Development of Low-Specific Speed New-Type Hydraulic Turbine Equipped with Volute: Numerical Investigation and Performance Prediction

T Irie¹*, W Takahashi¹, K Miyagawa¹, T Sugimoto², T Naganuma², and R Waku²

¹ Department of Applied Mechanics and Aerospace Engineering, Waseda University, Tokyo, 1698555, Japan
² Tohoku Small Hydro Co., Ltd., Akita, 0101623, Japan
iritatsumarines1@toki.waseda.jp

Abstract. With the goal of broadening the operation range of hydraulic turbines, we propose a new-type hydraulic turbine in our research. The new turbine has no draft tube, but instead is equipped with a parallel diffuser and a volute, i.e. a radial vaneless diffuser and a spiral discharge collector, respectively. In this research, we carried out a CFD investigation on specific speed 77 [m-kW] prototype turbine to predict its performance. The simulation results showed that under large discharge there is a high risk of cavitation or pressure fluctuation. In this condition, the amplitudes of the pressure fluctuation are 0.6 percent and 1.3 percent of the effective head, on the volute wall and the parallel diffuser wall, respectively. The possibility of flow instabilities such as a vortex rope precession and a rotating stall to occur is small enough to keep operating the turbine at both deep part-load and full load. The final design of the turbine has been decided based on this investigation. The turbine verification tests are going to start in early 2021.

1. Introduction

Francis turbines operated in off-design conditions experience momentous efficiency deterioration and the development of the flow instability [1][2]. This diminishes the capability of hydropower plants to operate at flexible conditions, resulting in the turbine shutdown when in an undesired discharge or head. Furthermore, it interferes with the turbine operation for electrical grid stabilization, which is becoming essential in recent years [3]. For the sake of the stable turbine operation, improvements such as radial grooves, fins on the draft tube (DT) wall, and an air injection method have been put into practical use [4~6]. These methods focus on the reduction of the magnitudes of a swirling flow or the mitigation of the precession of a vortex rope inside the DT. They have successfully broadened the operation range of the turbine. Unfortunately, they simultaneously act as an energy-loss factor. Therefore, those methods cannot fully achieve the requirement of the flexible turbine operation.

Our new-type turbine features both high efficiency and the mitigation of flow instability over a wide range of operating conditions. The new turbine is equipped with a volute (VD) and a parallel diffuser (PD) instead of the commonly used DT. The PD is a radial vaneless diffuser connects to the runner (RV) outlet and the VD inlet. We have performed Computational Fluid Dynamics (CFD) investigations to predict turbine performance. The previous papers have reported about the flow and the loss mechanisms of a specific speed $N_S=134$ [m-kW] new turbine. According to the previous reports, the following features of the new turbine were predicted [7,8]:

The turbine verification tests are going to start in early 2021.
The new turbine keeps higher efficiency at part load, i.e., a small discharged operation, on the contrary to the same specific-speed Francis turbine.

At overload, i.e., a large discharge, the PD outflow features low $V_U$ and high-uniformed $V_m$. It feeds the circulation inside the VD and generates a strong vortex. Concurrently, the supply from the PD exists all along the VD passage, managing the vortex stay in the center.

At deep-part load, the PD internal flow features high $V_U$ and low non-uniformed $V_m$.

Based on those characteristics, a new-type turbine with $N_S=77$ [m-kW] was designed as shown in Figure 1.

Figure 2 shows the velocity diagrams of the new turbine runner. The swirl of the runner outflow remains in the same direction as the runner revolution. L.E. and T.E. indicate the blade leading edge and the blade trailing edge, respectively.

![Figure 1. Configuration of the new turbine ($N_S=77$ [m-kW])](image1)

![Figure 2. Velocity diagrams of a new turbine](image2)

The main difference between $N_S=134$ [m-kW] turbine and $N_S=77$ [m-kW] turbine is the shape of the PD, as illustrated in Figure 3. Previous CFD studies showed that the reverse flow develops near the PD shroud at overload, and near the PD hub at deep-part load. Therefore, we introduced the modified PD passage to mitigate the loss and the flow instability.

The PD and the VD systems were introduced for the first time in hydraulic turbines. However, a similar component to the PD exists in pumps or compressors, known as a vaneless diffuser. Kurokawa et al [9] have reported that the rotating stall in the vaneless diffuser developed by the reverse flow consequently leads to severe pressure fluctuation. They have also pointed out that the complex and multiple rotating cells accrue in the diffuser with a high radial-expansion rate [9]. By following this report, the development of the rotating stall at part load is also possible in the new turbine. To reduce this possibility, it is indispensable to restrain the growth of the reverse flow region. On that account, we introduced the mixed PD passage.

