Computational Fluid Dynamics-based Simulation to Francis Turbine under a Runaway Condition

Liying Wang, Bingyao Li, Weiguo Zhao and Qingjiao Cao

***Abstract*—**When the turbine operates in a runaway condition, the rotating speed of the runner increases sharply which results in serious threat to the safe of the units. In view of the instability phenomenon, a Francis turbine is simulated through Computational Fluid Dynamics (CFD). According to the internal control equation, the stable calculation results under the rated operation are used as the initial condition in the simulation, when the runner torque reaches a certain minimum value, the runaway condition is coming. Through analyzing the distribution of streamline under different speeds with different openings and the pressure distribution in the blade pressure surface and suction surface, it can be concluded that there exists a serious collision and off-flow phenomenon when the water flows into the runner. Meanwhile, through analyzing the streamline of draft tube in the meridian surface, it can be seen that the water flow with high rotating speed impacts on the draft tube wall, which causes the rising of pressure in the wall. With the increase of the opening, the vortex in the straight cone section is gradually reduced, additionally, the retention and secondary reflux in the diffusion section are significantly improved.

***Keywords*—**Computational fluid dynamics, Francis turbine, control equation, runaway condition.

I. INTRODUCTION

WHEN a turbine is running at load rejection, the power generated by the turbine runner is a balance with the mechanical loss power produced by the increase of the turbine speed, and the speed reaches a certain fixed value without rising, it’s because of the failure of the governor or some cause of the water guide body refusing to move, this is called the runaway condition. In this condition, the speed of runner rises sharply until it reaches the runaway speed, and the flow in the runner drastically deviates from the impingement entrance and the normal exit condition, which results in a larger impulse loss of the inlet and a loss of the outlet kinetic energy. Therefore, the vibration generated by the runner under the runaway condition will bring a serious threat to the safe operation of the whole unit [1-3]. So, it is very important to analyze the characteristics of the turbine under the runaway condition.

In the methods of studying the runaway condition, some scholars deduced a numerical method for the analytical equations of transient process according to the geometry of the hydraulic machinery under the assumptions of the cohesionless and one-dimensional rigidity, the results laid stress on the entire hydraulic system. Yanze [4] calculated the parameters of the bulb tubular turbine under the runaway condition by using a numerical method of the internal characteristic analysis theory, according to the expression of hydraulic turbine parameters in basic dynamic conditions, and the comparative experiment results are within the allowable range in project. The above two methods have calculated some characteristics of the turbine in the runaway process, but there are still some deficiencies in the reaction of the characteristics of the internal flow in runner. The CFD technology has been in a variety of areas of rapid development and wide application. In recent years, and it is widely used in hydraulic turbine simulation. Linsheng [5] simulated the runaway process of the bulb turbine using CFD technology and got the process parameter in pressure pulsation, torque and axial water thrust, and the evolution law of flow pattern was obtained. Jintao [6] simulated the transient process of the pump turbine in load rejection using CFD method according to the dynamic mesh and mesh reconstruction, finally the external characteristic and internal flow of pump turbine were obtained, it also can be used to solve the problems of transient process of pump turbine in pumped-storage power station. Daqing [7] established an unsteady three-dimensional turbulent flow numerical simulation method to simulate the axial flow turbine, and then obtained the variation law of speed and flow with time, it showed the obtained results is good agreement with the experimental data, which provided a strong reference in analysis of turbine transient process.

In this paper, combined with the synthetic characteristic curve, a Francis turbine is used to a CFD simulation in ANSYS CFX software, the internal flow and pressure distribution are researched under the full channel of hydraulic turbine, and some meaningful conclusions drawn.

This work was supported in part by the Natural Science Foundation of Hebei Province of China No. F2017402142 and the Scientific Research Key Project of University of Hebei Province No. ZD2017017.

Liying Wang is with the School of Water Conservancy and Hydropower, Hebei University of Engineering, Handan 056021, Hebei, China.

Bingyao Li is with the School of Water Conservancy and Hydropower, Hebei University of Engineering, Handan 056021, Hebei, China.

Weiguo Zhao is with the School of Water Conservancy and Hydropower, Hebei University of Engineering, Handan 056021, Hebei, China (corresponding author; e-mail: zwg770123@163.com).

