Effect of Guide Vane Openings and Different Flow Rates on Characteristics in Pump Mode of Pump-turbine with Splitter Blades

Jinfeng Zhang¹, Yajing Xiao¹, Shibin Jin², Haikun Cai¹, Haiqing Song¹

¹Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, Zhenjiang, China
²Institute of Quality Supervision and Inspection, Wenling, China

E-mail:zhangjinfeng@ujs.edu.cn

Abstract. With the development of smart power grid, pumped storage has been developing on a large scale, and the pump-turbine with splitter blades is widely used. A combination of experimental and CFX numerical simulation was carried out to investigate the influence of guide vane openings on the flow characteristics in pump-turbine with splitter blades at different flow rates. The results show that change in the guide vane openings and flow rates have significant impacts on the internal flow characteristics in pump mode. The findings also revealed a poor flow stability in the condition of a small opening at a high flow rate and the condition of large opening at a low flow rate. Under the influence of pressure fluctuation, there exists local high-speed flow at the outlet of the draft tube and mid-span between the runner and the guide vane.

1. Introduction

Pumped storage power station uses the surplus power at the trough to pump water to the upstream reservoir for storage, and then uses it to supplement power at the peak load. It can also cooperate with other grid systems to carry out peak load regulation and valley filling [1-3]. It is recognized as a reliable peak load regulation power source in the world, so the stability of unit operation is crucial. The flow characteristic of conventional pump-turbine usually produces pressure disturbance and kinetic energy loss when it is in the working condition of the pump. However, its operational performance is characterized by challenges such as vibration, noise, mechanical fatigue, and so on, which are due to the unsteady flow state, pressure fluctuation, and energy loss during the operation. The interaction effect due to the guide vane opening and flow rate contributes to the pump external performance and flow characteristic [4-8].

Nicolet [9] carried out a one-dimensional acoustic simulation of the rotor-stator interaction of a mixed-flow pump-turbine under the resonant condition, and revealed the acoustic resonance mode and frequency characteristics generated by the rotor-stator interaction in the volute region, which laid a foundation for the simulation of the full acoustic model. Egusquiza [10] monitored the vibration of water turbine through the remote monitoring and control system and found that the vibration of the pump-turbine is much larger than the conventional hydraulic turbine, which is mainly because of rotor-stator interaction in the pump-turbine, especially in the working conditions of the turbine with a heavy load. Rodriguez[11] adopted the measurement method of sensor rotating along the shaft to
monitor the rotor-stator interaction of pump-turbine, which avoids the measurement influence brought by bearing response and provides a more reliable measurement method for monitoring the rotor-stator interaction characteristics. Casartelli [12] investigated a CFD simulation of the flow field and pressure fluctuation change combined with FSI technology during the guide vane closing process, which provided a convenient and accurate CFD strategy for analyzing the transient characteristics of the unstable conditions of the pump-turbine as well as developing the guide vane control rules. Yefang Wang [13] resolve the vortical flows in a typical single-stage side channel pump by using the unsteady Reynolds-averaged Navier–Stokes (URANS) and three hybrid Reynolds-averaged Navier–Stokes-large eddy simulation (RANS-LES) models, to evaluate the suitability of these advanced turbulence models in predicting the pump hydraulic performance and unstable swirling flows. Yonezawa [14] studied the phase resonance phenomenon in centrifugal rotating machinery. By comparing and analyzing the phase resonance in the working condition of the pump and turbine respectively, they revealed the important influence of the direction of rotor-stator interaction and the radial velocity pulsation on the phase resonance. Pochyly [15] studied experimentally the influence of inserting shunt blades on the pressure fluctuation of pump-turbine passage. It is observed that the shunt blade could enlarge the high efficiency area of the pump-turbine, and the pressure pulsation could also disperse to a lower frequency range in the working condition of the turbine. Meanwhile, the pressure fluctuation could be significantly reduced in the working condition of the pump, especially in the unstable area. Shigemitsu [16] studied the performance and internal flow characteristics of a small centrifugal pump with a large blade outlet angle by using a shunt blade using experimental and numerical works. Namazizadeh [17] discussed the influence of diverging blades with different lengths, leading edge positions and distances on centrifugal pump head and efficiency, and adopted experimental design technology (DOE) to establish optimal design space, and used response surface method to calculate the optimal geometric size. Fan Zhang [18] reveals the exact regions of high hydraulic losses for side channel pump models using the entropy loss production method and contributes to the fundamental knowledge of side channel pumps by providing the regions of high-energy losses for further optimization of the main geometrical parts especially the impeller.

