Numerical study of pressure fluctuation in the whole flow passage of a low specific speed mixed-flow pump

Wenwu Zhang¹, Baoshan Zhu², Zhiyi Yu¹ and Ce Yang¹

Abstract
The understanding of pressure fluctuation mechanism is the basis for stability and noise analysis in pumps. In this article, numerical study was carried out for the pressure fluctuation within the impeller and the guide vane of a low specific speed mixed-flow pump. Structured mesh was adopted in the whole flow passage. The numerical approach was validated by comparing with the experimental data of external characteristics and fluctuation in the guide vane passage. Through MATLAB code and Fourier analysis, the pressure fluctuation in the whole flow passage was obtained at four flow rate conditions. It is found that pressure fluctuation due to rotor–stator interaction is first enhanced and then weakened along the stream-wise direction with the maximal value located near the guide vane inlet. As a result of the structure design of passage and secondary flow, some obvious vortexes will form at the hub region near the pressure surface at small flow rate conditions, thus aggravating pressure fluctuation therein. On the whole, pressure fluctuation at small flow rate conditions is more remarkable with the fluctuation period being more disordered; the fluctuation intensity is higher in the guide vane than the impeller, while the area of strong fluctuation is also wider.

Keywords
Low specific speed, pressure fluctuation, mixed-flow pump, rotor–stator interaction, numerical study

Date received: 29 November 2016; accepted: 10 April 2017

Introduction
The structure and performance of mixed-flow pumps are between centrifugal and axial flow pumps. The features such as compact structure, easy starting, and high efficiency make it suitable for use in farmland irrigation, water-logging drainage, sewage treatment, power-plant cooling, and so on.¹ ⁴ Mixed-flow pumps are generally grouped into low specific speed, medium specific speed, and high specific speed mixed-flow pumps. And this range changes with market demand and R&D technology.

In recent years, research has been conducted on performance optimization, pressure fluctuation, and internal flow of the centrifugal and axial flow pumps.⁵ ⁹ For mixed-flow pumps, related research has mainly concentrated on the performance and flow instability. In the first aspect, Kim et al.¹⁰ explored the effect of tip clearance on head and hydraulic efficiency of a mixed-flow pump and found that the existence of tip clearance could improve the “saddle” phenomenon of head-flow characteristic; employing non-linear circulation distribution, Zhang et al.¹¹ optimized the flow pattern at the impeller outlet of a mixed-flow pump to increase the

¹School of Mechanical Engineering, Beijing Institute of Technology, Beijing, P.R. China
²State Key Laboratory of Hydroscience and Engineering, Department of Thermal Engineering, Tsinghua University, Beijing, P.R. China

Corresponding author:
Zhiyi Yu, School of Mechanical Engineering, Beijing Institute of Technology, Beijing 100081, P.R. China.
Email: yuzhiyi@bit.edu.cn

Creative Commons CC-BY: This article is distributed under the terms of the Creative Commons Attribution 4.0 License (http://www.creativecommons.org/licenses/by/4.0/) which permits any use, reproduction and distribution of the work without further permission provided the original work is attributed as specified on the SAGE and Open Access pages (https://us.sagepub.com/en-us/nam/open-access-at-sage).
hydraulic efficiency. For flow instability, the study with particle image velocimetry (PIV) technology by Miyabe et al. shows that the unstable head-flow characteristic is caused by the large-scale abrupt backflow from the vaned diffuser to the impeller outlet, and some measures are put forward to improve the positive slope of the head performance curve; in the analysis of Yamade et al., large eddy simulation was performed for internal flow of a mixed-flow pump and it was found that the instability characteristic took place when the flow rates are 55%-60% of the design condition.

On the whole, the specific speed of the mixed-flow pump in research so far is usually between 300 and 700. For the case with the specific speed less than 300, Zhang et al. explored the influence of tip clearance on the pressure fluctuation and found that the pressure fluctuation in the shroud region of the impeller inlet suddenly increased when the clearance value was 1.0 mm. In addition, as it is difficult to set monitoring points in the rotating passage, no much research is performed on the pressure fluctuation in the impeller passage. In this study, unsteady simulation was carried out with ANSYS_CFX 14.0 for the internal flow of a mixed-flow pump ($n_s = 149$), and the results were processed with Fourier analysis and MATLAB. The flow field and pressure fluctuation in the impeller and guide vane passages were explored to obtain a more comprehensive understanding of pressure fluctuation mechanism in such pumps, which will help in design optimization.