Qingjiao Cao is with the School of Water Conservancy and Hydropower, Hebei University of Engineering, Handan 056021, Hebei, China.

II. NUMERICAL METHOD

A. Internal Control Equation

In case of a zero external load, the rotational speed increases sharply since the runner is directly impacted by the water flow, and it finally reaches the runaway speed. The equation express of momentum moment in this process is expressed as

$$M - M_f = J\frac{D\omega}{dt} \tag{1}$$

Where, M represents the total moment in turbine runner which is affected by the water flow; M_f is the resistance torque of the hydraulic turbine; J is the inertia moment of the system; ω is the angular velocity vector of the runner [5, 8].

From this equation, the increment of the turbine runner in each time step t can be known, and then the rotational speed at the next time step can be calculated, when the runner torque tends to zero, the speed is runaway speed.

In the rotational area, the governing equations are still the Renault average continuous equation and *N-S* equation under rotating relative coordinate system. However, this equation only is applied in a constant speed. When the turbine operates in the runaway condition, the rotating relative coordinate is always in a constant acceleration state therefore, the Renault average continuous equation and momentum equation in the accelerated rotating relative coordinate are deduced as

$$\nabla \cdot W = 0 \tag{2}$$

$$\frac{dW}{dt} = f + \omega^2 r - 2(\omega \times W) - \frac{\nabla p}{\rho} + \upsilon \Delta W - \frac{d\omega}{dt} \times R \tag{3}$$

Where, W is the absolute velocity vector in the accelerated rotating relative coordinate which is the relative velocity vector in the Cartesian coordinate system, f is the volume vector, υ is the flow viscosity coefficient, r is the radius vector of the spin axis pointing to the fluid particle, R is the vector from the origin of coordinates pointing to the fluid particles.

Comparing the momentum equation, under the accelerated rotating relative coordinate, it can be seen that the momentum equation adds a source term force $d\omega/dt \times R$, by decomposing the source term force, the additional source force component in x, y and z three directions under the absolute coordinate can be obtained, they are $yd\omega/dt$, $xd\omega/dt$ and 0 respectively.

The above equations are the flow control equation in the turbine runner, but in the other parts, the equations are still the continuous equation and the *N-S* equation in Cartesian coordinate system, which can be described as

$$\nabla \cdot V = 0 \tag{4}$$

$$\frac{dV}{dt} = f - \nabla p / \rho + \upsilon \Delta V \tag{5}$$

Where, V is the absolute velocity vector of water.

B. Numerical Simulation Method

Since the k-ε model is suitable for the vast majority of engineering turbulence models, it has good stability, simple and economy, the standard k-ε turbulence model is used based on *N-S* equation.

Where k is the amount of change in speed fluctuation, ε is the velocity fluctuating rate. According to the synthetic characteristic curve of hydraulic turbine, the following boundary conditions can be set in CFX.

(1) A mass flow rate is given in inlet, the pressure outlet is given in outlet.

(2) The slip boundary condition is adopted at the solid wall, and the wall area function method is used to calculate the near avoidance area.

(3) The connection between the surface of guide vane and runner is using the rotor-stator method.

(4) Take a steady-state calculation results in a rated operating conditions under one certain vane opening as the initial calculation conditions.

III. NUMERICAL EXPERIMENT

A. Example Implementation

A Francis turbine is simulated in this part and it contains four parts: the spiral casing, the guide vane, the runner and the draft tube. The amount of the guide vane and blade are also 16, the calculation domain part is shown in Fig. 1. According to the synthetic curve, the 6 conditions near the zero-angle line are selected under different guide openings [9, 10], which are listed in Table 1. The above 6 conditions contains the small flow conditions (condition 1 and 2), the high flow conditions (condition 5 and 6) and the optimal condition (condition 3). The tetrahedron and hexahedron mesh are separately performed to the spiral casing and draft tube using the ICEM CFD software, in addition, the guide vane and runner are meshed using the Turbogrid.

Fig. 1. Computational domain of full channel of Francis turbine

In calculating the runaway condition, the rated operating conditions should be calculated under 6 different guide openings, and the calculated results are as the initial condition, and then the change of internal flow and pressure characteristics can be directly seen between the rated and runaway conditions.