Several studies have demonstrated that numerical simulation had a reasonable accuracy in calculating the flow characteristic of the pump-turbine. Therefore, unsteady numerical simulation, performance test, and CFX test for pump-turbine with different splitter blades are carried out at different guide vane opening at different operating flow rates in the present study. The test results are compared with the numerical works to illustrate the reliability of the calculation. Furthermore, using analyzing the pressure fluctuations, streamlines, and turbulence kinetic energy in the different parts of the pump-turbine, the impacts of the guide vane opening on the flow characteristic at different flow rates are investigated to provide some references in the design of guide vane opening.

2. Numerical simulation and tests

2.1. Description of geometry model

The model of pump-turbine with splitter blades in this study is scaled by 1:7.47 for a pump-turbine in a domestic pumped storage power station. The basic parameters of the pump-turbine model are shown in Table 1.

| Table 1. Basic parameters of the pump-turbine model. |
|-----------------------------------------------------|
| Parameter                                           | Data     |
| Number of stay vanes                               | 16       |
| Number of guide vanes                              | 16       |
| Number of impeller vane                            | 10       |
| impeller outlet minimum diameter D₁(mm)            | 300      |
| Impeller outlet maximum diameter D₂(mm)            | 584      |
As pump-turbine is a reversible unit, two working modes should be taken into account. The pump mode is that the runner rotates anticlockwise, and the flow firstly enters from the draft tube. After the work and activity of the runner, it is then guided into the volute by the double-row cascade with a fixed guide blade. At that time, the casing converts the kinetic energy of the fluid to static pressure energy, to pump the water to the upstream reservoir. Similarly, the turbine mode is that the flow enters via the inlet nozzle. Flow rate of water will get reduced along the length of the volute, thus decreasing the area of volute will ensure a uniform velocity of the flow as it enters. Stay vanes and guide vanes are fitted at the entrance of the runner to convert one part of pressure energy to kinetic energy. Stay vanes steers the flow towards the runner section thus reducing the swirl at the inlet flow. In the runner, water enters radially and leaves axially, while both impulse force and lift force make the runner rotate by clockwise. During the flow, water glides over runner blades. The blades have a thin airfoil cross-section and when water flows over it, a low pressure is produced on one side and high pressure on the other side which subsequently results in a lift force which will hit and produce an impulse before leaving the runner and ends up flowing out from the draft tube. Lastly, the runner is connected to a generator via a shaft for electricity production. In this paper, Creo software is used to carry out 3D modeling and the full flow geometric model is shown in Figure 1.

**2.2. Transient computation procedure**

The characteristics of structured mesh is that the nodes can be named orderly with fixed rules, which means that for small computations, the grid quality becomes easy to control, the boundary layer grid is guaranteed, and the convergence of computation is enhanced. The whole computational domain was split into five parts for modeling which are the volute, stay vanes, guide vanes, draft tube, and runner. The hybrid structured meshes were generated using mesh generation tool ANSYS-ICEM to the grid quantity, quality, and boundary layer to meet the requirements. Three groups of meshes were categorized into three groups of guide vane passages at the opening of 9.8°, 17.5° and 24.8° selected in this paper. The mesh quality was higher than 0.5, and the angles were all greater than 25°.

Figure 2 shows the boundary layer details of the volute section, the boundary layer details of the runner blades, and the single channel grid of the guide blade at different guide vane opening 9.8°, 17.5° and 24.8° respectively.

| runner outlet diameter (mm) | 55 (working condition) |
| volute outlet diameter (mm) | 320 (working condition) |
| Draft tube outlet diameter (mm) | 600 (working condition) |
The numerical calculations for different schemes were performed in the fluid simulation code ANSYS CFX by using the unsteady Reynolds-Averaged Navier-Stokes which is based on the finite volume method. [19,20] The boundary conditions in the calculation are shown in the Table 2.

| Boundary Conditions                  | Settings                                      |
|-------------------------------------|-----------------------------------------------|
| Turbulence model                    | Shear Stress Transport (SST)                   |
| Reference pressure                  | 101,325Pa                                     |
| Inlet                               | Total pressure                                |
| Outlet                              | Mass flow rate                                |
| Interface                           | Frozen-rator                                  |
| Wall roughness                      | Smooth                                        |
| Mass and momentum                   | No Slip                                       |
| Advection Scheme                    | Upwind, High Resolution                       |
| Convergence criterion               | $10^{-5}$                                      |

### 2.3. Mesh independence analysis

Since the number of mesh has a great impact on the calculation results, the mesh number independence analysis was performed for 5 sets of grids. According to the rated operating condition of 17.5° opening of the guide vanes, 8 groups of mesh number schemes were set at uniform intervals between 4 million and 10 million as shown in Table 3.

