RESEARCH ARTICLE

Experimental and numerical investigations on pressure pulsation in a pump mode operation of a pump as turbine

Jinfeng Zhang  |  Desmond Appiah  |  Fan Zhang  |  Shouqi Yuan  |  Yandong Gu  |  Stephen N. Asomani

National Research Center of Pumps, Jiangsu University, Zhenjiang, China

Correspondence
Fan Zhang, National Research Center of Pumps, Jiangsu University, Zhenjiang, 212013, China.
Email: fzhang@ujs.edu.cn

Funding information
National Key R&D Program Projects of China, Grant/Award Number: 2018YFB0606101; Natural Science Foundation of China, Grant/Award Number: 51809121; Natural Science Foundation of Jiangsu Province, Grant/Award Number: BK20170548; Senior Talent Research Start-up Foundation of Jiangsu University, Grant/Award Number: 17JDG04

Abstract
Hydropower has been the leading renewable source and cheapest ways to generate electrical energy in the world. In recent years, there has been a major upsurge in the hydropower development because of the use of pump as turbine (PAT). However, the operational reliability of a PAT is greatly affected by the unsteady flow fields; therefore, it is important to examine the unsteady flow behavior which can be used as a reference to reduce the noise, vibration, and cavitation performance for centrifugal pumps working as turbines. Thus, the objective of this study was to evaluate the unsteady flow fields by analyzing the distribution of the pressure pulsations using both numerical and experimental measurements in a PAT operating in pump mode. Firstly, the three-dimensional (3D) unsteady flow equations were solved using SST k-ω turbulence model during the numerical calculations. Secondly, the numerical results of the hydraulic pump performance were validated by the experimental measurements for numerical accuracy. Lastly, pressure transducers are positioned at certain monitoring points to measure the pressure in the PAT investigated. The numerical and experimental results show that the main frequency of the pressure pulsation is equal to the blade frequency, and as it deviates from the design operating condition, the magnitude of pressure pulsation intensifies. Furthermore, the impeller eye marked the lowest pressure coefficients especially at the design condition and makes it highly susceptible to cavitation. High pressure coefficients were obviously seen at the pressure side on the blade surface closer to the trailing edge at all studied operating conditions. Meanwhile, the rotor-stator interaction generated the highest pressure pulsation distribution at the volute tongue. Thus, modification of the volute tongue is an optimal approach of reducing the pressure pulsation intensity in the volute and pump as a whole.

KEYWORDS
experiment, frequency analysis, numerical simulation, pressure pulsation, pump as turbine
1 | INTRODUCTION

Hydropower is the most important renewable source to the primary energy supply. It represents more than 71% of all the worldwide renewable energy and continues to stand as the most important renewable energy source. A pump working as turbine (PAT) is a good substitute for power generation through small and micro hydropower schemes because of its unique advantages. These include relatively simple construction, inexpensive cost of supply, and the possibility to regulate the flow without difficulties. When a centrifugal pump runs as a turbine, high pressure fluid enters the machine and then leaves after the conversion of the energy, but in pump mode the rotating impeller increases the angular momentum by imparting energy on the flow, thereby increasing the pressure at the outlet. Figure 1 shows a typical PAT operating in the pump mode.

With the fast development of small hydropower nowadays, the operational reliability of a PAT is key and requires strict manufacturing requirements. However, the pump mode operation plays very fundamental roles in solving different related operational challenges (mainly cavitation and flow instabilities). The rotor-stator interaction between the impeller and the volute creates strong pressure pulsation, which not only affects the hydraulic performance but also causes noise, vibration, and harshness (NVH) in the hydropower plant. This subsequently affects its longevity. Hence, it is very important to examine the causes of pressure pulsations to improve the overall performance and reliability. Several researches have studied the unsteadiness of the flow in hydraulic pumps and among them are as follows: Minggao et al employed the numerical simulations to investigate the inner flow patterns of six different models of centrifugal pump at different working conditions. Jafarzadeh et al simulated the three-dimensional (3D) pressure and velocity intensity fields in a low-specific speed centrifugal pump. Pei et al numerically predicted the unsteady pressure field within the whole flow passage of a radial single-blade pump. The rotor-stator interactions in radial centrifugal pumps were studied by Arndt et al. The pressure pulsation distribution in the impeller and diffuser was analyzed. Yuan et al used computational dynamics fluids (CFD) codes to calculate the flow field including the suction chamber, impeller, and volute flow channel to obtain the pressure pulsations characteristics. Guo and Maruta used the high speed pressure transducers to measure the magnitude of the pressure pulsations caused by the impeller-volute interactions. González et al numerically predicted the unsteady flow fields in a centrifugal pump. They showed that the variations in the pressure pulsation distribution in the impeller are a result of the blade-tongue interactions with the flow at the impeller outer radius. Khalifa et al, Pavesi et al, and Spece all investigated the pressure pulsation fields in centrifugal pumps and concluded that the magnitude of the pulsation is induced by the rotor-stator interactions at all operating conditions.