Table 1. Basic parameters in calculation conditions

Condition	Guide opening $a(°)$	Unit speed n_{11} $(r \cdot min^{-1})$	Unit flow rate $Q_{11}(rm^3 \cdot s^{-1})$
1	16	64	0.854
2	20	77.8	1.009
3	24	63.7	1.157
4	28	64.2	1.257
5	32	62	1.355

B. Characteristic Analysis of Runner Blades

In order to investigate the interaction between the water and the blade under the runaway condition, according to the different speeds, the distributions of meridian streamlines at different rotational speeds under different conditions are analyzed [11,12], the results are shown in Fig.2-4.

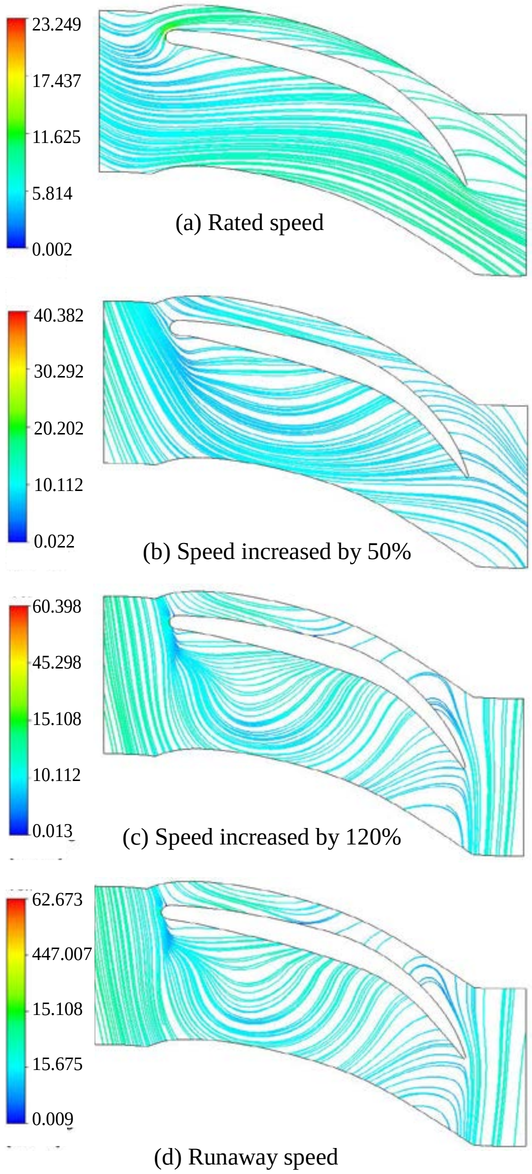
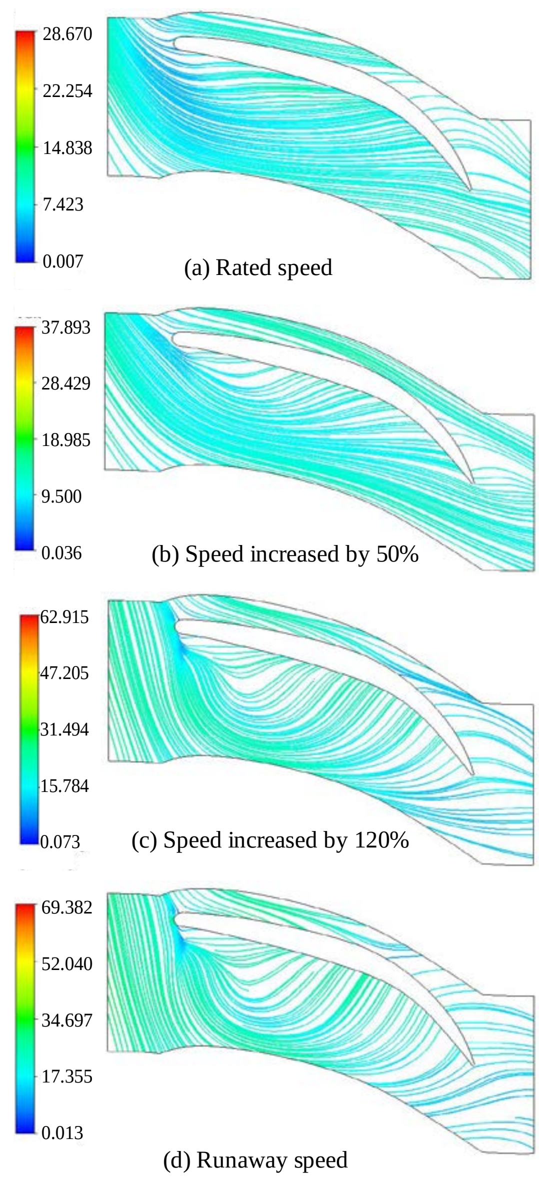