Table 1 shows the specification of the prototype new turbine. The targeted ranges for the turbine operation are 15%~110% as the discharge and 50%~150% as the effective head.
2. Numerical Setup
We predicted the performance of the new turbine with CFD simulation using ANSYS CFX 19.0 commercial code. Two types of CFD investigations were carried out in this paper, simulation with partial domain and with full domain model. The partial domain model of the turbine consists of guide vanes (GV), a runner (RV), a PD, and a VD domain. Meanwhile, the full domain model also includes a casing (SC) and stay vanes (SV). Each were used for unsteady and steady state simulation, respectively.

Figure 4 indicates the grid dependency of the model using the domain for unsteady simulation. Concerning the simulation time cost, those simulations on the figure run in a periodic boundary condition.

According to the result of efficiency versus element number, the mesh around 1.92 million (8.7 million for the unsteady condition) is found reasonable for the main computation. This investigation run in conditions of Guide Vane Opening (GVO) 90% and effective head $H_e$ 100%.

Table 1. Turbine Specification

| Items               | Units | Values     |
|---------------------|-------|------------|
| Specific speed      | $N_S$ | m-kW       |
| Turbine output      | $P_T$ | kW         |
| Rated net head      | $H_{e100}$ | m   |
| Maximum discharge   | $Q_{max}$ | m$^3$/s |
| Rotation speed      | $N$   | min$^{-1}$ |
| Inlet diameter      | $D_1$ | m          |
| The number of vanes | Stay vane | -     |
|                     | Guide vane | -     |
|                     | RV       | -         |

![Figure 4. Grid dependency](image-url)
Figure 5 shows the mesh configuration.

![Mesh Configuration](image)

**Figure 5. Mesh configuration**

Since the prototype is yet to be installed, the CFD parameters are decided based on the comparison and the validation results reported by Nakamura et al [10]. Table 2 shows the CFD analyses conditions.

| Items                  | Conditions (Unsteady / Steady)            |
|------------------------|-------------------------------------------|
| Turbulence Model       | SAS, SST / RANS, SST k-ω                  |
| The number of Elements |                                           |
| SC                     | 1360000 (Unstructured)                    |
| SV                     | 3680000 (Turbo Grid, full channel)        |
| GV (15 ~ 110%)         | 4000000-5000000 (Turbo Grid, full channel)|
| RV                     | 4280000 (Turbo Grid, full channel)        |
| PD                     | 810000 (Turbo Grid, full channel)         |
| VD                     | 313000 (Unstructured)                     |
| Intersection           |                                           |
| GV ~ RV                | Stage Mixing Plane                        |
| RV ~ PD                | Transient rotor stator / Frozen Rotor     |
| Boundary Condition     |                                           |
| Inlet                  | Total Pressure $P_t$ [Pa-G]               |
| Outlet                 | Static Pressure $P_s = 0$ [Pa-G]          |
| Run $GVO$              | 15, 30, 110 / 15, 30, 60, 80, 90, 110 %   |
| Run effective head $He$| 100 % = 47.4[m]                           |
| Time Step (Unsteady Simulation) | 0.000111 [s]                         |

### 3. Unsteady Simulation Results

Figure 6 shows the computational domain in the unsteady simulation. It consists of one GV, full RV, full PD channel, and a VD. The monitor points on the VD wall are arranged circularly, as illustrated in the sectional view of Figure 6 b.). The points in the PD are also plotted circularly on both shroud and hub walls. The amplitude of the pressure pulsation and the phase difference could be predicted from the simulation.

S1 and S2 indicate the points on the shroud wall, H1 and H2, the points on the hub wall of the PD. Suffix 1 and 2 indicate points in domain inlet and outlet, respectively. From the PD upstream to downstream, the suffix of the point is defined as 1, A, B, C, D, and 2.

Figure 7 and Figure 8 show the raw pressure waveforms observed in those monitor points. The variation and the fluctuation of the static pressure on the wall is becoming wide at large discharge.
3. Pressure fluctuations
Figure 9 shows the FFT of the pressure waveforms observed on the wall of PD inlet and outlet (The monitor point in the position of 90 degrees). The frequency of the peak pressure is normalized by the runner rotational frequency. In this case, the blade Passage Frequency (BPF) is 14.

