nicolet, C., Yonezawa, K., Farhat, M., Avellan, F. and Tsujimoto, Y., 2008, “One-Dimensional Analysis of Full Load Draft Tube Surge", ASME J. Fluids Eng., 130, pp.041106-1 – 10.
[4] Chen, C., Nicolet, C., Yonezawa, K., Farhat, M., Avellan, F., Miyagawa, K. and Tsujimoto, Y., 2010, “Experimental Study and Numerical Simulation of Cavity Oscillation in a Conical Diffuser", Int. J. Fluid Machinery & Systems, 3, No. 1, pp.91 – 101.
[5] Mossinger, P., Conrad, P. and Jung, A., 2014, "Transient Two-Phase CFD Simulation of Overload Pressure Pulsation in a Prototype Sized Francis Turbine Considering the Waterway Dynamics ", Proc. 27th IAHR Symp. Hydraulic Machinery & Systems, Montreal, Canada, Paper No. 7.2.3.
[6] Shingai, K., Okamoto, N., Tamura, Y. and Tani, K., 2014, "Long-Period Pressure Pulsation Estimated in Numerical Simulations for Excessive Flow Rate Condition of Francis Turbine", Trans. ASME, J. Fluids Eng., Vol.136, p.071105.
[7] Tamura, Y., Okamoto, N., and Tani, K., 2014, "Experimental and Numerical Investigation of Unsteady Behavior of Cavitating Vortices in the Draft Tube of Low Specific Speed Francis Turbine", Proc. 27th IAHR Symp. Hydraulic Machinery & Systems, Montreal, Canada, Paper No. 2.2.6.
[8] Tamura, Y., Tani, K. and Ootani, A., 2015, "Experimental and Numerical Study on Expansion of
Steady Operating Range of Low Specific Speed Francis Turbine", Proc. 13th AICFM Symp.,
Tokyo, Japan, Paper No.AICFM13-123.