Fig. 2. Distributions of meridian streamlines at different rotational speeds under condition 1

From the figures, it can be seen that under the condition 1 (small flow condition), when the turbine rotates at the rated speed, the water flow impinges on the pressure side of the blade at a certain angle of incidence, thus causing a certain hydraulic loss at the blade inlet. Meanwhile, under the condition 3, the water flows into the runner with no impingement, at the outlet of blade, the streamline basically coincides with the normal exit of the blade. With the increasing of the speed, the flow of the diversion position gradually towards to the blade suction surface which can be seen in the inlet of the blade, so in this condition, the water flow deviates significantly from the flow of the impingement at the blade.



inlet.

Fig. 3. Distributions of meridian streamlines at different rotational speeds under condition 3

At the same time, the water flow increases rapidly with increasing speed, and the vortex appears in the internal of the runner, and the blade is also impacted by the turbulent flow. In the outlet of the blade, the direction does not flow in the normal direction. Moreover, the water with the high rotational speed flows into the draft tube under the runaway condition. It also will lead to the pressure fluctuation in the draft tube. Meanwhile, a large number of rotating water under the impact of cavitations bubble will form the vortex core which can produce the eccentric and hit the draft tube wall, thus resulting in the swing of the turbine. It will also induce the fatigue crack which can reduce the service life of the runner, finally, it will cause the greater energy loss.

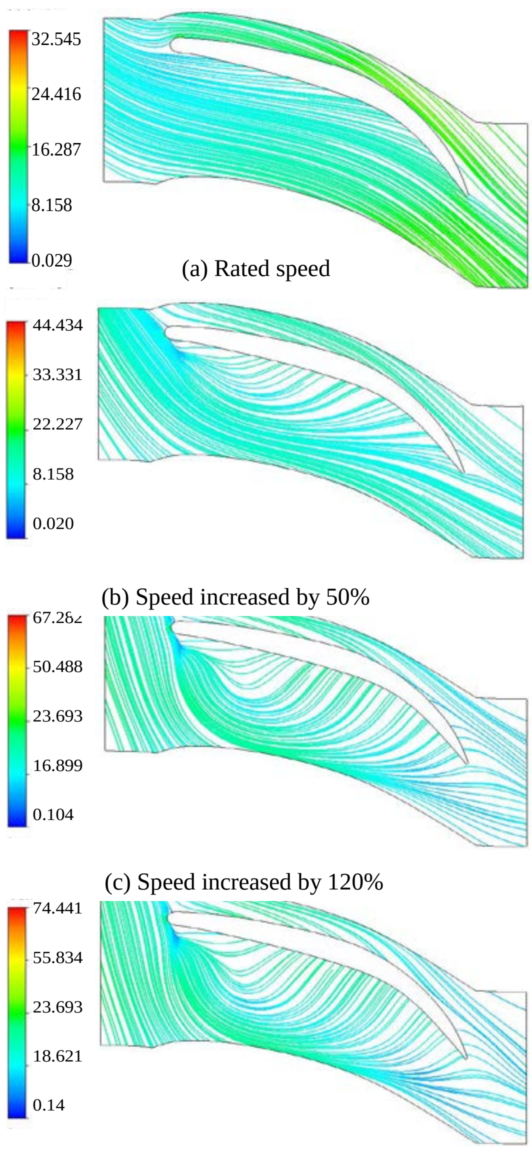


Fig. 4. Distributions of meridian streamlines at different rotational speeds under condition 5

Fig. 5-6 represents the pressure distribution of the blade pressure and suction surfaces at different speeds under condition 3. It can be seen from the pictures that the pressure in the inlet of blade shows a rapid increase in the trend. It is because that the circumferential velocity of the water flow at the inlet of the blade increases, but on the other hand, the water inlet angle gradually reduces which leads the serious strike in the inlet of the suction surface, and the off-flow phenomenon appears in the pressure surface. However, in the distribution of pressure surface, the pressure increases gradually in the shroud of the blade, and the negative pressure at the inlet close to the shroud reaches the maximum, which indicates that the off-flow phenomenon increases here. So, the airfoil cavitations are more likely to occur in this position.

