Experimental and numerical investigation of unsteady behavior of cavitating vortices in draft tube of low specific speed Francis turbine

Y Tamura¹, K Tani¹ and N Okamoto²
¹Hitachi Mitsubishi Hydro Co., Japan
²Shikoku Electric Power Co., Inc., Japan
tamura.yuta.ws@hm-hydro.com

Abstract. At both partial and full load of Francis turbines, the unsteady behavior of cavitating draft tube vortices occurs and leads to undesirable matters such as power house vibration, noise and power swing in some cases. This paper presents the investigation of the interaction between the flow pattern at runner outlet and the unsteady behavior of cavitating vortices in draft tube with experimental and numerical approaches. On the experimental research, the pressure pulsation in the draft tube is measured and the unsteady behavior of cavitating vortices is taken pictures with a high speed camera in the model test. On the numerical research, by Computational Fluid Dynamics (CFD) adopting a two-phase unsteady analysis, the analysis domain from the guide vane to the draft tube is carried out for investigating the interaction between the runner outlet flow and the vortex pattern. The pressure pulsation at the upper draft tube and the unsteady behavior of cavitating vortices obtained from CFD results are similar to those obtained in the model test. Detailed analysis of CFD results at overload indicates the repeat of expansion and contraction of cavitating vortices, which were shaped helical vortices with opposite direction of runner rotation, and the corresponding flow pattern in every time step of the pressure pulsations.

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 high efficiency and high reliability from the partial load to the full load in high head plants. Generally the runner outlet flow at the design point has little circumferential component. One of the causes of the pressure fluctuation, the power house vibration and the noise is the unsteady behavior of the cavitating vortices which occur due to the circumferential flow after runner outlet 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 circumferential flow after runner outlet was smaller and 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 at the partial load. Then the full
load is relatively further from the design point. As the result, the potential for the occurrence of the unsteady phenomena, which is different from that of the partial load, is increased by the increased circumferential flow after the runner outlet at the full load. To achieve both high efficiency at the partial load and control of the unsteady phenomena at the full load, the study focused on overload condition, which may cause the unsteady phenomena more notably than full load, is important. As for the overload cavitating vortices, there are only a few reports that have been presented. Koutnik et al. represented the effect of the cavitating vortices under the overload conditions by a simplified one-dimensional mathematical model [1]. They and Alligné 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. tried to clarify the cause of the unstable oscillation [3, 4]. 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 inner 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 inner flow and develop strategies. Our research group had found uniquely pulsating structures by using the CFD simulation and the model test in the previous study [5]. In the present work, a series of simulation was performed and model test was carried out. To confirm the accuracy of simulation, the characteristics of pressure fluctuation and behavior of cavitating vortices are introduced by comparing the results of CFD and the model test. Furthermore, to investigate the cause of the unsteady phenomena, the inner flow at the overload is checked out.

2. Specification of hydraulic plant and way of model test and simulation
A Francis turbine scaled model of specific speed \( n_s = 81.3 \text{(m-kW)} \) was an investigation object. The specification of model test was shown in Table 1.

The model test set was shown in Fig. 1. The turbine contained 18 guide vanes, a runner with 15 blades and a bent draft tube. The runner outlet diameter was \( \phi 318.1 \text{mm} \). The monitoring point of the pressure fluctuation was placed on the upper draft tube. 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. While the net head of the model test was 35m, the pressure fluctuation was measured with changing the guide vane opening.

| Table 1 Specification of model |
|-------------------------------|
| Reference Diameter D (mm) | 318.1 |
| Speed Factor \( n_{ED} \) (-) | 0.2034 |
| Discharge Factor \( Q_{ED} \) (-) | 0.1650 |
| Specific Speed \( n_s \) (m-kW) | 81.3 |
| Thoma number \( \sigma \) (-) | 0.0445 |

The simulations were carried out with the use of the commercial CFD software ANSYS CFX 13.0. 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 inner flow of runner was influenced by the guide vane outlet flow. So, the simulation contained all guide vanes, whole runner blades, and a draft tube in the computational domain in Fig. 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. Model Test Results and CFD simulation Results