**Structure and parameters of the pump**

The three-dimensional (3D) geometric model of the whole flow passage and the test pump model of the mixed-flow pump are shown in Figures 1 and 2, respectively. Its design parameters are listed in Table 1 where the specific speed is computed by

$$n_s = \frac{3.65n\sqrt{Q_d/60}}{H_d^{3/4}}$$

where, $w$ is the relative velocity, $\omega$ is the rotational frequency of the impeller, and $\tau$ denotes the viscous stress tensor concerning the fluid viscosity as well as the turbulence viscosity. The turbulence is modeled by the shear stress transport (SST) model, a combination of the $k - \omega$ model applied in the near-wall region and the $k - e$ model employed for the main flow. This model has high accuracy in predicting flow separation under an adverse pressure gradient by taking the transport effects into account. Here, the eddy viscosity is computed by

**Numerical methods**

**Governing equations**

To explore the characteristics of pressure fluctuation in this pump, unsteady Reynolds-averaged Navier–Stokes (RANS) equations are solved for the internal flow in the framework of ANSYS_CFX 14.0. The continuity and momentum equations for incompressible flow can be written as

$$\nabla \cdot \mathbf{w} = 0$$

$$\frac{\partial (\rho \mathbf{w})}{\partial t} + \nabla \cdot (\rho \mathbf{w} \mathbf{w} - \tau) = -\nabla p - 2\rho \omega \times \mathbf{w} + \rho \omega^2 \mathbf{r}$$

where, $\mathbf{w}$ is the relative velocity, $\omega$ is the rotational frequency of the impeller, and $\tau$ denotes the viscous stress tensor concerning the fluid viscosity as well as the turbulence viscosity. The turbulence is modeled by the shear stress transport (SST) model, a combination of the $k - \omega$ model applied in the near-wall region and the $k - e$ model employed for the main flow. This model has high accuracy in predicting flow separation under an adverse pressure gradient by taking the transport effects into account. Here, the eddy viscosity is computed by
where $a_1$ is the model constant ($a_1 = 5/9$), $S$ is an invariant measure of strain rate, $F_2$ is the blending function, and $k$ and $\omega$ are the turbulence kinetic energy and the turbulence frequency, respectively.

**Mesh and boundary conditions**

Separate meshes were generated for the inlet pipe, outlet pipe, impeller, and guide vane. ICEM_CFD and TurboGrid were used for the mesh generation of the former two parts and the latter two parts, respectively. The distance of the first node from the wall was controlled ensuring that the $y^+$ values are between 20 and 30. Meanwhile, scalable wall functions were used and about 15 boundary layers were placed in the near-wall region. The mesh view of the four flow parts are demonstrated in Figure 3. Figure 4 shows the mesh independence analysis of the mesh system at the design condition. Here, efficiency_CFD represents the hydraulic efficiency of the pump calculated by ANSYS_CFX. The numbers of mesh elements and nodes in each part of the computation domain are shown in Table 2.

A constant velocity was specified according to experimental values at the inlet of the computational domain, with the flow direction normal to the inlet surface. At the outlet, the free outflow condition was used. No-slip condition was imposed on all wall boundaries. The transient rotor–stator approach was adopted for data exchange in the rotor–stator interaction region (namely, the impeller–inlet pipe interaction region and the impeller–guide vane interaction region), through which the true transient interaction of the flow can be predicted.

**Setting and validation of monitoring points**

According to the rotational speed listed in Table 1, the period of the rotation is 0.03 s. Meanwhile, according to Wang et al., the time step for unsteady calculation was set to 0.0002 s in the simulation, corresponding to the impeller rotation of 2.4° a time step. The total time of simulation is set as 15 periods, that is, 0.45 s.

To obtain the pressure information in the mixed-flow pump, four surfaces were set in both the impeller and guide vane passages from the inlet to the outlet, respectively, as shown in Figure 5, where $P_{iC}$ ($i = 1–8$) denotes the center point of the surface, while $P_{ih}$ ($i = 1–8$) denotes the middle point of the hub side of these surfaces. What’s more, the setting of $P_{ih}$ in the simulation is in accordance with the experiment for reasonable comparison.