Moreover, the pressure pulsations induced by the rotor-stator interactions in axially split centrifugal pumps were investigated by Yao et al at design and off-design operation conditions. They demonstrated that the blade passing frequency is very high around the volute region. Cong and Wang used the large eddy simulation to study the unsteady flow fields at the volute tongue in a double suction centrifugal pump. Tan et al demonstrated that at the design operating condition, the unsteady pressure and velocity pulsations seem relatively uniform at the volute tongue of a centrifugal pump. Yang et al examined the influence of the volute tongue on the pressure pulsations of a PAT. It was concluded that as the radial clearance increases, the magnitude of the pressure pulsation at the volute tongue decreases. Yang et al studied the unsteady flow characteristics regarding hump instability in the first stage of a multistage pump-turbine in pump mode. The parametric uncertainty methods and its application can be reference to Xu et al.

From the above literature, most studies on the pressure pulsations at various locations within the pump were carried out only through numerical methods failing to validate with their respective experimental measurements. Moreover, since no operational data are provided for the turbine mode from the suppliers, many investigations on PAT mostly emphasize on the development of accurate prediction models for the turbine operation based on different designs of centrifugal pumps and their design optimization.

Therefore, the improvement of the operational reliability of a PAT by analyzing the unsteady flow behavior in the pump mode is very useful. This study aims to examine, analyze, and predict regions of unsteady pressure pulsation distributions in the pump mode of a PAT. Firstly, the pressure component...
The above Reynolds-Averaged Navier-Stokes (RANS) equations in the pump are solved using k-ω-turbulence model. This model is chosen because of its fast convergence and capabilities to predict free-stream and inner wall-layered flows. The transport equations for turbulent kinetic energy, \( k \), and dissipation rate, \( \omega \), in SST k-ω model are expressed below with references to Menter \(^{24,25}\) and Wilcox \(^{26,27}\) k-ω equations

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \rho u_i \frac{\partial u_i}{\partial x_j} \right] - \frac{\partial}{\partial x_j} \left( \rho \frac{\partial u_i}{\partial x_j} \right) + \rho \frac{\partial u_i}{\partial x_j} - \frac{1}{3} \delta_{ij} \left( \rho k + \mu \frac{\partial u_i}{\partial x_j} \right) \]

where \( \rho \) is the fluid density, \( u_i \) is the velocity component in the \( i \)-direction, \( k \) is the turbulent kinetic energy, \( \omega \) is the dissipation rate, \( \omega_{ij} = \frac{\partial u_i}{\partial x_j} \) is the shear stress, \( \sigma_{ij} = -\frac{2}{3} \delta_{ij} \left( \rho k + \mu \frac{\partial u_i}{\partial x_j} \right) \) is the turbulent viscosity, \( \mu \) is the molecular viscosity, and \( \delta_{ij} \) is the Kronecker delta.

\[
\sigma_{ij} = -\frac{2}{3} \delta_{ij} \left( \rho k + \mu \frac{\partial u_i}{\partial x_j} \right)
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho U_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial \omega}{\partial x_j} + \frac{\partial \omega}{\partial x_i} \right) - \rho \frac{\partial u_i}{\partial x_j} \frac{\partial \omega}{\partial x_j} \right] + \alpha_1 \frac{\partial u_i}{\partial x_j} \frac{\partial \omega}{\partial x_j} + \alpha_2 \frac{\partial u_i}{\partial x_j} \frac{\partial \omega}{\partial x_j} \]

The blending functions \( F_1 \) and \( F_2 \) are:

\[
F_1 = \tanh \left( \Gamma_1^4 \right)
\]

\[
F_2 = \tanh \left( \Gamma_2^2 \right)
\]

The turbulent viscosity, \( \mu_t \), is calculated using:

\[
\mu_t = \alpha_1 k \rho \max \left( \alpha_1 \omega_{ij} \sigma_{ij}^2 \right)
\]

The k-ω equation for the inner boundary layer prediction is applied using the following coefficients: \( \sigma_{k1} = 1.176, \sigma_{\omega1} = 2, \beta_1 = 0.075, \alpha_1 = 5/9, \beta' = 0.09 \).

