Analysis of flow characteristics in pumped storage unit during start-up in turbine mode

Tao Liu 1, Chao Wang 1, Zhiming Chen 1, Funan Chen 2, Huili Bi 2, Yongyao Luo 2, Zhengwei Wang 2, *

1 Construction and Management Branch of CSG Power Generation Co. Ltd, Guangzhou 510620, China
2 Department of Energy and Power Engineering, Tsinghua University, Beijing 100083, China

*Corresponding author e-mail: wzw@mail.tsinghua.edu.cn

Abstract. During the start-up, the unit enters into speed-no-load from static state. The runner is full of complex vortex, which hinders the water flow into the runner, and even leads to the failure of starting up. It is necessary to study the vortex and flow state to evaluate the stability of start-up. In this paper, the coupling method of one-dimensional pipeline system and three-dimensional flow inside the unit is used to simulate the start-up process and analyse the three-dimensional flow state in runner. The flow instability during the start-up process is deeply studied by analysing the pressure distribution, streamline and turbulent kinetic energy. The results show that the turbulence intensity in the runner decreases with the increase of discharge. During start-up, the vortices in the runner occupy the whole blade passage, and then disappear. With the decrease of discharge, the turbulent kinetic energy at the runner inlet rises and small vortex appears at runner inlet.

1. Introduction
The flow state in the unit is chaotic during start up, and the vortex is complex and changeable. In addition, the pressure wave in the pipeline system will also interfere with the flow in the unit, aggravating the internal flow. The complicated and chaotic flow conditions during start-up may cause strong pressure pulsation, increase the vibration of the unit, and make it difficult to connect to the grid, which brings great safety hazards to the unit [1-2].

The start-up process is a kind of transient process. The one-dimensional method of characteristic (1D MOC) was adopted to solve this problem. Allievi [3] and Bergeron [4] established the water hammer theory and calculation method. Streeter and Wylie [5-6] further elaborated on this theory and developed the numerical calculation technology. Since then, MOC has been widely used to analyze the transient process including start-up process. In recent years, a full three-dimensional computational fluid dynamics method (3D CFD) has been developed to model the unit and piping system and simulate the 3D characteristics of the flow field [7-9]. Another way to consider the 3D flow characteristics of the unit is to couple the 1D piping system with the 3D flow in the unit. Due to the long pipeline system, the 3D characteristics of the flow in the pipeline are not obvious, so it can be simplified to 1D flow; while the 3D characteristics of the flow in the unit are strong, which is suitable for analysis using
computational fluid dynamics methods. Cherny S [10] and Zhang X X [11] and others have further improved the method.

At present, the research on the flow instability in the unit during the small opening stage of the start-up process is not in-depth, and the related work needs to be improved. Therefore, this paper adopts the calculation method of coupling the 1D pipeline system with the 3D flow in the unit. The study of the 3D flow characteristics is helpful for further understanding of the flow instability of the unit during start up.

2. Pump-turbine unit

In this paper, the prototype pump turbine is used for research. The rated speed of the pump turbine is 500 r/min and the diameter is 4.3 m on the high-pressure side. The unit has 16 guide vanes. The runner consists of 5 main blades and 5 splitter blades. The 3D fluid domain of the pump-turbine unit is shown in Fig. 1.

Fig. 2 is a schematic diagram of the power station piping system. The upstream and downstream pipelines, upstream and downstream reservoirs, surge tanks and other components of the power station are all simplified into 1D models to participate in the calculation.

3. Numerical model and boundary conditions

3.1. One-dimensional flow calculation

It is assumed that upstream and downstream boundaries are constant water level boundaries, and the upstream and downstream reservoir water levels are 762.1m and 98m, respectively. The full characteristic curve of runner obtained from the model test is used, and the characteristic of runner can be obtained through interpolating by the Suter method. The start-up process calculated in this study is that the unit starts from stand still state until it stabilizes to speed-no-load state.