Figure 6 is the frequency domain diagrams of the points $P_{ih}$ and $P_{ih}$ located in the guide vane passage. It
can be observed that the dominant frequencies of the two points are all 6fn in both the simulation and the experiment. The amplitudes of the dominant frequency agree well with each other, and the frequency 12fn occurs in both conditions. This is resulted from the action of the impeller on the guide vane as the impeller blade number is 6. Meanwhile, the fluctuation amplitude of point P_{sh}, which is nearer to the guide vane inlet, is larger than that of point P_{sb}, and there are more low frequencies from experiment.

**Results and discussions**

**Performance comparison with the experiment**

At eight different flow rate conditions, simulation was done for the transient flow in the pump, as shown in Figure 7. The performance curves (involving head, efficiency, and power performance) from the simulation agree well with the experiment, and the relative error of efficiency, head, and power at the design condition is 7.8%, 6.3%, and 2.1%, respectively, which manifests

---

**Figure 5.** Location of monitoring points (ps, ss, and hub_s denote the pressure surface, the suction surface, and the hub surface, respectively): (a) impeller passage and (b) guide vane passage.

**Figure 6.** Frequency domain diagrams of the points P_{sh} and P_{sb}.

“Amp” is the amplitude of the pressure fluctuation, while “f/fn” denotes the frequency ratio of pressure fluctuation to impeller rotation.
the reliability of the adopted numerical methodology. The relative error herein is defined as

\[ \delta = \frac{X_{\text{sim}} - X_{\text{exp}}}{X_{\text{exp}}} \times 100\% \]  

(5)

where \(X_{\text{sim}}\) and \(X_{\text{exp}}\) denote the performance value from simulation and experiment, respectively. However, the performance curves demonstrate that the pump not only has a wide area of high efficiency but also avoids the saddle phenomenon of a traditional mixed-flow pump. Meanwhile, the maximum error is efficiency prediction at large flow rate conditions. This may be due to more severe cavitation at such conditions. What’s more, this simulation only considered hydraulic efficiency, ignoring volumetric and mechanical losses.

**Pressure fluctuation in the impeller passage**

The pressure fluctuation of the four center-monitoring points in the impeller passage is presented in Table 3. Here, the pressure fluctuation coefficient is defined as

\[ C_p = \frac{S}{\rho g H_d} \times 100\% \]  

(6)

where \(H_d\) is the design head, and \(S\) is the standard deviation of pressure fluctuation of the monitoring points, namely

\[ S = \sqrt{\frac{1}{N} \sum_{i=1}^{N} [P_i(t) - \overline{P_i}]^2} \]  

(7)

where \(P_i(t)\) is the pressure at time \(t\) while \(\overline{P_i}\) is the time average pressure.

As shown in Table 3, the pressure fluctuation coefficients increase gradually from inlet to outlet of the impeller passage at four flow rate conditions. The maximum values appear near the impeller outlet, which is closely related to the rotor–stator interaction and the complicated flow state at the outlet region. Here, the pressure fluctuation coefficient of point \(P_{4C}\) at \(0.5Q_d\), \(0.75Q_d\), \(Q_d\), and \(1.25Q_d\) conditions are 6.27, 2.95, and 1.22 times as large as that at the design condition, respectively.

The pressure fluctuation of point \(P_{4C}\) at different flow rates is demonstrated in Figure 8. Periodicity of the pressure fluctuation of point \(P_{4C}\) at \(Q_d\) and \(1.25Q_d\) conditions is better, while pressure fluctuation is more disordered in the \(0.5Q_d\) and \(0.75Q_d\) conditions. Through fast Fourier transform (FFT), corresponding frequency-domain diagram is obtained and shown in Figure 9. The dominant frequencies of point \(P_{4C}\) are all \(8fn\) in the four conditions, which is resulted from the action of the guide vane on the impeller (the blade number of guide vane is 8). The amplitude of fluctuation for the dominant frequency in the \(0.5Q_d\), \(0.75Q_d\), and \(1.25Q_d\) conditions are 4.68, 2.12, and 1.29 times as great as that in the design condition, respectively. In \(0.5Q_d\) and \(0.75Q_d\) conditions, there are also the secondary frequencies, that is, \(0.37fn\) and \(3.25fn\), respectively, accompanied by many other frequencies. According to Table 3 and Figures 8 and 9, it can be concluded that