Meanwhile, the k-ε equation for the free-stream flows is solved using the following coefficients: \( \sigma_{k2} = 1.0, \sigma_{\omega2} = 1/0.856, \beta_2 = 0.828 \).
2.3 | Pump specifications

The investigated pump is designed with Siemens NX 11 as shown in Figure 2. The pump model is a six-bladed closed radial impeller which operated at a rotational speed of 1480 r/min with specific speed \( n_s = \frac{nQ^{0.5}}{H^{0.75}} \) of 2.3 at the design flow point. The nominal flow rate and head at the design flow point are 6.3 m\(^3\) h\(^{-1}\) and 8.0 m, respectively. Other main parameters are summarized in Table 1, while Figure 3 shows a detailed meridian view of the pump impeller.

2.4 | Mesh generation

The computational flow domains in the pump are divided into three main parts, namely suction pipe, impeller, and volute and transported into ICEM CFD 14.0 to generate the mesh. The hexahedral structured grid is used by applying the blocking method. Figure 4 shows the top view on mesh topology of the impeller detailing the hub region. At the surface of the suction and pressure sides of the blade, the size of the grids at impeller surface is 0.01 mm. This made the mesh finer with a \( y^+ \) less than 20 and capable of capturing the detailed flow behavior of the boundary layer.

In contrast to the generation of the mesh of the impeller blades, the O-Grid blocking method is used to generate the mesh of the volute. At the inner volute diameter, the cells are gridded with size of 0.01 mm in Figure 5. The tongue region is refined with high grid numbers due to the sudden change in geometry.

The effect of the grid number on the accuracy of the numerical calculations cannot be neglected; thus, it is vital to carry out a mesh-independence analysis on the generated meshes. This is done to determine the optimum grid numbers suitable for the numerical calculations. Steady simulations of six different grids ranging from 1.3 million to 6.5 million are carried out for the mesh-independence analysis. Figure 6 shows the mesh-independence analysis carried out using six different grid numbers. Mesh D with total grid number of 4.1 million elements is selected based on Celik et al\(^{28}\) procedures for estimating and reporting of discretization error in CFD application. The number of elements of impeller, volute with discharge, and suction pipe flow domains are 2.2, 1.6, and 0.3 million, respectively. The pump head is calculated using Equation 12. The head is determined by finding the mean of all the head values at each time step for the last three complete impeller rotations of the steady numerical simulations.

\[
H = \frac{P_{\text{out}} - P_{\text{in}}}{\rho g}
\]  

(11)

2.5 | Steady and unsteady numerical calculations

To investigate the developed unsteady flow characteristics resulting from the pressure distribution within the full flow field of the investigated PAT model, both steady and unsteady numerical calculations of the turbulent flows under different operating flow conditions are carried out. The steady-state calculations are performed with the interface model set to frozen rotor. The fluid used is water at 25°C. The boundary condition of the suction pipe inlet is defined as total pressure of 1 atm while that of the discharge pipe outlet is defined as mass flow. For the steady-state calculations, a convergence criterion of \( 1.0 \times 10^{-6} \) is chosen. The flow was directed normal to the boundary condition with turbulence intensity of 5%. The wall roughness of every boundary of each domain is defined as “smooth wall,” while the mass and momentum setting is set to “no slip wall.”

The impeller is set to operate at 1450 r/min, and 1500 iterations

| TABLE 1 | Parameters of the pump |
|-----------|-------------------------|
| Parameters | Value | Unit |
| Nominal head, \( H \) | 8 | m |
| Nominal flow rate, \( Q \) | 6.3 | m\(^3\)/hr |
| Specific speed, \( n_s \) | 2.3 | – |
| Blade numbers, \( Z \) | 6 | – |
| Inlet radius, \( R_1 \) | 25 | mm |
| Outlet radius, \( R_2 \) | 80 | mm |
| Leading edge radius, \( R_L \) | 18 | mm |
| Impeller outlet, \( b \) | 6 | mm |
| Shroud radius, \( R_s \) | 10 | mm |
| Hub radius, \( R_H \) | 20 | mm |
| Hole diameter, \( D_H \) | 5 | mm |

FIGURE 3 | Meridian view of the impeller geometry
are calculated in the steady state. The solutions from the steady calculations are to compute the hydraulic performance of the pump as shown in Figure 8. However, to capture more detailed characteristics of the flow the interface model is updated to transient-rotor-stator and the solutions from the steady-state calculations are used to initiate the transient state calculations. The impeller is set to rotate at a time step of 0.000114943 s representing one degree of each impeller turns. The total time for the entire calculation is 0.248 s depicting six revolutions of the impeller. The settings of the steady and unsteady calculations are done with reference to Pei et al.5

