Analysis of Transient Force Performances of Pump-Turbine during Stop Process in Pump Mode

Yao Rao¹, Bi Huili¹, Wang Jun², Liu Tao², Chen Funan¹, Fan Honggang¹, Wang Zhengwei¹, Liu Jingshi³

¹ Department of Energy and Power Engineering, Tsinghua University (Beijing, CN)
² Construction and Management Branch of CSG Power Generation Co. Ltd. (Guangzhou, CN)
³ Harbin Electric Machinery Company Limited.

Abstract. The stop process occurs frequently for pump turbine due to the requirement of switching from pump mode to turbine mode. During stop process, the operating parameters such as the axial thrust, radial thrust and torque of the runner change rapidly, causing vibration of runner. In some cases, the rotating part can be lifted. In this paper, the unbalanced force acting on a pump-turbine during stop process in pump mode was studied using numerical simulation method. The CFD analysis was based on Reynolds averaged equation and the SST model was used to predict the transient flow characteristics. The clearance flow was also taken into consideration. Results show that the axial thrust was in opposite direction of gravity and changed with the variation of the head and torque. The axial thrust reached the maximum value in no-load condition.

1. Introduction
Among sustainable renewable energy sources, Hydraulic power generation has been recognized as an effective, flexible and advanced grid-regulating technology. Pumped storage power stations have the advantages of peak and valley filling, frequency and phase modulation. In particular, hydropower pumped storage has been recognized as, perhaps, the only commercially proven grid-scale energy-storage technique. The main equipment in pumped storage is the pump-turbines, which is different from hydro-turbines. In order to achieve load levelling, grid-frequency regulation, and reserve spinning, pump-turbines in pumped storage power stations frequently perform transient processes. Pump-turbines undergo all types of transient processes during their typical four-quadrant operations. During the transition process, the external characteristics change drastically, the internal flow conditions are complicated, and the unit will be damaged under some working conditions.

Method of numerical simulation has been adopted in extant researches to investigate transient processes. The one-dimensional method of characteristics (1-D MOC) is used to study the transient process and transient flows could be described using governing equations. 1-D MOC is a simplified method so that many complex nonlinear pulsating characteristics of transient processes cannot be captured and reproduced by using this method. Then, three-dimensional (3-D) flow simulation in water-conveyance system was proposed [1,2]. In order to reduce the calculation cost, some researchers only focus on hydraulic machinery, so they only use the 3-D method to simulate the transient flow inside the hydraulic machine. They focused on four issues: guide-vane movement, variation in rotational speed with time, unsteady boundary conditions at inlet and outlet of hydraulic machines, and turbulent-flow simulations [3-5]. To overcome these shortcomings, a coupled 1-D–3-D simulation
method was proposed to simulate interactions between transient flows in pipeline system and unsteady turbulent flows within hydraulic machines [6].

Based on numerical simulation methods of transient processes, some research results concerning transient characteristics and unsteady unstable flows within hydraulic machines during transient processes have been obtained as follows. Zhang et al. [7] simulated and produced looping dynamic characteristics during the runaway process. According to research results reported by Liu et al. [8], during a load-rejection process, vortex ropes in draft tubes rotate in the same direction as the runner.

A vortex rope within a draft tube is a general unstable unsteady flow. Its nature is still not fully understood. However, vortex ropes within draft tubes have been captured many times during transient processes [9]. Others have studied the force characteristics during the transient process. But there are still many issues that need to be studied in depth.

This paper studies the transient force performances of pump-turbine during stop process in pump mode. It adopts a coupled 1-D–3-D simulation method. First, the 1-D MOC is used to calculate the transmission of pressure waves in the duct system, and the external characteristics of the unit are obtained. At the same time, the outlet flow of the case and the inlet pressure of the draft tube are obtained as the boundary conditions for the three-dimensional CFD calculation. The axial thrust of the unit is analyzed, and the internal flow characteristics of the pump during stop process are discussed.

2. Numerical method

2.1. Governing equations and turbulence model

Pumped storage power station adopts one pipe and three machines arrangement. During the transient process, the transmission of pressure waves in the pipeline system of the power station is calculated using the Method of Characteristics (MOC). The control equation of the one-dimensional pipeline system is:

\[
\frac{\partial V}{\partial t} + V \frac{\partial V}{\partial x} + g \frac{\partial H}{\partial x} + \frac{f}{2D} V |V| = 0 \tag{1}
\]

\[
V \frac{\partial H}{\partial x} + \frac{\partial H}{\partial t} + \frac{\alpha^2}{g} \frac{\partial V}{\partial x} + V \sin \beta = 0 \tag{2}
\]

Here, \( H \) denotes the pressure head (m); \( V \) denotes velocity (m/s); \( t \) denotes time (s); \( x \) represents the distance along the pipeline axis (m); \( g \) denotes acceleration due to gravity (m/s\(^2\)); \( f \) is the coefficient of friction; \( D \) refers to pipeline diameter (m); \( \beta \) denotes water-hammer wave speed (m/s); \( \alpha \) represents the angle between pipeline axis and horizontal plane (degrees).