---

**Table 3.** Pressure fluctuation coefficients in the impeller passage (%).

| Conditions | Points |
|------------|--------|
|            | \(P_{1C}\) | \(P_{2C}\) | \(P_{3C}\) | \(P_{4C}\) |
| \(0.5Q_d\) | 1.36    | 1.84    | 2.39    | 3.76    |
| \(0.75Q_d\)| 0.94    | 1.14    | 1.39    | 1.77    |
| \(Q_d\)    | 0.17    | 0.19    | 0.23    | 0.60    |
| \(1.25Q_d\)| 0.23    | 0.60    | 0.92    | 0.73    |
the pressure fluctuation of the impeller passage in the small flow rate conditions is affected significantly by the rotor–stator interaction. Therefore, off-design conditions, especially the small flow rate conditions, should be avoided in the running of the mixed-flow pump.

Figure 10 shows the streamlines’ distribution in the impeller passage at 0.298 s on the span of 0.1. Here, the span denotes the dimensionless distance of the analyzed surface from the hub. It can be seen that the streamlines are well distributed in the design and 1.25\(Q_d\) conditions, while in 0.5\(Q_d\) and 0.75\(Q_d\) conditions, obvious vortexes occur at the hub region near the pressure surface. The formation of vortexes is closely related to the passage design of the low specific speed mixed-flow pump. Impeller blades were designed with a great distortion and a large wrap angle. Due to the viscous effect, vortexes and dead zones will emerge at the convex surface (located at pressure surface) near the impeller outlet at 0.5\(Q_d\) condition, namely, vortex 1 in Figure 10(a). Because of this vortex 1, the cross-sectional area is reduced, and hence, the flow near the pressure surface becomes more disordered. Meanwhile, it hinders the succeeding fluid from upstream, leading to the thickening of upstream boundary layer, and thus forms vortex 2, as shown in Figure 10(a). At 0.75\(Q_d\) condition, due to the larger velocity in the main flow area, no obvious dead zone can form near the pressure surface in the impeller outlet region, but the upstream near-wall region is still affected and vortexes appear there. With the further increment in flow rate (design and 1.25\(Q_d\) conditions), vortexes near the pressure surface will gradually weaken and finally disappear.

According to the vortex distribution at 0.5\(Q_d\) condition in Figure 10(a), the central hub-to-shroud surface can be drawn, which passes through the vortex
center. Figure 11 demonstrates the meridional flow field corresponding to the central hub-to-shroud surface. The pressure distribution shows larger axial component of the pressure gradient near the impeller outlet, which results in the pressure gradient from the hub to the shroud. Meanwhile, because of wide flow passage of the pump, the pressure gradient effect in this direction becomes more prominent, which leads to the secondary flow in an opposite direction near the outlet. Influenced by this secondary flow and adverse pressure gradient in the main flow direction, thickness of the boundary layer at hub wall increases, and hence, flow separation and vortex occur (Figure 11(b)).

From the above analysis, it is found that due to the special passage design of this low specific speed mixed-flow pump, obvious vortexes will arise in small flow rate conditions at the hub region near the pressure surface. Thus, the generated vortexes will aggravate pressure fluctuation therein. Correspondingly, the pressure fluctuation intensity of the monitoring point P4C at small flow rate conditions will be much higher relative to the design and large flow rate conditions.