3 | EXPERIMENTAL SETUP

A test rig is set up at the National Research Center of Pumps at the Jiangsu University, China, as depicted in Figure 7. To measure the pressure pulsations at the suction and discharge, three pressure sensors from WIKA suppliers with accuracy of 0.15% are used. One is installed on the suction pipe, whereas other two sensors are installed on the discharge. To separate the two on the discharge pipe, one sensor is defined as pressure-sensor-north ($PS_{\text{north}}$) and the other at the opposite side is pressure-sensor-south ($PS_{\text{south}}$). At each opening of the valve, 2000 data samples are acquired. The measured data are sent via a data acquisition (DAQ) device for further data processing. The data are collected twice at each operating condition for reliability.
4 | RESULTS AND DISCUSSION

4.1 | Hydraulic performance characteristics

As mentioned later in Section 2.2, the head and efficiency are determined by averaging all values at each time step for the last three complete impeller rotations of the steady numerical simulations. The head and efficiency are calculated using Equations 12 and 13, respectively.

\[ \eta = \frac{\rho g Q H}{M \omega} \]  

Figure 8A shows the mean head characteristics curve with respect to the flow rate ratio. The simulated and experimental head curves decrease gradually with an increase in the flow rate. As it can be seen, the simulated values are higher than the experimental results at all operating conditions. At 0.6 and 1.9 \( Q_D \), the simulated heads mark 8.55% and 11.88% more than the experimental head, respectively. The largest deviation occurred at the design condition at 7.85%. Figure 8B depicts the mean efficiency curve vs the flow rate. The simulated values of the efficiency recorded higher values than the experimental at all operating conditions. This can be attributed to the neglect of mechanical losses on the flow. The disc friction loss in low-specific speed pump is dominant, which mainly influences the efficiency prediction. From the efficiency curves, the efficiency increases as the flow increases until it reaches the maximum peak (BEP). The maximum peak is flat; hence, the BEP is assumed in between 1.0 and 1.9 \( Q_D \).

From the above discussions, the pump under investigation is designed to operate at part-load operating condition. To end with, the numerical method employed in this work is feasible and shows a good agreement with the experimental data. Thus, part-load, design load, and overload conditions at flow rate ratios 0.6, 1.0, and 1.9 \( Q_D \), respectively, are chosen for the detailed pressure pulsations analyses in the sections below.
4.2 | Analysis of internal flow

4.2.1 | Static pressure distribution

Figure 9 shows the static pressure distribution in the impeller and volute at the part-load, design load, and part-load conditions at 0.6, 1.0, and 1.9 $Q_D$, respectively. At impeller eye, the static pressure reached the lowest value. From leading edge to trailing edge, the pressure increases gradually in radial direction and this is due to the asymmetric shape of the volute. At the trailing edge, a pressure drop is observed as the blade approaches the tongue. Meanwhile, at the volute outlet and discharge pipe the static pressure reached the highest value. It can also be seen that the static pressure in the whole pump decreases and becomes more uniform with increasing flow rate. Thus, the 0.6 $Q_D$ recorded the highest mean static pressure, whereas the 1.9 $Q_D$ records the least. Meanwhile, the design condition, 1.0 $Q_D$, marked a moderate static pressure compared to the two operating conditions.

4.2.2 | Velocity distribution

Figure 10 shows velocity streamline distribution at 0.6, 1.0, and 1.9 $Q_D$. The velocity component of the fluid has an
influence on the pressure behavior inside the pump at the different flow ratios. A similar observation to static pressure distribution is observed in the velocity distribution. The asymmetric geometry of the volute creates the shortest distance between impeller outlet and volute wall at below volute tongue region. The accelerated flow is then guided to volute tongue, and this generates a vortex below the volute tongue at 0.6 $Q_D$. As the flow increases, the vortex at the same position diminishes at the design and 1.9 $Q_D$ conditions.

### 4.2.3 Pressure pulsation intensity

#### Impeller

To evaluate the pressure pulsation intensity generated within the flow passage, the static pressure is normalized using:

$$C_p = \frac{\Delta p}{\frac{1}{2} \rho u^2}$$

(13)

Figure 11 represents the intensity of the pressure pulsation, which is also known as pressure coefficient in the impeller
**FIGURE 11** Pressure pulsation in the impeller flow passage