With the increase of mesh number, the relative head error tends to 1 as shown in Figure 3 which validates the reliability and accuracy of the simulation results. When the total number of grids in each computational domain is more than 7 million, the results of external characteristics remain stable and the number of grids tends to be independent. Therefore, Case 4 is selected, and the total number of grids is 7213,000.

| Total(10^6) | $\Delta H(\%)$ | Total(10^6) | $\Delta H(\%)$ |
|-------------|----------------|-------------|----------------|
| Case1       | 4.266          | 2.732       | Case5          | 8.181          | 0.826 |
| Case2       | 5.184          | 2.093       | Case6          | 9.202          | 0.825 |
| Case3       | 6.153          | 1.285       | Case7          | 10.170         | 0.824 |
| Case4       | 7.213          | 0.828       |                |                |      |
3. Experimental validations

To study the external characteristics of the pump-turbine in different opening conditions, opening 9.8°, 17.5° and 24.8° are selected for analysis in this paper. The flow rate and head are normalized as flow rate coefficient $\phi$ and head coefficient by the following equations:

$$\psi = \frac{2gH}{u_2^2}$$
$$\phi = \frac{Q}{u_2R_2^2}$$

$H$—head; $u_2$—outlet circumferential velocity of runner; $g$—acceleration of gravity; $Q$—flow rate; $R_2$—radius of high pressure side of runner

As showed in Figure 4, the simulation head and efficiency both revealed a satisfactory agreement with the test. The maximum error of the lift coefficient in each condition is less than 4.5%, and the maximum error of the efficiency is less than 3.2%. Therefore, the prediction correctness of CFD calculation strategy in this paper can be evidenced, and the calculation results can be used as a benchmark for analysis. Furthermore, it is apparent from the flow-head curve that the head is large when the opening is small and the flow is small, the head is small when the opening is large and the flow is large; hump phenomenon exists in the vicinity of $0.5Q_d$-$0.75Q_d$, especially when the opening is large. It can be seen from the flow-efficiency curve that the small opening is more efficient at a low flow rate while a large opening is more efficient at a high flow rate, and the small opening is more sensitive to the change of flow rate. With the increase of the opening, the rated flow point $1.0Q_d$ is shifted towards the large flow, and the high efficiency interval becomes wider. When the guide vane opening is 17.5°, the rated flow point had the highest efficiency, signifying that different flows have the corresponding optimal opening and too large or too small guide vane opening will lead to additional energy loss.
Figure 4. Comparison of performance between simulation and experiment.

4. Results and discussion

Three operating conditions corresponding to the above three different openings, namely $0.75Q_d$ for small flow, $1.0Q_d$ for rated flow and $1.25Q_d$ for large flow, were selected respectively to analyze the internal flow characteristics of each channel and passage in different operating conditions of the pump with different guide vane openings and flows rates. Specific operating condition point parameters are shown in Table 4.

| Opening | $0.75Q_d$ | $1.0Q_d$ | $1.25Q_d$ |
|---------|-----------|-----------|-----------|
| 9.8°    | $\varphi=0.0760$ | $\varphi=0.1013$ | $\varphi=0.1266$ |
| 17.5°   | $\varphi=0.0964$ | $\varphi=0.1285$ | $\varphi=0.1606$ |
| 24.8°   | $\varphi=0.1149$ | $\varphi=0.1532$ | $\varphi=0.1915$ |