One-dimensional calculation can only get the external characteristics of the unit. In order to further study the internal flow performance and force characteristics of the unit, this paper selects the critical point of time during the stop process of the pump mode and performs CFD calculation. In the calculation process, the flow in the unit is regarded as a three-dimensional incompressible flow. The governing equation is:

\[
\frac{\partial u_i}{\partial t} = 0 \tag{3}
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = f_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \nabla^2 u_i \tag{4}
\]

Here, \( u_i \) and \( u_j \) are both transient velocity components (i, j=1, 2, 3); \( x_i \) and \( x_j \) denote Cartesian coordinate components; \( t \) denotes time; \( f_i \) is the body force component; \( \rho \) denotes the density of water; \( p \) represents the transient pressure; and \( \nu \) is the kinematic viscosity of water.

The SST model was used to predict the transient flow characteristics.
2.2. **Calculation model and boundary conditions**

The one-dimensional calculation domain is composed of upstream reservoir, water diversion well, diversion pipeline, downstream reservoir, tail water surge well, etc. In the calculation, the water levels of the upstream and downstream reservoir are 745m and 98m respectively. The characteristic curves of runner used in the calculation was obtained by the model test.

The domain for 3D CFD calculation is shown in figure 1. The calculation considers the flow in the clearance, that is, the clearance between the hub and the head cover and the clearance between the shroud and the bottom ring, as shown in figure 2. The grid of runners and clearance is shown in figure 3. The total number of nodes is 4,643,402. The domain of runner was discretized by tetrahedral mesh with 1,171,402 nodes and the clearance was discretized by hexahedral mesh with 1,771,714 nodes.

The boundary conditions of the CFD calculation are obtained through the results of one-dimensional calculation. Total pressure was set at the inlet of drafttube, and static pressure was set at the outlet of casing. The no-slip wall boundary conditions were set at the solid walls. Rotor-stator interfaces were used between runner and static domains.

3. **Results and discussion**
3.1. Performance characteristics

During the stop process in pump mode, the discharge and torque of blades decrease as guide vanes close. Then the rotating rate of runner begins to reduce after the discharge drops to zero. The guide vane closing process with rated rotating speed was calculated using 1D MOC. The relative discharge and torque of blades are shown in figure 4, which can be expressed as \( \frac{Q}{Q_n} \) and \( \frac{T}{T_n} \), where \( Q \) is the discharge, \( Q_n \) is the rated value, \( T \) is the torque of blades and \( T_n \) is the torque under rated power.

During the stop process, 10 time points are selected for three-dimensional flow calculation, as shown in figure 4. In the first eight seconds, the discharge changes slowly, and the interval of two adjacent time points is selected as 2s. The discharge changes dramatically after 8s and the interval of two time points is 1s. The comparison of 3D CFD simulation results and 1D MOC results is shown in figure 4. The relative values of discharge and torque predicted by 3D CFD are consistent with those of 1D MOC results, which indicates that the mesh used in CFD simulation is sufficient to get accurate 3D flow characteristics.

![Figure 4. Performance characteristics of the unit](image)

3.2. The axial hydraulic thrust of runner

The hydraulic thrust of runner during stop process is shown in figure 5. This article specifies that the hydraulic thrust takes the downward direction as positive. During the stop process, the hydraulic thrust direction is always upward. At the initial stage of stop process, the axial hydraulic thrust decreased slowly. Within 8s-12s, the axial force is greatly reduced. Later, when the opening of guide vane is very small, the hydraulic thrust rebounds and shows an increasing trend. Although the guide vanes are completely closed after 14s, the water in the clearance still rotates at a high speed under the drive of the runner, and will not be thrown out of the clearance immediately. Therefore, it can be speculated that after the guide vane is closed, the hydraulic thrust of the runner will not immediately drop to zero.
Figure 5. The hydraulic thrust by calculating.

The axial hydraulic thrust in figure 5 is the sum of the axial forces on all the flow surfaces of the runner. However, the flow surface of the runner can be divided into three parts: S1, S2 and S3, as shown in figure 6. Since the flow of clearance is considered in the CFD calculation process, the axial hydraulic thrust on the S2 and S3 surfaces can be directly obtained.