**FIGURE 12** Pressure pulsation in the volute flow passage

**FIGURE 13** Monitoring points around pump modeling and around volute tongue
flow passage at 0.6, 1.0, and 1.9 \( Q_D \). Inside the impeller, pressure coefficients developed are nonuniform and increases in the radial direction but tends to decrease close to the impeller tip. This can be attributed to the velocity difference generating an unsteady flow shock between the impeller and volute.\(^2\) For all three operating conditions, the impeller eye rendered the lowest pressure coefficients especially at the design condition. This makes the impeller eye highly susceptible to cavitation. At the eye and leading edge of the blade, the flow had attained lower velocity as indicated in Figure 11. High pressure coefficients are obviously observed at the pressure side on the blade surface closer to the trailing edge for all three studied operating conditions. In addition, operating condition 0.6 \( Q_D \) records the highest at that point. Meanwhile, the suction side generates very low pressure pulsation. Gülich\(^2\) pointed out that the Coriolis acceleration generates secondary flows creating velocity differences in the blade flow passage. This is the reason for the significant pressure pulsation difference observed between the pressure and suction sides of the blade. Thus, the blade geometry optimization especially at the leading and trailing edges is recommended. At design flow rate

| Monitoring point | \( x \) | \( y \) | \( z \) |
|------------------|------|------|------|
| C1               | 0.025| 0.089| 0    |
| C2               | 0.0275| 0.087| 0    |
| C3               | 0.028| 0.084| 0    |
| C4               | 0.025| 0.082| 0    |
| C5               | 0.022| 0.0825| 0  |
| R1               | 0.025| 0.089| -0.009|
| R2               | 0.0275| 0.087| -0.009|
| R3               | 0.028| 0.084| -0.009|
| R4               | 0.025| 0.082| -0.008|
| R5               | 0.022| 0.0825| -0.008|
| PS_{NORTH}      | 0.016| 0.27| 0    |
| PS_{SOUTH}      | -0.016| 0.27| 0    |

**Volute**

Figure 12A shows the pressure coefficient in volute flow passage at different flow rate ratios at part-load, design load, and overload conditions, respectively. Similar to the pressure pulsations around impeller blades, the pressure coefficients at design condition are lower compared to the part-load and overload conditions. High deviations are observed below the volute tongue at part-load condition and decrease with increasing sectional area. Moreover, the high pressure pulsation intensity developed in front of volute tongue is mainly because of the asymmetric geometry of the volute\(^5\) which causes the pressure pulsation to increase in the spiral channel as the radius increases. This eventually aids in the conversion of the kinetic energy to potential energy. The high pressure coefficients at a flow rate of 0.6 to 1.9 \( Q_D \) are related to the vortices generated in the impeller. Moreover, the abrupt decrease in velocity at the small distance between impeller outlet and volute wall enhances pressure pulsations. The major losses occurring in the pump are due to flow separation at the blade leading edge and at the tongue region; thus, Figure 12B is used to show the enlarged view of the tongue region. These losses influence the high pressure pulsation at the tongue region which inherently lowers the performance and reliability of the pump. In this case, cavitation is likely to be caused at the surface of the volute tongue especially in the 1.9 \( Q_D \) operating condition where the pressure fluctuated to the maximum coefficient and covered a larger area in the tongue. Higher pressure coefficients are also observed at the tongue region in 0.6 \( Q_D \) but covered a small area compared to 1.9 \( Q_D \). The design operating condition rather revealed moderate values. During a period of a complete impeller rotation, the unsteadiness of the inner flow field is the dominant factor of pressure pulsation, which results in the uneven temporal variation of the pressure curves discussed in the
sections below. The rotor-stator interaction generates the highest pressure pulsation distribution at the volute tongue. Thus, modification of the volute tongue is a better approach of reducing the pressure pulsation intensity in the volute and pump as a whole.

4.3 | Time domain analysis of the pressure pulsations

The pressure pulsations in the pump are analyzed clearly by selecting some relevant monitoring points as shown in Figure 13. Five monitoring points are situated around the volute tongue. The Cartesian coordinate values of the monitoring points are depicted in Table 2.

4.3.1 | Experimental and simulation comparison at the outlet

Figure 14 shows the experimental pressure pulsations compared to the simulated at 0.6, 1.0, and 1.9 \( Q_D \) in one shaft revolution. Six peaks and valleys are clearly depicted in all cases at both experimental and simulated results. The six peaks and
valleys represent the six blades of the impeller passing within a shaft revolution. The experimental amplitudes in all operating conditions are similar to the simulated ones. The operating condition, $0.6 Q_D$, recorded the highest amplitude of about 190-195 kPa, whereas $1.0 Q_D$ marked about 185-190 kPa. Lower amplitude was revealed at the $1.9 Q_D$ according to both experimental and numerical measurements. The simulated amplitudes of (155-167.5 kPa) at that operating condition are marginally higher than the experimental amplitudes (155-167.5 kPa). Thus, with increasing flow rate, the amplitudes of the pressure pulsation tend reduce.
4.3.2 Pressure pulsation at monitoring points