4.1. Analysis on the mid-span between the runner and stay-guide vanes

Two different working conditions of the pump-turbine were taken into account, the double row cascade adopts the combination form of guide vanes and stay vanes in the turbine. The cascade plays the role of regulating diversion flow to ensure that the inflow of the runner inlet has an appropriate ring volume. While in the working condition of the pump, the double row cascade and volute transform the dynamic pressure of the runner's high-speed flow into the static pressure energy. The rotor-stator interaction in the bladeless region is stronger than that in the working condition of the turbine. Flow instability is more likely to occur at the mid-span between the runner and stay-guide vanes.
Figure 5 displays the streamline distribution of the flow surface streamlines at the mid-span between the runner and stay-guide vanes. The flow surface in the middle of the runner is set as span=0.5 by ‘Blade to Blade’. It is apparent that the streamlines distribution in the runner is relatively uniform, and no vortex flow occurred at the inlet of the conventional blades, indicating that the design of splitter blades can improve the inflow of the runner and reduce the flow loss in the passage of runner. It was also revealed that at a low flow rate of 0.75Qd, separation vortex congestion occurs in the flow passages of stay vanes of double-row cascade at different openings. When the guide vane opening is large, the impact angle between the guide vanes and the inlet flow tends to be larger, leading to the separation of the vortex generated in advance to the flow passage of the guide vanes. Consequently, the flow velocity and hydraulic loss of adjacent guide vane passages are greatly increased. At rated flow rate 1.0Qd, the flow state in the condition of 17.5°exhibited the most stable small energy loss and higher efficiency. However, the vortex separation phenomenon still exists in the stay vane passage with a small opening and the guide vane passage with a large opening leading to large energy loss and reduced efficiency.

Therefore, it indicates that 17.5°is the optimal opening in the working condition of the pump. When the flow rate is 1.25Qd, the overflow capacity of both the runner and the double-row cascade is improved. The flow distribution in all channels with a large opening is similar. Under the influence of rotor-stator interaction, high-speed flows are mainly concentrated in the bladeless region, while the small opening will block the runner's outflow resulting in obvious flow separation in the stay vane channel near the tongue.
Figure 5. Streamlines at the mid-span in the runner and stay-guide vanes.

To visually reflect the energy loss caused by complex flows in the runner and double-row cascade, Figure 6 shows the turbulent kinetic energy distribution at the mid-span between the runner and stay-guide vanes. It can be seen that: at a low flow rate $0.75Q_d$, with a small opening, the high turbulent kinetic energy region is mainly bladeless region and stay-guide vane passage. However, the flow separation becomes problematic when the opening becomes large, which leads to greater turbulent kinetic energy and extremely uneven distribution at the mid-span between the runner and stay-guide vanes. In the cascade channel near the tongue, the energy loss is relatively apparent, indicating that it should try to avoid operating with the large opening at a low flow rate. At rated flow rate $1.0Q_d$, the flow state becomes stable with $17.5^\circ$ opening, resulting in smaller turbulent kinetic energy, whilst higher turbulent kinetic energy region at a small opening is transferred to the stay vane channel. However, in the condition of large opening affected by the interaction of flow separation and rotor-stator interaction due to the increased impact angle, there exist 5 turbulent kinetic energies in the guide vane passage accompanied by intense energy exchange. At high flow rate $1.25Q_d$, the turbulent kinetic energy of the stay vane passage with a small opening is further strengthened, the turbulent kinetic energy is small with a large opening and the overall turbulent kinetic energy of each channel is evenly distributed, and the energy loss is reduced at the same time. In general, with the increase of the flow rate, the turbulent kinetic energy of the stay vane passage at the small opening is gradually intensified, while the turbulent kinetic energy at the large opening is gradually decreased, indicating that the flows at small and large openings are the most disordered while the energy losses are the most severe.
Figure 6. Turbulence kinetic energy at the mid-span between the runner and stay-guide vanes.

4.2. Analysis on the draft tube

Figure 7 shows the streamline distribution of the draft tube and different sections at the rated flow rate at 17.5°. Section A is the outlet of the draft tube, section B is the inlet of the straight taper tube (500mm away from section A), and section C is the normal section of the elbow tube along the profile. As shown in Figure 7, the flow is relatively regular in the working condition of the pump. The flow through the elbow can change the flow direction more smoothly, which is greatly affected by the impact on the outer wall of the elbow, and the flow velocity near the inside of the elbow which is relatively large.

From the streamline distribution of section C, it is also observed that, due to the impact of the outer wall of the elbow, the water collects from the outer wall to the inner wall, resulting in more dense streamlines at the middle position of the inner wall. From the streamline distribution of section B, it was also revealed that the inlet of the cone section is influenced by the centrifugal inertia force of the upstream cubital tube and the viscosity of the tube wall, hence leading to the formation of eddy currents in a small area near the outside of the tube wall. From the streamline distribution of section A, it was also revealed that, affected by the rotor-stator interaction, the coarse flow line of the draft tube presents 5 similar eddy current regions.

Figure 7. Streamlines distribution of draft tube at 1.0Qd (17.5°).