Here, we observed the greatest pressure fluctuation magnitude through the whole simulation results. Figure 10 and Figure 11 show the magnitudes of observed peak pressure in every section. The amplitude of the pressure fluctuation is normalized by the effective head $H_e$ 100% (47.4 [m]). Among the four points in circular disposition in every section, the largest amplitude is picked up and shown on Figure 10 and Figure 11. The background-transparent bars in each figure are the pressure fluctuation amplitudes observed in the CFD investigation of $N_s=134$ [m-kW] turbine. The amplitudes represented by 0 in are the pressure fluctuation observed in the fluid region close to the center, inside the VD 0 degree points.
Focusing on the four points in every section, there is no phase difference in the section located on the PD outlet and the VD. Meanwhile, the clear correlation between phase difference and the point location that is observable in the DT, is found on the shroud wall of the PD inlet.

The pressure fluctuation amplitude in the VD is maximal in section 315, where the VD spiral starts. In this section, the flow structure is intricate due to the recirculation $0 \rightarrow 315 \rightarrow 270$ from the pieced passage (The passage through section 0 to 315 is spatially connected by a pierced hole cut with the tongue.).

### 3.2. Cavitation

Figure 12 shows the static pressure contour on the runner blade surface. The areas of which absolute static pressure is less than 60000 [Pa] or the saturation water vapor pressure $P_v$, are highlighted with the solid colors: blue or red. As shown in the figure, a large discharge corresponds to a high-velocity region at the runner outlet. This induces the pressure drop at the suction side of the blade trailing edge shroud.

### 3.3. Discussion

The pressure fluctuation performance of the new turbine is evaluated based on of those unsteady simulation results. To keep track of the flow characteristic change inside the PD, we performed further...
FFT analyses using PD monitor points. These points are located on 90 degrees position of the PD hub and the shroud wall referring to Figure 6 b). The results are shown in Figure 13 and Figure 14.

![Figure 13. Pressure amplitude on PD shroud wall](image1)

| Monitor Points | Pressure Amplitude $\frac{\Delta p}{H_e}$ [%] |
|----------------|-----------------------------------------------|
| 90 S1, 90 S2  | $f/f_s = 14$                                   |
| 90 S3, 90 S4  | $f/f_s = 27$                                   |
| a.) $H_e 100\%$ / $GVO 15\%$ | b.) $H_e 100\%$ / $GVO 110\%$ |

![Figure 14. Pressure amplitude on PD hub wall](image2)

| Monitor Points | Pressure Amplitude $\frac{\Delta p}{H_e}$ [%] |
|----------------|-----------------------------------------------|
| 90 S1, 90 S2  | $f/f_s = 14$                                   |
| 90 S3, 90 S4  | $f/f_s = 27$                                   |
| a.) $H_e 100\%$ / $GVO 15\%$ | b.) $H_e 100\%$ / $GVO 110\%$ |

Figure 15 shows the $V_m$ distribution in the runner and the PD. The representing velocity averages each individual velocity in 360 degrees. Figure 16 shows the pressure field and the flow direction in 90 degrees position of the VD.

![Figure 15. The meridional velocity contours (at last time step)](image3)

| Velocity Contours | $V_m$ [m/s] |
|-------------------|-------------|
| 10.00 RV          | 3.75 PD     |
| Reverse Flow      | Hub         |
| Shroud            | Detachment  |
| a.) $H_e 100\%$ / $GVO 15\%$ | b.) $H_e 100\%$ / $GVO 30\%$ |
| c.) $H_e 100\%$ / $GVO 110\%$ |

![Figure 16. The pressure distribution in the volute (at last time step)](image4)

| Pressure Contours | $C_{p_v}$ |
|-------------------|-----------|
| 0.001 RV          | 0.005 PD  |
| 0.010 RV          | 0.015 PD  |
| Detachment        | Shroud    |
| a.) $H_e 100\%$ / $GVO 15\%$ | b.) $H_e 100\%$ / $GVO 30\%$ |
| c.) $H_e 100\%$ / $GVO 110\%$ |
The modification and the redesign resulted in the remarkable mitigation of pressure fluctuations at part load. The improvement of the pressure fluctuation in Figure 10 a.) H1 is here brought about by the mitigation of reverse flow which is strongly observed in $N_\text{s}=134$ [m-kW] turbine. With the shorter length-mixed passage, the improvement of pressure fluctuation on the PD shroud wall is achieved. This has been encouraged by its shorter length and gentle bend of the PD passage.