Due to the flow action caused by the rotating impeller and stationary volute, the pump exhibits an unsteady flow characteristic. The monitoring points, which are set around volute tongue region, compared the pressure pulsation in mid-plane (C) and volute wall (R) during one complete impeller rotation as shown in Figure 15. In accordance with the above discussions, the pressure pulsation at design load condition is lower compared to the other operating conditions. The highest pressure value is reached at part-load operating condition. The intensity of the pulsation shows a decreasing trend with increasing flow rate. Slight differences are identified between maximum and minimum values at C1 and R1. Meanwhile, at overload operating condition, the fluctuating pressure shows a nonperiodic behavior at volute wall. Moreover, the irregular pressure pulsation is still predicted at the volute wall at high flow rates. The range where the pressure is fluctuating decreased by about 10 kPa. Observing the shape of the curves at the monitoring points at volute tongue, the simulated pressure of flow rate ratio 0.6 $Q_D$ is below flow rate ratio of 1.0 $Q_D$, whose pressure range increased. The difference between maximum and minimum values at part-load as well as overload operating conditions is more intense in the mid-plane. At the volute wall, the pressure values are generally lower. The shape of curves at C4 and C5 showed a decreasing trend of pressure pulsation at part-load and design operating conditions. The highest deviation of maximum pressure ranges is between upper and lower volute tongue surface at lower flow rates. Smaller deviations are also observed at higher flow rate ratios. Monitoring points C4 and C5 are showing the fluctuating behavior between each impeller blade at part-load and design load conditions. The pressure pulsation at overload condition shows a periodic behavior at volute wall region. Comparison between the internal flow behavior and pressure pulsations at part-load condition shows a good accordance. The increase in pressure from C4 to C5 can be seen clearly in Figure 15E. R4 and R5 did not show a fluctuating behavior at the peaks. This shows that the irregular flow pattern caused by rotating stall between impeller blades has a higher influence on the pressure generation around volute tongue. The rotating stalls have less influence on the flow pattern with regard to pressure generation between impeller outlet and volute tongue at part-load and design load conditions.

4.4 Frequency analysis

To make more deductions from the flow behavior inside the pump, the investigations of the pressure pulsations have been explained further in frequency domain. With this, it is possible to identify any irregularities of flow pattern during the operation. The measured data from the north pressure sensor and the simulated results from the monitoring points around mid-plane of each compared load condition are used for the frequency domain using Fast Fourier Transform (FFT). Figure 16 shows the experimental results of frequency domain at different operating conditions. Frequency range between 0 and 1500 Hz of each flow rate is plotted on the $x$-axis. The $z$-axis shows the dimensionless amplitude of the pressure. The shaft frequency, $f_s$, occurred at 145 Hz which happened to be the highest peak at operating conditions. Meanwhile, 1.9 $Q_D$ recorded the highest among the other operating conditions. The blade passing frequency, $f_b$, excited at multiples of the shaft frequency. The next highest peak excited at 1015 Hz which gives a hint for noise and vibration generations at...
all operating conditions. The frequency domain of the simulated values around volute tongue shows the highest peaks at shaft frequency are shown in Figures 17-19. In the frequency domain of the measured data, the amplitudes at the blade passing frequency can be clearly seen. As the frequency increases, the amplitude decreases for all operating conditions. The lowest amplitude occurred at $f_s$ at flow rate ratio 0.6 $Q_D$ can be seen at monitoring point C3 in Figure 16. Higher amplitudes at $f_s$ as well $f_b$ are observed below the volute tongue. These are the regions where the pressure pulsations reach the highest intensity at part-load condition. High amplitudes are also recognized at monitoring point C4 in 0.6 and 1.0 $Q_D$, while all the other amplitudes of the other monitoring points are seem uniform. Nonetheless, the amplitudes are considerable lower than at part-load and overload operating conditions. The flow behavior below volute tongue exhibited more influence on the pressure pulsation, which can identified in the higher amplitudes at each blade passing frequency. At overload operating condition, the highest amplitude is recorded in front of volute tongue. Globally, a good agreement is established between the experimentally measured and the simulated pressure pulsating data.

5 | CONCLUSIONS

Pressure pulsation intensity within the whole pump passage of a PAT in the pump mode has been investigated through using numerical and experimental methods at different operating conditions. Generally, it can be concluded that the low-specific speed pump selected as a PAT is characterized with the problem of strong pressure pulsations. Thus, reducing the pressure pulsations during the operation of the PAT in the pump mode still remains a major challenge. Below are some of the major findings and recommendations which can be taken into consideration during the design optimization of the pump in order to increase the pump operational reliability.

1. High pressure coefficients are recorded at the pressure side on the blade surface close the trailing edge for all three studied operating conditions compared to the suction side. Thus, the blade geometry optimization especially at the leading and trailing edges is recommended.