3.1. Model Test Results
The pressure fluctuation as the discharge in the model test was shown in Fig. 3. Here, the discharge was normalized by the value of the best efficiency point (BEP). The magnitude of the pressure fluctuation was normalized by the net head. Between \( Q/Q_{\text{BEP}} = 0.4 \) and 0.8, the pressure fluctuation peaked at \( Q/Q_{\text{BEP}} = 0.6 \). And the pressure fluctuation increased over \( Q/Q_{\text{BEP}} = 1.1 \). By the way, between \( Q/Q_{\text{BEP}} = 0.8 \) and 1.1 the pressure fluctuation was pointed the minimum value. The pressure fluctuation around \( Q_{\text{BEP}} = 0.6 \) and 1.2 were so high that this operating points were focused on as the representative points of the partial load and the overload, and the CFD simulations were carried out at these points.
3.2. CFD simulation Results
The comparison of the pressure fluctuation between the CFD simulation and the model test were shown in Fig. 4. Here, the pressure fluctuation was defined as the pulsation of the static pressure as advance of time at the monitoring point in the upper draft tube showed in Fig. 1. The time $t_n$ was normalized by using the rotational speed of runner $n$ as follows

$$t_n = \frac{t \cdot n}{60}$$

(1)

This normalization converted the times or periods into equivalent numbers to the runner rotation. The pressure in both the CFD simulation and the model test were normalized by the net head.

The pressure pulsation obtained at $Q/Q_{BEP}=0.6$ in the CFD simulation was the almost similar profile to that of the model test shown in Fig. 4 (a). The averaged period of the pulsation was roughly estimated to be 3 times longer than the runner rotation period as with the model test. These results indicated that the present simulation could accurately estimate the wave shape and frequency of the pressure pulsation at the partial load condition. The amplitude in the CFD simulation was a little lower than that of the model test, but could be estimated with enough accuracy with the unsteady flow pattern taken into consideration.

The pressure fluctuation at $Q/Q_{BEP}=1.2$ described the very low-frequency and uniquely shaped profile in the CFD simulation shown in Fig. 4 (b). The estimated period at $Q/Q_{BEP}=1.2$ was much longer than that of $Q/Q_{BEP}=0.6$. The frequency of the CFD simulation was obtained about 15 times as long as the runner rotating period. On the other hand the model test frequency was randomly obtained from 10 to 20 times as long as the runner rotating period. This was due to the unsteady inner flow pattern discussed in the following subsection. In light of the above, the averaged period in the whole measurement time in the model test seemed to agree with that in the CFD simulation. A quantitative discussion about the amplitude and frequency of the pressure pulsation is provided later. The wave shape of the primary component in the CFD simulation was similar to the model test, except for the high frequency components in the model test at the low pressure moment. At the low pressure moment, the pressure was almost constant in the CFD simulation. This unique wave shape was strongly related to the flow structure, which consisted of water and cavities. The relation between the flow structure and pulsation is discussed in the following subsection.

![Fig. 4 Comparison of pressure fluctuation between CFD and model test](image-url)
3.3. Investigation of Inner Flow

The shapes of a cavity at each quarter period of the pressure fluctuation were visualized by using an isosurface, which represented the constant vapor volume fraction $\gamma = 0.5$, to understand the structure of the flow field at $Q/Q_{\text{BEP}}=0.6$ (see Fig. 5). The photographs obtained in the model test were compared with the CFD simulation in the corresponding phases. The velocity and static pressure profiles in the horizontal section that crosses the monitoring point were also shown. The velocity was normalized by using the peripheral velocity of the runner outlet and the static pressure was normalized by using the net head. The velocity from the upstream to the downstream was evaluated by the negative value.

![Fig. 5 Behavior of cavitating vortices at $Q/Q_{\text{BEP}}=0.6$ (partial load)](image)
In the smaller flow rate condition corresponding to partial load, what is called vortex rope was obtained. The estimated shape of cavity changed little, while the position and the orientation enormously changed as time progressed. The pressure pulsation at the wall surface of the upper draft tube had a close connection with the revolution of the cavity. The shape of a cavity in the CFD simulation was similar to that in the model test.

A detailed investigation of the velocity profiles in the section indicated that the high-speed region of the axial velocity was located close to the cavitating vortex core. The negative component of axial velocity was especially enhanced at the outer side of the vortex core. In the center of the draft tube, a wide stagnation area was spread. These meant that the greater part of the main flow was concentrated at a limited area between the cavitating vortex and wall surface of the draft tube.