To analyze the influence of rotor-stator interaction on the outlet of the draft tube in different working conditions, Figure 8 shows the streamline and pressure distribution of the draft tube outlet of section A in different working conditions. It can be seen that, when the flow rate is 0.75Qd and the draft tube with small guide vane opening, the pressure distribution becomes relatively regular. Affected by the rotor-stator interaction especially long blades, there exist 5 high-speed watersheds formed at the tube wall. Therefore, low-pressure areas and high-pressure areas are formed, while the flow state at the outlet of the draft tube with large guide vane opening becomes unstable, resulting in uneven and irregular distribution of the eddy current. With the increase of the opening as well as the flow rate, the flow distribution becomes more regular and the pressure changes tend to be more evenly. The high-pressure area gradually shifts to the center, demonstrating that the flow state at the outlet of the draft tube is more stable with the large opening at a high flow rate. In general, the outlet of the draft tube is most significantly affected by the inlet edge of the long blades with a small opening at a low flow rate. The flow distribution is more uniform with a large opening and a large flow rate, while the flow distribution is the most unstable with a large opening at a low flow rate, hence resulting in irregular flow line and pressure distribution.
Figure 8. Streamline and Pressure distribution of draft tube outlet.
4.3. Analysis on the volute

In the working condition of the pump, the large section may not match the diffusion degree of the incoming flow, and the outflow of double-row cascade becomes disordered, leading to poor flow in the volute runner than in the working condition of turbine.

To analyze the flow characteristics of the volute, four volute flow passage sections were intercepted at a uniform angle in the circumferential direction, as shown in Figure 9.

Figure 9. Diagram of volute section location.

Figure 10 shows streamlines in volute sections at 1.0 \( Q_d \). It is observed that the inlet of the volute is perpendicular to all sections in the working condition of the pump. Water then spreads in an axial direction after entering the volute, which results in vortex rings at different sections. At the rated flow rate 1.0 \( Q_d \), both sections a-a and b-b are evenly distributed on both sides, while vortex appears in the middle line of sections c-c and d-d. It shows that the flow distribution in the large section is more uniform at the rated flow rate, but at the same time, the backflow blockage in small section becomes worse. The flow differences between different sections are the most significant when the guide vane opening is small. With the increase of the openings, the flow in the volute section tends to a more roughly symmetrical distribution of backflow vortex on both sides.
Figure 10. Streamlines in volute sections at 1.0Qd.

5. Conclusion

In this paper, the influence of different guide vane opening on the flow characteristic in a pump-turbine with splitter blades has been investigated at different flow rates 0.75Qd, 1.0 Qd, and 1.25 Qd. Performance and experimental data records both revealed a satisfactory agreement with the numerical simulation under different guide vane opening 9.8°, 17.5° and 24.8°. Several monitor cross-sections were installed in the draft tube and mid-span between the runner and stay-guide vanes to capture the streamline and pressure signals during the transient numerical simulations. By analyzing the streamline and pressure distribution of draft tube and turbulence kinetic energy at the mid-span between the runner and stay-guide vanes at different rates and different openings, some conclusions are made as follows:

(1) Compared with different openings at different flow rates from 0.75 Qd to 1.25 Qd, the phenomenon of hump becomes more obvious when the opening is large and efficiency is more sensitive to changes in flow rates, specifically in a small guide vane opening. However, with the increase of the guide vane opening, the rated flow point 1.0Qd shifts towards the large flow rate and at the same time, the high efficiency interval becomes wider, indicating that too large or too small opening will lead to additional energy loss. In this paper, the efficiency of the pump-turbine rated flow point is the highest at the opening 17.5°, which indicates that 17.5° is the optimal opening of the pump working condition.

(2) Compared with streamline, pressure distribution, and turbulence kinetic energy at different openings at different flow rates from 0.75Qd to 1.25Qd, changes in the guide vane opening and flow rate had a significant impact on the internal flow in the working conditions of the pump. The flow stability under the small opening and large flow tends to be poor. Especially at the small opening, the hydraulic loss caused by flow separation in the double-row cascade becomes very severe. Finally, under the influence of rotor-stator interaction, there exists a local high-speed flow at the outlet of the draft tube and bladeless region between the runner and stay-guide vanes.
References

[1] Haban V, Koutnik J, Pochyly F. 1-D Mathematical model of high-frequency pressure oscillations induced by RSI including an influence of fluid second viscosity[C]. Proceedings of the 21st IAHR Symposium on Hydraulic Machinery and Systems, 2002.