2. During a period of a complete impeller rotation, the unsteadiness of the inner flow field is the dominant factor of pressure pulsation, which results in the uneven temporal variation. The rotor-stator interaction generates the highest pressure pulsation distribution at the volute tongue. Thus, modification of the volute tongue is better approach of reducing the pressure pulsation intensity in the volute and pump as a whole.

3. At overload operating condition, the low pressure pulsation coefficients recorded at the trailing edge results in the increase of pressure pulsation amplitude at the monitoring points. However, at part-load operating condition, pressure pulsation intensity mainly depends on the structure of vortex flow at blade passage exit because of the high velocity attained by the flow.

4. The shaft frequency occurred at 145 Hz which happened to be the highest peak at operating conditions. Furthermore, the blade passing frequency excited at multiples of the shaft frequency. The next highest peak excited at 1015 Hz which gives a hint for noise and vibration generations at all operating conditions.

ACKNOWLEDGMENT

This study is financially supported by the National Key R&D Program Projects of China (Grant No. 2018YFB0606101), Natural Science Foundation of China (Grant No. 51809121), and Natural Science Foundation of Jiangsu Province (Grant No. BK20170548) and supported by Senior Talent Research Start-up Foundation of Jiangsu University (Grant No. 17JDG040). The supports are gratefully acknowledged.

NOMENCLATURE

| GLOSSARY |
|-----------|
| $b$       | Impeller outlet, mm |
| $d$       | Inner diameter of impeller, mm |
| $D_H$     | Hole diameter, mm |
| $F_1$, $F_2$ | Blending or auxiliary functions in turbulence model |
| $f_b$     | Blade passing frequency, Hz |
| $f_s$     | Shaft frequency, Hz |
| $g$       | Acceleration due to gravity, m/s$^2$ |
| $H$       | Head, m |
| $k$       | Kinetic energy of turbulence, m$^2$/s$^2$ |
| $M$       | Torque, Nm |
| $n$       | Rotational speed, r/min |
| $n_s$     | Specific speed |
| $P$       | Pressure, Pa |
| $Q$       | Nominal flow rate, m$^3$/h |
| $R_1$     | Inlet radius, mm |
| $R_2$     | Outlet radius, mm |
| $R_H$     | Hub radius, mm |
| $R_L$     | Leading edge radius, mm |
| $R_S$     | Shroud radius, mm |
| $S$       | Scalar measure of the vorticity tensor |
| $t$       | Time, s |
| $u$       | Impeller outlet velocity, m/s |
### Abbreviations

| Symbol | Description |
|--------|-------------|
| Z      | Blade number |
| $\Gamma$ | Auxiliary variable in turbulence model |
| $\rho$ | Density, kg/m$^3$ |
| $\epsilon$ | Dissipation of kinetic energy of turbulence, m$^2$/s$^3$ |
| $\mu$ | Dynamic viscosity, Pa.s |
| $\eta$ | Efficiency |
| $\delta_{ij}$ | Kronecker delta |
| $y^*$ | Nondimensional wall distance |
| $-\rho u'_i u'_j$ | Reynolds stress tensor |
| $\omega$ | Specific dissipation of turbulence kinetic energy, s$^{-1}$ |
| $\beta'$ | Turbulence model coefficients |
| $\mu_t$ | Turbulent viscosity, m$^2$/s |
| $\sigma_k$ | Turbulence model coefficients |
| $\sigma_\omega$ | Turbulence model coefficients |

### Subscript

| Symbol | Description |
|--------|-------------|
| $i$, $j$ | Components in different directions |
| $\xi$, $\xi'$ | Cartesian coordinates: $x$, $y$, $z$ |