3.2. Three-dimensional flow calculation

The volute, stay vane, guide vane, runner, draft tube and clearances are meshed. The number of nodes is 4.73 million in the whole domain. There are 1.17 million nodes in runner and 1.77 million nodes in clearances. Based on the fluid domain shown in Fig. 1, the flow during start up is calculated. 9 time points are selected to understand the flow characteristics in runner. The total pressure boundary condition is set at the casing inlet, and static pressure is set at the draft tube outlet. The values of boundary conditions are determined according to the 1D calculation results. The steady-state calculation is performed first, and the result is used as the initial value for transient calculation. There are 180 time steps in a revolution.
4. Results and discussion

4.1. Energy characteristics

During the start-up process, the change of relative rotating rate is shown in Fig. 4. The relative rotating rate refers to the ratio of rotating rate to the rated value. The change of relative discharge and relative torque is shown in Fig. 5. The relative discharge and relative torque are the ratio of discharge and torque to rated value. In order to explain the changes of discharge and torque, the relative guide vane opening, which is the ratio of guide vane opening to the maximum value is also shown in Fig. 5. During the start-up process, as the opening of guide vane increases, water flows into the runner, and the discharge, the hydraulic torque and the rotating rate increases at the same time. At 18s, the rotating rate increased to 0.82 of the rated value, and the relative discharge and relative torque are 0.50 and 0.58 at this time. In order to make the runner smoothly reach the speed-no-load state, the guide vanes begin to close, the opening of the guide vanes is gradually reduced, and the flow and torque of the unit drop. Since the water is still doing work on the runner, the rotating rate of the runner is still increasing. At 25.7s, the torque of the runner drops to 0. At this time, the relative rotating rate is 1.02, and the relative discharge is 0.16. It can be considered that the runner has reached speed-no-load.

The pressure at casing inlet and draft tube outlet is obtained by 1D calculation. Therefore, the head of the unit is a given value in 3D calculation, and the discharge and torque can be calculated. Fig. 5 compares the 3D and the 1D calculation results. It can be seen that the two results have good consistency. Since the 1D calculation is carried out based on the model test, the 1D calculation result strictly corresponds to the model test result. Therefore, the consistency between the 3D calculation results and
the 1D calculation indicates that the 3D calculation grid and method used in this paper are suitable for this research.

4.2. Flow characteristics of runner

In this paper, the pressure coefficient $C_p$ is defined as

$$C_p = \frac{P - P_{ref}}{\rho g H} \quad (1)$$

Where $P$ is the pressure, $P_{ref}$ is the pressure at the end of the downstream pipeline, and $H$ is the rated head. Fig. 6 shows the pressure distribution at the middle section of the runner. At the initial stage of start-up, water cannot flow into the runner normally, but hits the front of the blade, forming a high-pressure zone, as shown in Fig. 6 (a). Part of water flows out of runner, and rotates in the vaneless area. Other part of water flows into runner and gradually forms a vortex motion, causing the low-pressure zone in runner, as shown in Fig. 6 (a). As the discharge and rotating rate increase, the high-pressure zone on the front of the blade gradually weakens, as shown in Fig. 6 (b). After 18 s, the pressure distribution in the runner decreases from the inlet to the outlet, as shown in Fig. 6(c)-(e).

![Figure 6. Pressure distribution at the middle section of runner](image)

Fig. 7 shows the streamline at the middle section of the runner. At first the rotating water flow is formed at the entrance of the runner. After the water flows into the runner, another rotating water flow is formed in the middle of the runner. The vortex movement in blade channel is complicated, as shown in Fig. 7 (a). As the discharge increases, part of water flows into the draft tube along the front of the blade while more water flow forms vortex on the back of the blade and occupies the entire blade, as shown in Fig. 7(b). At 18s, the discharge reaches the maximum, and the rotating rate also tends to the rated value. The direction of the velocity is still deviated from the blade profile, but water can flow into
and out of the runner smoothly. Because the splitter blades are short, their ability to deflect the flow is insufficient, so small vortex is formed on the back of the splitter blades, as shown in Fig. 7(c). Then, as the discharge decreases, the flow becomes chaotic, and the small vortex structure begins to appear, as shown in Fig. 7(d). At speed-no-load, although no large vortex structure is formed in the runner, the flow in the runner is chaotic, and small vortex are formed. The runner is in a state of half turbine and half pump, as shown in Fig. 7(e).