Rotating stalls did not develop for both $GVO$ 15% and 110%, and the amplitude of the pressure fluctuations never exceeds 1.3% of the effective head.

At $GVO$ 15%, the RV and the PD main flow lean toward the shroud wall due to the centrifugal force, as shown in Figure 15 a.). The circulation in the PD originates in the forward direction as PD outflow. Due to the low flow angle and the velocity field, the reverse flow and the mixing develop in the PD, inducing great energy loss. After passing the B section, the peaks are attenuated at both walls.

The greatest pressure fluctuation is observed at $GVO$ 110%. The peak arises in the BPF, indicating the passing of the cavitation region. Figure 14 b.) shows the unique transition of the pressure fluctuation peak. We can see that the peak in BPF rises when it reaches the HA to HB region. Shortly after the incline, it decreases at point HC and HD. At the same time, the peak pressure on the shroud wall experiences a sudden drop. This is explained by the detachment of the main flow from the PD shroud, as shown in Figure 15 c.). At high discharge, the flow features almost no swirl and passes directly through the PD channel. As reported in the previous paper, it is difficult for the high-velocity flow to adhere to the curvature of the PD shroud wall [8]. The flow is turning along with the rounded intake of the PD, and consequently recirculates in the reversed direction with the designated way as in Figure 16 a.) and b.). The collision between the PD outflow and the VF recirculation induces energy loss and the pressure fluctuation.

At the PD outlet, the peak is appearing in $f/f_N=4$ on both walls, nearly one-third of the BPF. Two considerable resources are feeding this peak, the VF circulation and the PD deceleration. Due to the collision in the PD, the part of the recirculating flow enters into the shroud side of the PD as reverse flow, affecting its pressure field. However, the circulation has a small dependency on simulation time, indicating that there should be no peak at $f/f_N=4$. Another considerable resource is the high or low-pressure region that is appearing near the PD intake, as shown in Figure 16 c.). The generation and extinction of the region repeat within a cycle. With the improvement in this rounded intake, this peak could be mitigated.

4. Steady Simulation Results
Taking those features into consideration, we designed the final turbine configuration, as shown in Figure 17. Since the new turbine runner has a greater angle than that of the Francis turbine’s (Figure 2), the runner can be manufactured without casting but with cutting. Our unsteady investigation is carried out with the b.) and c.) designs. The final design d.) is used for steady simulations.

Figure 18 shows the loss distribution and the discharge versus $GVO$ under 100% effective head. The losses are normalized by that of the design point. As shown in the figure, the loss of the GV is greater at part load. Referring to the velocity triangle Figure 2, the runner absolute inflow angle is designed largely in comparison to the Francis turbines. The swirl component exists at the outlet of the runner, even at the design point. For hydraulic turbines, the swirl of the runner outflow is a waste of energy. To
compensate this untouched energy, the extra angular momentum is required at runner inlet. The turbine is therefore designed to compensate for this by the smaller angle of the GV trailing edge.

The loss in Figure 18 does not include disk friction loss due to the limitation of the CFD capability. Our next interest is the estimation of the thrust force and the disk friction loss of the prototype. Regarding its low-specific speed, the effect of the runner disk friction becomes important because of its relatively large diameter.

Figure 18. Loss distribution and discharge versus GVO

5. Conclusion
In this research, we have developed a low-specific speed new turbine equipped with a volute and a parallel diffuser. By performing the unsteady and the steady CFD simulations, the characteristics of the turbine are predicted as follows:

- The risks of the pressure fluctuation and the cavitation become high when in large discharge. The magnitude of the pressure fluctuation amplitude is growing up into 1.3 percent of the effective head.
- At overload (100% effective head and 110% GVO), the largest pressure amplitude is observed on the shroud wall of the parallel diffuser inlet. This behavior contains the BPF and the cavitation is also critical at this condition. Due to the detachment of the stream from the shroud wall, the velocity distribution at the volute inlet is leaning toward the hub side. Therefore, the circulation in the volute section collides with the inflow from the parallel diffuser.
- At part load (100% effective head and 110% GVO), the pressure fluctuation quickly disappears in the parallel diffuser. Although the PD internal flow has the reverse flow and is featuring a low flow angle, the stall cell reported in a radial vaneless diffuser could not be found.
- According to the unsteady simulation results, vortex core precession is unlikely to develop. The rotating stall accompanied by important pressure fluctuation is not confirmed through the simulations.
- The final design of the prototype turbine has been completed. However, the computation cannot determine the influence of disk friction loss and thrust force. Through the verification tests of the prototype turbine, those features should be confirmed.