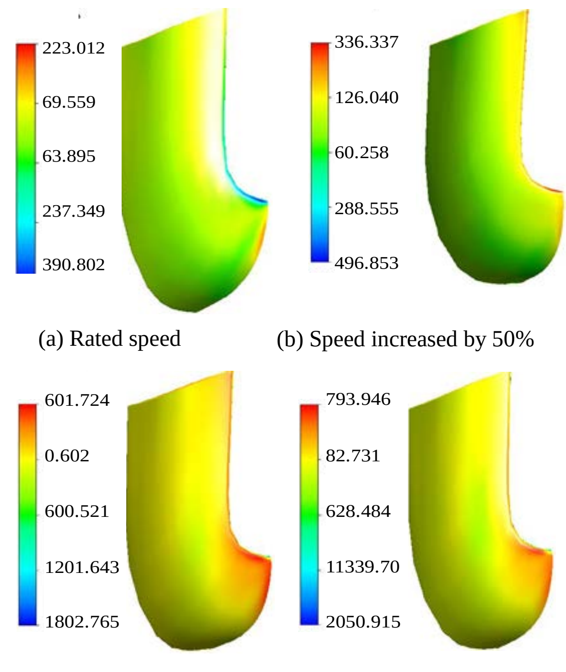


Fig. 5. Pressure distribution of blade pressure surface at different speeds under condition 3

C. Analysis of Dynamic Characteristics of Draft Tube

When the turbine operates in the runaway condition, the distributions of meridian streamline in draft tube are shown in Fig. 7-8. As the speed increases the water with the high rotational velocity flows into the draft tube. When the guide opening is very small, the abundant eddy and lower pressure area appear in the straight section which disturbs the water flows. At the same time, the ship diversion in the pipe wall is also very obvious which leads to the sharp increasing of pressure in the pipe wall in the straight section. The hydraulic stability in straight section will also be destroyed that will have a serious influence on the draft tube safety and then cause a vibration in draft tube. However, when the water flows into the diffusion section through the elbow, the water flow is more superior to that in the straight section, when the guide opening is 16, there is some obvious retention and secondary reflux phenomenon in the diffusion section. When it turns to 20, the phenomenon becomes more evident. With the increasing of the guide opening, the flow also increases, and the eddy in the straight section decreases obviously, but there is still a large pressure in the pipe wall. In the diffusion section, the water flows more regularly, the retention and secondary reflux phenomenon basically disappears, then the water stably flows to the downstream.

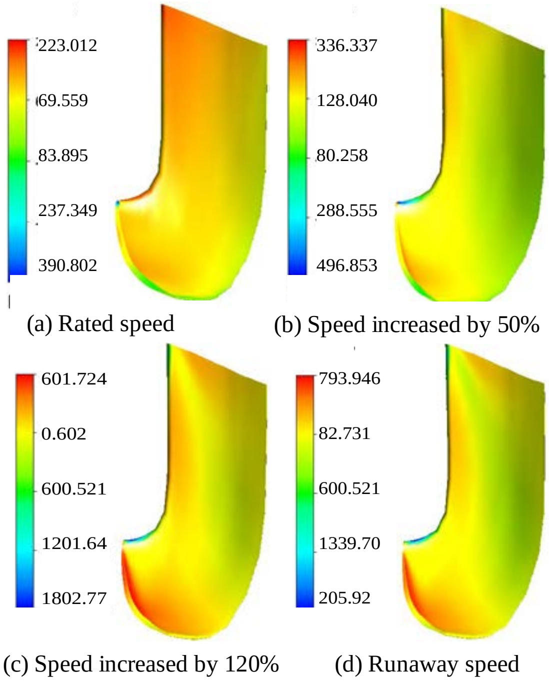


Fig. 6 Pressure distribution of blade suction surface at different speeds under condition 3