Figure 6. The flow surfaces of runner.

The hydraulic thrust on the three sets of flow surfaces is shown in figure 7. The axial hydraulic thrust direction of the S1 surface is upward, and its value is in a downward trend, and it is slow first and then fast. The change trend is basically the same as the torque. As the discharge of the unit gradually decreases, the pressure difference between the front and back of the blade decreases, and the torque of the blade decreases. At the same time, the axial hydraulic thrust value also shows a downward trend, so the axial force on the S1 surface continues to decrease. The axial force on the S2 and S3 surfaces show a trend of decreasing first and then increasing. With the decrease of the opening of guide vane, the discharge of the unit does not change much in the first few seconds, but then the discharge of the unit gradually decreased, and the head of the pump-turbine under the pump mode increased. In addition, when the opening of guide vane is small, the tangential velocity of the water flowing out of the runner is high, forming a "water ring" that rotates at high speed in the leafless area,
which hinders the outflow of water in the runner and causes the pressure in the leafless area to increase. The pressure in the clearance connecting the leafless areas also increases, as shown in figure 8. Therefore, the axial hydraulic thrust values on the S2 and S3 surfaces show an upward trend.

![Figure 7. The axial hydraulic thrust on S1, S2 and S3.](image)

(a) The axial hydraulic thrust on S1  
(b) The axial hydraulic thrust on S2 and S3

![Figure 8. Pressure on the section of clearance.](image)

(a) 4.06s  
(b) 8.13s  
(c) 11.98s

4. Conclusion
In this paper, the numerical simulation is used to analyze the hydraulic thrust characteristics of pump-turbine during stop process in pump mode. In the calculation, the one-dimensional method of characteristics was first used to obtain the external characteristics of the unit, and then the three-dimensional CFD calculation was performed to obtain the internal flow characteristics of the unit. The results show that the axial hydraulic thrust direction of the runner is upward, and the value continuously decreases at the initial stage of the stop process. When the opening of guide vane is extremely small, an upward trend appears again. The change of axial hydraulic thrust on each flow surface of the runner is related to the change of external characteristics and internal flow field.

In this paper, the interval of CFD calculation for the stop process in pump mode is large, and the effect of the flow field at the previous moment on the next moment cannot be considered. Therefore, it can only reflect the trend of the axial hydraulic thrust, and lack the detailed changing characteristics of hydraulic thrust, this is also the direction of the next work.
Acknowledgments

Special thanks are due to the National Key R&D Program of China [grant 2016YFC0401905] and the National Natural Science Foundation of China (Grant No. 51876099) for supporting the present work.

References

[1] Zhang L G and Zhou D Q 2013 CFD research on runaway transient of pumped storage power station caused by pumping power failure IOP Conference Series: Materials Science and Engineering 52(5) 052027

[2] Li J, Yu J and Wu Y 2010 3D unsteady turbulent simulations of transients of the Francis turbine IOP Conference Series: Earth and Environmental Science 12 012001

[3] Fu X, Li D, Wang H 2018 Influence of the clearance flow on the load rejection process in a pump-turbine Renewable Energy 127 310-321

[4] Pavesi G, Cavazzini G and Ardizzon G 2016 Numerical Analysis of the Transient Behaviour of a Variable Speed Pump-Turbine during a Pumping Power Reduction Scenario Energies 9(7) 1-1

[5] Ruprecht A and Helmrich T 2003 Simulation of the Water Hammer in a Hydro Power Plant based on IB-LB Journal of Huazhong University of Science and Technology (Natural Science Edition) 44(1) 122-127

[6] Zhang L G and Zhou D Q 2013 CFD research on runaway transient of pumped storage power Caused by Draft Tube Surge Proceedings of the ASME/JSME 2003 4th Joint Fluids Summer Engineering Conference 1 2811-2816

[7] Zhang X X, Cheng Y G and Xia L S 2014 Dynamic characteristics of a pump-turbine during hydraulic transients of a model pumped-storage system: 3D CFD simulation IOP Conference Series: Earth and Environmental Science 22(3) 032030

[8] Liu J T, Liu S H and Sun Y K 2012Numerical study of vortex rope during load rejection of a prototype pump-turbine IOP Conference Series: Earth and Environmental Science 15(3) 032044

[9] Stens C and Riedelbauch S 2016 Investigation of a fast transition from pump mode to generating mode in a model scale reversible pump turbine IOP Conference Series: Earth and Science 49 112001