6. Acknowledgements
The authors would like to thank New Energy Foundation (NEF) supporting us for the development of the Yoroibata P/S hydro turbine.

The authors would like to thank Mr. Shigenai and his colleagues of Akita Prefecture (Japan) for their cooperation.
The authors would like to thank the Waseda Research Institute for Science and Engineering (WISE) for providing the support to the presented research, in the context of the project: ‘High performance and high reliability research for turbomachinery systems.’

The prototype turbine in Yoroibata power station, Akita prefecture, is going to carry out the field test on early 2021.

7. Nomenclature

| Symbol | Description                                      |
|--------|--------------------------------------------------|
| $N$    | Rotation speed [min$^{-1}$]                      |
| $g$    | Acceleration of gravity [m/s$^2$]                |
| $H_e$  | Effective head [m]                              |
| $N_S$  | Specific speed, $NP_t^{0.5}H_e^{-1.25}$          |
| $\beta$| Relative flow angle [degrees]                   |
| $V$    | Absolute velocity [m/s]                         |
| $V_m$  | Relative velocity at runner [m/s]               |
| $\_d$  | Subscript for ‘design point’ e.g.) $Q_d$        |
| $f$    | Frequency [Hz]                                   |
| $P_t$  | Turbine output [kW]                             |
| $\alpha$ | Absolute flow angle [degrees]                 |
| $U$    | Circumferential velocity at runner TE [m/s]     |
| $W$    | Relative velocity at runner [m/s]               |
| $V_U$  | Circumferential component of $V$ [m/s], $V_m \tan \alpha$ |
| $\eta_h$ | Hydraulic Efficiency [%], $Pr/(\rho g Q H_e) \times 100$ |
| $f_N$  | Runner rotational frequency [Hz]                |
| $P_s$  | Static pressure [Pa]                            |
| $P_v$  | Saturation Water Vapor Pressure [Pa], 3.17 [kPa] at 25 °C |
| $C_P$  | Static pressure coefficient [-], $P_s / (\rho g H_e)$ |

8. References

[1] Miyagawa K et al 2002 Flow instability in an elbow draft tube for a Francis pump-turbine 21st IAHR Symposium on Hydraulic Machinery and Systems (Lausanne)

[2] Miyagawa K et al 2006 Mitigation of draft tube flow instability with auxiliary parts in high head pump-turbine 23rd IAHR Symposium on Hydraulic machinery and Systems (Japan)

[3] Gaudard L and Romerio F 2014 The future of hydropower in Europe: Interconnecting climate, markets and policies Environmental Science & Policy 37 pp 172-181

[4] Kurokawa J et al 2010 Effect of J-Groove on the Suppression of Swirl Flow in a Conical Diffuser ASME. J. Fluids Eng. July 2010 132(7) 071101

[5] Susan-Resiga R et al 2006 Jet Control of the Draft Tube Vortex Rope in Francis Turbines at Partial Discharge 23rd IAHR Symposium conference (Yokohama, Japan) pp 67-80

[6] Nicolet C et al. 2011 Experimental investigations on upper part load vortex rope pressure fluctuations in francis turbine draft tube International Journal of Fluid Machinery and Systems 4(1) pp 179-190

[7] Irie T et al 2018 Proposal of a new hydraulic turbine capable of high efficiency operation over a wide range of flow rate and effective head 29th IAHR Symposium on Hydraulic Machinery and Systems (Japan)

[8] Irie T et al 2019 Prediction of the characteristics of a new hydraulic turbine capable of offering a high efficiency and a stable flow at part-load operation 2nd IAHR-Asia Symposium on Hydraulic Machinery and Systems (Korea)

[9] Saha S L et al 2000 Passive Control of Rotating Stall in a Parallel-Wall Vaned Diffuser by J-Grooves ASME. J. Fluids Eng. September 2001 123(3) pp 507–515

[10] Nakamura Y et al 2015 Development of Shroudless Francis Turbine AJKFluids 2015-ASME/JSME/KSME Joint Fluids Engineering Conference (Korea)