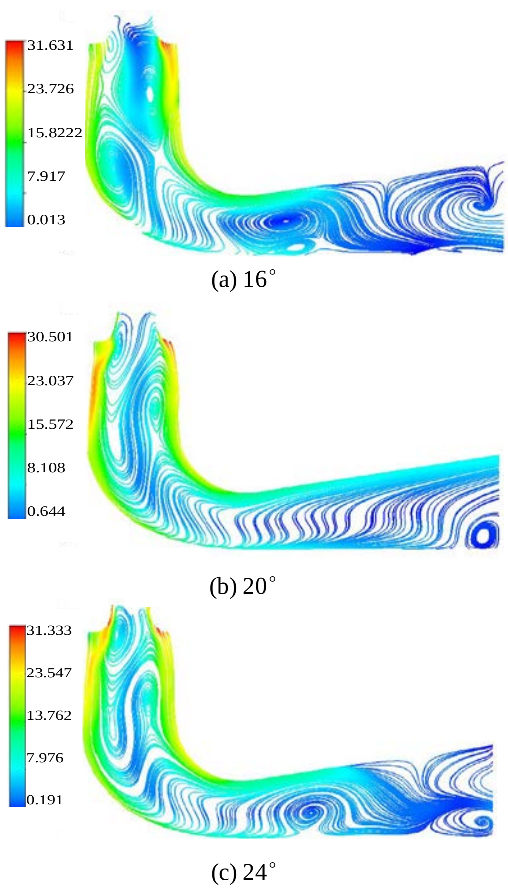


Fig. 7. Distribution of meridian streamline in draft tube at runaway condition under 16°, 20° and 24° openings

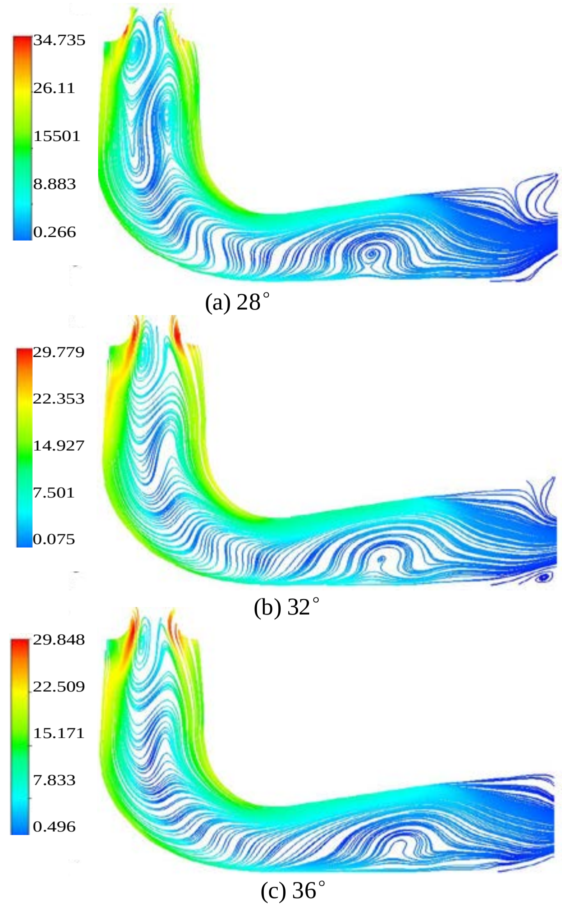


Fig. 8 Distribution of meridian streamline in draft tube at runaway condition under 28°, 32° and 36° openings

IV. CONCLUSIONS

In this paper, the ANSYS CFX software and the internal control equation of turbine runner are combined to three-dimensionally simulate the full channel of a Francis turbine, the distribution of meridian streamline of blade, the pressure distribution of pressure and suction surfaces of blade and the distribution of streamline of draft tube meridian are analyzed. The obtained conclusion accords with the theoretical properties and provides some references for the turbine design, and the conclusions can be drawn as follows.

(1) With the increasing of the rotational speed, the water flowing into the internal of the runner, seriously strikes the pressure surface of blade, which can result in the serious ship diversion, in the runaway condition, the water with the high rotational velocity has an influence on the draft tube which causes the vortex and vibration.

(2) The blades bears the strong flow impact and ship diversion in the runaway condition, the pressure sharply

increases in the inlet of the suction surface of blade, but in the pressure surface, there is a negative pressure in the inlet position, it will easily lead to the airfoil cavitations.