### References

1. World Energy Council. *World Energy Resources Hydropower*. London, UK: World Energy Council; 2016.
2. Gülich JF. *Centrifugal Pumps*. Berlin, Germany: Springer; 2008.
3. Minggao T, Houlín L, Yong W, Kai W, Liang D. CFD analysis on effects of impeller outlet diameter on flow field in centrifugal pump [J]. *J Drain Irrig Mach Eng*. 2009;27(5):314-318.
4. Jafarzadeh B, Hajari A, Alishahi MM, et al. The flow simulation of a low-specific-speed high-speed centrifugal pump. *Appl Math Model*. 2011;35(1):242-249.
5. Pei J, Wang WJ, Yuan SQ, Zhang JF. Optimization on the impeller of a low-specific speed centrifugal pump for hydraulic performance improvement. *Chin J Mech Eng*. 2016;29(5):1-11.
6. Arndt N, Acosta AJ, Brennen CE, Caughney TK. Experimental investigation of rotor–stator interaction in a centrifugal pump with several vaned diffusers. *ASME J Turbomach*. 1990;112(1):98-108.
7. Yuan S, Ni YY, Pan Z, Yuan J. Unsteady turbulent simulation and pressure fluctuation analysis for centrifugal pumps. *Chin J Mech Eng*. 2005;22(1):64-69.
8. Guo S, Maruta J. Experimental investigations on pressure fluctuations and vibration of the impeller in a centrifugal pump with vaned diffusers. *JSME Int J B*. 2005;48(1):136-143.
9. González J, Fernández J, Blanco E, Santolario C. Numerical simulation of the dynamic effects due to impeller volute interaction in a centrifugal pump. *Trans ASME J Fluid Eng*. 2002;124(2):348-355.
10. Khalifa AE, Al-Qutub AM, Ben-Mansour R. Study of pressure fluctuations and induced vibration at blade-passing frequencies of a double volute pump. *Arab J Sci Eng*. 2011;36(7):1333-1345.
11. Pavesi G, Cavazzini G, Ardzizzon G. Time–frequency characterization of the unsteady phenomena in a centrifugal pump. *Int J Heat Fluid Flow*. 2008;29(5):1527-1540.
12. Spece R, Amaralíteixier J. A CFD parametric study of geometrical variations on the pressure pulsations and performance characteristics of a centrifugal pump. *Comput Fluids*. 2009;38(6):1243-1257.
13. Yao ZF, Wang FJ, Qu LX, Xiao RF, He CL, Wang M. Experimental investigation of time-frequency characteristics of pressure fluctuations in a double-suction centrifugal pump. *J Fluids Eng*. 2011;133:101303.
14. Cong GH, Wang FJ. Numerical investigation of unsteady pressure fluctuations near volute in a double-suction centrifugal pump. *Trans Chin Soc Agric Mach*. 2008;39:60-67 (in Chinese).
15. Tan L, Wang YC, Cao SL, Zhu BS. Characteristics of unsteady flow around the tongue region in a centrifugal pump. *Trans Beijing Inst Technol*. 2014;34:670-675 (in Chinese).
16. Yang SS, Kong FY, Chen B. Research on volute design method of pump as turbine using CFD. *Int Agric Eng J*. 2011;20(3):1-7.
17. Yang SS, Liu HL, Kong FY, Xia B, Tan LW. Effects of the radial gap between impeller tips and volute tongue influencing the performance and pressure pulsations of pump as turbine. *J Fluids Eng*. 2014;136:054501.
18. Xu B, Chen D, Zhang H, Zhou R. Dynamic analysis and modeling of a novel fractional-order hydro-turbine-generator unit. *Nonlinear Dyn*. 2015;81(3):1263-1274.
19. Xu B, Wang F, Chen D, Zhang H. Hamiltonian modeling of multi-hydro-turbine governing systems with sharing common penstock and dynamic analyses under shock load. *Energy Convers Manage*. 2016;108:478-487.
20. Xu B, Chen D, Behrens P, Ye W, Guo P, Luo X. Modeling oscillation modal interaction in a hydroelectric generating system. *Energy Convers Manage*. 2018;174:208-217.
21. Williams AA. The turbine performance of centrifugal pumps: a comparison of prediction methods. *Proc Inst Mech Eng A*. 1994;208(1):59-66.
22. Derakhshan S, Mohammadi B. Efficiency improvement of centrifugal reverse pumps. *ASME J Fluids Eng*. 2009;131:1-9.
23. Stepansoff AJ. *Centrifugal and Axial Flow Pumps*. New York, NY: John Wiley; 1957.
24. Menter FR. Two-equation eddy viscosity models for engineering applications. AIAA J. 1994;32:1598-1605.
25. Menter FR. Zonal two equation turbulence models for aerodynamic flows. In Proceedings of the 24th Fluid Dynamics Conference, Orlando, FL, USA, 6-9 July 1993; AIAA Paper; pp. 93-2906.
26. Wilcox D. Turbulence Modeling for CFD (p. 4). La Canada, CA: DCW Industries Inc.; 1993.
27. Wilcox D. Reassessment of the scale-determining equation for advanced turbulence models. AIAA J. 1988;26:1299.
28. Celik IB, Ghia U, Roache PJ, Freitas CJ, Coleman H, Raad PE. Procedure for Estimation and Reporting of Uncertainty due to Discretization in CFD applications. J Fluids Eng. 2008;130:078001.
29. Lobanoff VS, Ross RR. Centrifugal Pumps: Design & Applications, 2nd edn. Houston, TX: Gulf Publishing Company; 1985.

How to cite this article: Zhang J, Appiah D, Zhang F, Yuan S, Gu Y, Asomani SN. Experimental and numerical investigations on pressure pulsation in a pump mode operation of a pump as turbine. Energy Sci Eng. 2019;7:1264-1279. https://doi.org/10.1002/ese3.344