[2] Rodriguez C, Egusquiza E, Santos I. Frequencies in the Vibration Induced by the Rotor Stator Interaction in a Centrifugal Pump Turbine[J]. Journal of Fluids Engineering, 2007, 129(11): 1428-1435.

[3] Nicolet C, Zobeiri A, Maruzewski P, et al. On the upper part load vortex rope in Francis turbine: Experimental investigation[C]. IOP Conference Series Earth and Environmental Science, 2010: 793-798.

[4] Keller J, Blanco E, Barrio R, et al. PIV measurements of the unsteady flow structures in a volute centrifugal pump at a high flow rate[J]. Experiments in Fluids: Experimental Methods and Their Applications to Fluid Flow, 2014, 55(10): 1820-1-1820-14.

[5] Chitrakar S. Fully coupled FSI analysis of Francis turbines exposed to sediment erosion[J]. International Journal of Fluid Machinery and Systems, 2014, 7(3):101-109.

[6] Ali Rezghi, Alireza Riasi. The interaction effect of hydraulic transient conditions of two parallel pump-turbine units in a pumped-storage power plant with considering “S-shaped” instability region: Numerical simulation[J]. Renewable Energy, 2018, 118.

[7] Yabin Liu, Yadong Han, Lei Tan, Yuming Wang. Blade rotation angle on energy performance and tip leakage vortex in a mixed flow pump as turbine at pump mode[J]. Energy, 2020, 206.

[8] Zobeiri A, Kueny J L, Farhat M, et al. Pump-Turbine Rotor-Stator Interactions in Generating Mode: Pressure Fluctuation in Distributor Channel[C]. Proceedings of 23th IAHR Symposium on Hydraulic Machinery and Systems, Yokohama, Japan, 2006.

[9] Nicolet C, Zobeiri A, Maruzewski P, et al. On the upper part load vortex rope in Francis turbine: Experimental investigation[C]. IOP Conference Series Earth and Environmental Science, 2010: 793-798.

[10] Rodriguez C, Egusquiza E, Santos I. Frequencies in the Vibration Induced by the Rotor Stator Interaction in a Centrifugal pump-turbine[J]. Journal of Fluids Engineering, 2007, 129(11): 1428-1435.

[11] Rodriguez C G, Mateos-Prieto B, Egusquiza E. Monitoring of Rotor-Stator Interaction in Pump-Turbine Using Vibrations Measured with On-Board Sensors Rotating with Shaft[J]. Shock and Vibration, 2014, 2014(pt.1): 276796.1-276796.8.

[12] Casartelli E, Mangani L, Ryan O, et al. Application of transient CFD-procedures for S-shape computation in pump-turbines with and without FSI[J]. Iop Conference, 2016, 49(4): 042008.

[13] Yefang Wang, Fan Zhang, Shouqi Yuan, et al. Effect of Unrans and Hybrid Rans-Les Turbulence Models on Unsteady Turbulent Flows Inside a Side Channel Pump, ASME Journal of Fluids Engineering, 2020, 142(6): 061503, doi:10.1115/1.4045995

[14] Yonezawa K, Toyahara S, Motoki S, et al. Phase Resonance in Centrifugal Fluid Machinery -A Comparison between Pump Mode and Turbine Mode Operations and a Discussion of Mechanisms of Flow Rate Fluctuation through a Stator-[J]. International Journal of Fluid Machinery and Systems, 2014, 7(2): 42-53.

[15] Pochyly F, Haluza M, Veselý J. The Francis Pump Turbine With Stochastic Blades[J]. Procedia Engineering, 2012, 39: 68-75.

[16] Shigemitsu T, Fukutomi J, Wada T, et al. Performance Analysis of Mini Centrifugal Pump with Splitter Blades[J]. Journal of Thermal Science, 2013, 22(6): 573-579.

[17] Namazizadeh M, Talebian Gevari M, Mojaddam M, et al. Optimization of the Splitter Blade Configuration and Geometry of a Centrifugal Pump Impeller using Design of Experiment[J]. Journal of Applied Fluid Mechanics, 2020, 13(1): 89-101.

[18] Fan Zhang, Desmond Appiah, Feng Hong, et al. Energy loss evaluation in a side channel pump
under different wrapping angles using entropy production method, International
Communications in Heat and Mass Transfer, 2020 (113).

[19] Menter F R. Two-equation eddy-viscosity turbulence models for engineering applications[J].
AIAA journal, 1994, 32(8): 1598-1605.

[20] Hinze J O. Turbulence[M]. New York: McGraw-Hill Publishing Co., 1975.