(3) With the increase of the guide opening, the eddy and ship diversion phenomenon in the draft tube gradually, decrease but the retention and secondary reflux also can be found in the small flow in the diffusion section, when the flow turns to large, but the phenomenon disappears under the great flow condition and the water flow changes regularly.

REFERENCES

[1] H. Zhang, C. Diyi, W. Changzhi et al, "Dynamic modeling and dynamical a nalysis of pump-turbines in S-shaped regions during runaway operation", *Energy Conversion and Management*. vol. 138, no. 15, pp. 375-382, 2017.

[2] A. Rezghi, and A. Riasi, "Sensitivity analysis of transient flow of two parallel pump-turbines operating at runaway", *Renewable Energy*, vol. 86, pp. 611-622, 2016.

[3] M. Fortin, S. Houde, and C. Deschênes, "Validation of simulation strategies for the flow in a model propeller turbine during a runaway event", *Earth and Environmental Science*, vol. 22, no. 3, pp. 1-12, 2014

[4] L.Yanze, and C.Jinshi, "Numerical Method Based on Internal Character for Load Rejection Transient Calculation of a Bulb Turbine Installation", *Journal of China Agricultural University*, vol. 13, no. 1, pp. 89-93, 2008.

[5] X. Linsheng, C. Yongguang, Z.Xiaoxi et al, "3D CFD Simulation of the Runaway Transient of Bulb Turbine", *Journal of Sichuan University*, vol. 16, no. 5, pp. 35-41, 2014.

[6] L. Jintao, L. Shuhong, S.Yuekun et al, "Three dimensional flow simulation of load rejection of a prototype pump-turbine", *Engineering with Computers*, vol. 29, pp. 417-426, 2013.

[7] Z. Daqing, W. Yulin, and L. Shuhong, "Three-dimensional CFD simulation of the runa way transient of a propeller turbine model", *Journal of Hydraulic Engineering*. vol. 41, no. 2, pp. 233-238, 2010.

[8] L. Jinwei, L. Shuhong, Z. Daqing et al, "Three-dimensional unsteady simulation of the runaway transient of the Francis turbine", *Journal of Hydraulic Engineering*, vol. 28, no. 1, pp. 178-182, 2009.

[9] Z. Yaping, L. Weili, L. Zhihua et al, "Numerical study on the performance of bulb turbine by using runaway condition to flood discharge and sediment dredging", *Earth and Environmental Science*, vol. 22, no. 5, pp. 1-6, 2014.

[10] S. Xiangzhong, Z. Wenlu, Y. Xuechun et al, "Study on mechanical properties of two-way fluid-structure interaction of Francis turbine runner", *Yangtze River*, vol. 47, no. 14, pp. 72-75,89, 2016.

[11] W. Yulin, L. Jintao, S. Yuekun et al, "Numerical analysis of flow in a Francis turbine on an equal critical cavitation coefficient line", *Journal of Mechanical Science and Technology*, vol. 27, no. 6, pp. 1635-1641, 2013.

[12] C.Trivedi, J. M. Cervantes, and B. K. Gandhi, "Investigation of a high head Francis turbine at runaway operating conditions", *Energies*, vol. 149, no. 9, pp. 1-22, 2016.

Liying Wang was born on Jan. 6, 1978, she received the PhD. degree in vehicle engineering from Beijng Jiaotong University, China in 2014. Now she is an associate professor and works in Hebei University of Engineering. Her current research interests include intelligent computing, control systems engineering, and intelligent fault diagnosis.

Bingyao Li was born on June 18, 1992, he is a graduate student at Hebei University of Engineering, his major research interests include fluid solid coupling and vibration analysis of hydraulic machinery.

Weiguo Zhao was born on Jan. 23, 1977, he received the PhD degree in electrical engineering from Hebei University of Technology, China in 2016. Currently, he is a researcher (assistant professor) at Hebei University of Engineering, China. His major research interests include intelligent computing, and intelligent fault diagnosis.

Qingjiao Cao was born on Sep. 9, 1990, she received the M.S. degree from hydraulic engineering in Hebei University of Engineering, China. Currently, she is a laboratory technician at Hebei University of Engineering, China. Her major research interests include automation detection and control.