As for the static pressure, the spatial variation was prominent compared with the transition caused by the time advancement. This characteristic was in striking contrast to the overload condition.

The estimated shapes of cavity at $Q/Q_{\text{BEP}}=1.2$ in the CFD simulation were compared with the model test in Fig. 6. Here, the corresponding velocity and static pressure profiles were also shown. The CFD simulation results showed good agreement with the model test. Both the CFD simulation and the model test indicated that the shape and size of the cavity changed drastically as time progressed. The pressure fluctuation at $Q/Q_{\text{BEP}}=1.2$ indicated that the wave in Fig. 4 (b) was roughly composed by 3 phase of the cavity, that is, the constant low pressure phase, the increased pressure phase and the decreased pressure period, and the frequency was longer than that at $Q/Q_{\text{BEP}}=0.6$. Therefore, the characteristic 4 moments were picked up in Fig. 6.

During the time range of $38 < t_n < 43$, the bell-shaped cavity was elongated in the upper draft tube. The shape and size of the cavity around the monitoring point was almost constant, while the underside of a cavity, which was presented as uneven shape in Fig. 6 (a), was rotating in the opposite direction to the runner rotating drastically below the monitoring point as the time progressed. In the model test, the pressure of the cavities seemed to fluctuate due to the rotation of the cavity underside. On the other hand, in the CFD simulation, the pressure of the gas is not decreased under the saturated vapor pressure. Therefore, the obtained static pressure kept a constant value.

After the period in which the size of the cavity kept constant below the monitoring point, the cavity shrunk toward the upstream direction (see Fig. 6(b)). In this phase, a remarkable reverse flow occurred in the center of the draft tube.

Around the maximum pressure at the monitoring point, a uniquely shaped cavity appeared in a short time, as shown in Fig. 6(c). At this moment, the shapes of the cavity, velocity, and static pressure profiles were similar to those at $Q/Q_{\text{BEP}}=0.6$ except for the inversed spiral direction. However, the cavity at $Q/Q_{\text{BEP}}=1.2$ was rapidly elongated, while it was almost constant at $Q/Q_{\text{BEP}}=0.6$.

In a short while, the spiral-shaped cavity was covered by a bell-shaped cavity (Fig. 6(d)) and still continued to enlarge. In this phase, the reverse flow disappeared, and whole flow direction was downstream, even in the center of the draft tube.

When the end of the cavity reached the bent part of the draft tube, the expansion and constriction calmed down in a certain period and returned to the condition showed in Fig. 6 (a).

A comparison with $Q/Q_{\text{BEP}}=0.6$ pointed out the special features of velocity and static pressure profiles at $Q/Q_{\text{BEP}}=1.2$. That is, the flow field was extremely variable with the advance of time. In particular, the axial velocity component in the center of the draft tube was increased to remove the cavity and decreased to extinguish the reverse flow area.
To investigate the cause of the reverse flow which seemed to make the cavity enlarge and shrink at \( Q/Q_{BEP} = 1.2 \), the \( z-x \) sectional velocity contours in the draft tube at the same moment as Fig. 6 were shown in Fig. 7. Here, the velocity was normalized by using the peripheral velocity of the runner outlet. To understand the flow direction, the typical velocity vector was also shown. As for the sectional velocity, the main flow to downstream was always biased to the wall and the reverse flow occurs near a center around which the cavity was occurred. At \( t_n = 38.31 \), which was the moment with the elongated bell-shaped cavity in Fig. 6 (a), the flow seemed to separate at the inner side of the draft.
tube bend, and the direction of the separated flow seemed to be reverse flow (see Fig. 7 (a)). It was considered that the separation at the bent draft tube was influenced by the shape of the draft tube, that is, the diffuser and bent channel, on which the downstream pressure was higher and pressure deviation was caused. So the flow tended to be separated from the wall. During the shrinkage of cavity at \( t_n = 44.97 \) and 46.64 in Fig. 6 (b) and (c), the flow separation was disappeared in the draft tube bend and reverse flow was located more upstream than at \( t_n = 38.31 \) (see Fig. 7 (b) and (c)). At the enlarged period in Fig. 6 (d), the reverse flow was hardly occurred and the flow separation seemed to be incipient in the draft tube bend.