![Streamline at the middle section of runner](image)

**Figure 7.** Streamline at the middle section of runner

Fig. 8 is the turbulent kinetic energy distribution in the middle section of the runner. In the initial stage of start-up, the turbulence in runner is relatively severe. The turbulence at inlet is quite strong, and decreases as radius decreased. Due to the difference of the main and splitter blades, the turbulent kinetic energy distribution in the two neighbor blade passages is quite different. The turbulence in channel 1 is relatively violent, and is mainly concentrated in the front half of the channel, as shown in Fig. 8 (a). As the discharge increases, the turbulence at the inlet of runner is reduced, and middle part is stronger. There are two areas with more turbulence in channel 1, and turbulence in channel 2 is mainly concentrated on the back of the splitter blade, as shown in Fig. 8 (b). As the discharge further increases and the rotating rate increases, the turbulence in the runner decreases. When the discharge is maximum, there is still a part of the turbulent area on the back of the splitter blades in channel 2, and it appears as two vortex motions strung together, as shown in Fig. 8 (c). Then, the turbulence intensity is further reduced, as shown in Fig. 8 (d). After reaching speed-no-load, there is relatively strong turbulent kinetic energy at the inlet of the runner. The distribution of turbulent kinetic energy loses axial symmetry, and there is no obvious core with strong turbulent kinetic energy, which indicates that the flow at the runner inlet is very chaotic at this time, and there are many smaller vortex motions at the runner inlet, as shown in Fig. 8 (e).
5. Conclusion
In this paper, the method of coupling the 1D pipeline and the 3D internal flow is used to calculate the start-up process in turbine mode, and analyze the flow characteristics in runner. The main conclusions obtained are as follows:

Based on the coupling method, this study modeled the flow in the pump turbine, including the clearances, and took into account the changes of the boundary conditions caused by the pressure wave during start-up. The 3D flow phenomenon in the unit was simulated. This method can effectively predict the energy characteristics and flow characteristics of the unit, and provide a better understanding for the safe and stable operation of the unit;

In the initial stage of start-up, vortex appears at the inlet and the middle of the runner and gradually occupies the entire blade channel. As the rotating rate and discharge increase, the vortex is squeezed into the back of the blade and gradually disappears. This process is accompanied by the weakening of the turbulent kinetic energy in the runner. At speed-no-load condition, the turbulence at the inlet of the runner increased violently, and small vortex appeared at the inlet of the runner again. In the initial stage and speed-no-load stage of start-up, the turbulent motion is strong, which will cause problems such as vibration and noise in the unit, and affect the safety and stability of the unit.

Acknowledgments
This work was financially supported by National Natural Science Foundation of China (No. 51876099; No. 51779122).

References
[1] Frunzăverde D, Muntean S, Mărginean G, Failure analysis of a Francis turbine runner, In: IOP Conference Series: Earth Environmental Science: IOP Publishing 2010: 012115.
[2] Tanaka, Hiroshi, Vibration Behavior and Dynamic Stress of Runners of Very High Head Reversible Pump-turbines. International Journal of Fluid Machinery & Systems, 1990, 4.

[3] Allievi L. Theory of Water Hammer. Rome: Riccardo Garoni, 1925.

[4] Bergeron L. Water hammer in hydraulics and wave surges in electricity. New York: John Wiley and Sons, 1961.

[5] Wylie E B, Streeter V L. Fluid Transient. New York: Osborne McGraw-Hill, 1983.

[6] Streeter V L, Wylie E B, Bedford K W. Fluid Mechanics. New York: Osborne McGraw-Hill, 1998.

[7] Kolšek T, Duhošnik J, Bergant A. Simulation of unsteady flow and runner rotation during shut-down of an axial water turbine. Journal of Hydraulic Research, 2006, 44(1):129-137.

[8] Xia L-S, Zhang C-Z, Li H. Influences of runner blade shape on the transient behaviours of pump-turbines in load rejection. Journal of Hydraulic Research, 2020: 1-15.

[9] Li Z, Bi H, Wang Z. Three-dimensional simulation of unsteady flows in a pump-turbine during start-up transient up to speed no-load condition in generating mode. Proceedings of the Institution of Mechanical Engineers Part A Journal of Power & Energy, 2016, 230(6):570-585.

[10] Cherny S, Chirkov D, Bannikov D. 3D numerical simulation of transient processes in hydraulic turbines, In IOP Conference Series Earth and Environmental Science. IOP Publishing, 2010, 12(1):012071.

[11] Zhang X X, Cheng Y G. Simulation of hydraulic transients in hydropower system using the 1D-3D coupling approach. Journal of Hydrodynamics, Ser. B, 2012, 24(4): 595-604.