![Fig. 7 Velocity contour in the draft tube](image)

The meridian velocity and pressure distribution at the same moment as Fig. 6 and 7 was shown in Fig. 8. The measuring section was the section of monitoring point of the pressure pulsation located upstream of the bend (section A) and the middle of the draft tube bend (section B) shown in Fig. 7 (a). Here, the meridian velocity was normalized by using the peripheral velocity of the runner outlet, the pressure was defined as pressure coefficient normalized by the net head and the length was defined by the length from the outer side to the inner side of the measuring section of the draft tube as -1 to 1. The velocity from the upstream to the downstream was evaluated by the negative value.

The meridian velocity at the section A showed that the main flow near the wall at the moment of \( t_n = 44.97 \) and 46.64, at which the cavity was shrinking, was lower than that at \( t_n = 38.31 \) and 49.97 (see Fig. 8 (a-1)). This meant that the instantaneous discharge of the main flow on the upper draft tube became small with the cavity shrinking. The reverse flow around the center was the maximum at \( t_n = 44.97 \), at which the cavity was shrinking in Fig. 6 (b), and the order was \( t_n = 44.97, 38.31, 46.64, 49.97 \). This results were also showed in Fig. 7, and meant that reverse flow was involved with the shrinkage of the cavity. The pressure at the section A became higher in the order of the reverse flow to \( t_n = 46.64, 44.97, 49.97, 38.31 \) shown in Fig. 8 (a-2). Because this order was showed the next period of the descending order of the reverse flow, it was considered that the reverse flow brought the high pressure from downstream to upstream. The pressure curve as the length at \( t_n = 44.97 \) and 46.64 were wavy because the cavity was shaped spiral showed in Fig. 6 (b) and (c).
The meridian velocity at the section B showed that the velocity near the inner wall indicated the positive value meant separated flow at $t_n=38.31$, while at $t_n=44.97$ and 46.64 the velocity near the inner wall indicated the negative value (see Fig. 8 (b-1)). The pressure at the section B in Fig. 8 (b-2) became higher in the order to $t_n=46.64, 44.97, 49.97, 38.31$. This order was interrelated with the order of the cavity size and the distance between the underside of the cavity and the section B (see Fig. 6). Because of the diffuser channel, the pressure of the section B was higher than that of the section A.

![Normalized meridian velocity at section A](image1)

![Pressure coefficient at section A](image2)

![Normalized meridian velocity at section B](image3)

![Pressure coefficient at section B](image4)

**Fig. 8 Velocity and pressure distribution on the draft tube**

The following summary was obtained from the above results. The cavity was elongated by the strong circumferential flow (see Fig. 6 (d) and (a)). After the cavity had been elongated to downstream near the draft tube bend, the flow near the inner side of the bent draft tube was separated due to the diffuser and bent effect (see Fig. 7 (d) and (a)). The reverse flow increased with appended the separated flow, which brought the high pressure fluid (see Fig. 7 (b) and Fig. 8). Then, the pressure around the cavity became higher than the saturated vapor pressure and the cavity was shrunk (see Fig. 6 (b) and (c)). Because the pressure at the upper draft tube was increased, the net head from the casing to the draft tube reduced, and then the instantaneous discharge became small (see Fig. 8 (a-1)). The separated flow at the bent draft tube disappeared with the reduction of discharge, and then the circumferential flow make the cavity was elongated. This cycle is repeated with 15 times as long as the runner rotation period.
4. Conclusion

A series of numerical simulations for a Francis turbine was carried out to estimate the unsteady motion of the cavity in the draft tube under the overload condition. To confirm the accuracy of the simulations, model tests for corresponding operating points were also performed.

As a result, a long-period pulsation for pressure was estimated by the numerical simulation under the overload condition. The obtained period was about 15 times as long as the runner rotation period. Shrinkage of cavitation by the remarkable reverse flow was found to occur in each period. After the shrinkage, the quick elongation of spiral shaped cavity was also obtained. As for the reverse flow, the separated flow at the bent draft tube was occurred with the diffuser and bent effect after the elongation of the cavity, and flowed from the downstream to the upstream. It was considered that the reverse flow brought the high pressure fluid and the cavity was shrunk. The occurrences of the long-period pressure pulsation and above mentioned drastic changes of cavitation structure were confirmed by using model tests. Both the numerical and experimental approaches pointed out that the long-period oscillation was caused by the drastic changes of cavitation structure.

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] Allign´e, 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] 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.