**Pressure fluctuation in the guide vane passage**

Due to the rotor–stator interaction, pressure fluctuation in the impeller and guide vane passages is closely related to each other. The distribution of velocity vector in the interaction region at 0.1 span, when is 0.298 s, is shown in Figure 12. At small flow rate conditions...
vortexes and flow separation occur near pressure surface at the impeller outlet region, which is in accordance with the previous analysis. The vortexes at the impeller outlet region will have impact on the flow in the guide vane inlet region, resulting in the strong pressure fluctuation therein. From Figure 12, it can be observed that impact loss in the guide vane inlet region is the most serious at $0.5Q_d$ condition (see Figure 12(a)), followed by $0.75Q_d$ condition and then $1.25Q_d$ condition. In addition, when impeller blades are close to the guide vane blades, velocity in the vicinity will increase significantly, which is especially remarkable at $0.5Q_d$ condition. In fact, the rotor–stator interaction between the impeller blades and the guide vane blades is the main reason causing the pressure fluctuation in the pump passage.
Through the post-processing with MATLAB on the result data at all the moments, distribution of $C_p$ in the guide vane passage was obtained according to equation (6). Figure 13 shows the distribution of $C_p$ on the central hub-to-shroud surface of the guide vane passage in four flow rate conditions. It can be seen that the intensity of pressure fluctuation in these four conditions decreases from inlet to outlet in the guide vane passage. The gradient of pressure fluctuation in the inlet region is larger than in the central and outlet regions, which illustrates that the impact of the rotor–stator interaction on pressure fluctuation in the guide vane passage is attenuated along the flow direction. On the whole, the difference of pressure fluctuation at four flow rates lies mainly in the inlet and central regions of the passage, while the difference in downstream region is small. The descending order of pressure fluctuation in the guide vane is $0.5Q_d$, $0.75Q_d$, $1.25Q_d$, and $Q_d$ which is in accordance with the impact strength at the inlet of the guide vane passage. According to the above analysis, the hydraulic design of such pumps should be focused on the rotor–stator interaction region and the guide vane inlet region to reduce the pressure fluctuation. The matching between the impeller outlet angle and the guide vane inlet angle, as well as the distance between the impeller and the guide vane may be two important factors, and this will be our study in the future.

The pressure fluctuation coefficients in the guide vane passage (%). Table 4.

| Conditions | Points | P5C | P6C | P7C | P8C |
|-----------|--------|-----|-----|-----|-----|
| $0.5Q_d$  | 4.79   | 3.04| 2.84| 1.74|     |
| $0.75Q_d$ | 3.83   | 2.74| 2.69| 1.66|     |
| $Q_d$     | 2.37   | 1.99| 1.99| 1.32|     |
| $1.25Q_d$ | 3.26   | 2.96| 2.24| 1.54|     |

Figure 14. Frequency domain characteristic of point P5C.
which is similar to the case of the monitoring point $P_{4C}$, illustrating the flow complexity in the pump at the small flow rate conditions.

**Conclusion**

Through unsteady numerical calculation, the characteristics of pressure fluctuation in a low specific speed mixed-flow pump were explored at four flow rate conditions. The results can be summarized as follows:

1. Relative to the design and large flow rate conditions, fluctuation is more remarkable at small flow rate conditions, and the fluctuation period is more disordered. Moreover, the amplitude of the secondary frequency is greater, and more other frequencies are accompanied in small flow rate conditions.

2. Due to the passage design, the low specific speed mixed-flow pump can obtain a wider high-efficiency area, and it also avoids the saddle phenomenon existing in the traditional mixed-flow pumps. Meanwhile, obvious vortices form at the hub region near pressure surface at small flow rate conditions, and the generated vortices will aggravate pressure fluctuation therein. In the design and large flow rate conditions, as the velocity in main flow area is larger, the intensity of the vortices near the pressure surface gets weaker.

3. At four flow rate conditions, the difference of pressure fluctuation in the guide vane mainly lies in the inlet and central regions, while the difference in the downstream region is smaller. The impact and pressure fluctuation in the inlet region of the guide vane is stronger at small flow rates, which is directly related to the vortices at the impeller outlet in such conditions.

4. The pressure fluctuation from the impeller inlet to the guide vane outlet is enhanced first and then weakened, with the maximal value located near the guide vane inlet. So, the hydraulic design of such pumps should be focused on the rotor–stator interaction and the guide vane inlet regions to reduce the pressure fluctuation. Besides, the pressure fluctuation intensity in the guide vane is higher compared to the impeller, and the distribution area of strong fluctuation is wider.

**Declaration of conflicting interests**

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

**Funding**

The author(s) disclosed receipt of the following financial support for the research, authorship, and/or publication of this article: This work was supported by the National Natural Science Foundation of China (nos 51579006 and 51679122).

**References**

1. Murugesan C and Rudramoorthy R. Experiment and numerical study of efficiency improvement by surface coating on the impellers and diffusers of mixed flow submersible bore well pumps. *J Sci Ind Res* 2016; 75: 300–305.

2. Heo M, Kim K, Kim J, et al. High-efficiency design of a mixed-flow pump using a surrogate model. *J Mech Sci Tech* 2016; 30: 541–547.

3. Bing H, Cao S, He C, et al. Experimental study of the effect of blade tip clearance and blade angle error on the performance of mixed-flow pump. *Tech Sci* 2013; 56: 293–298.

4. Zhang W, Yu Z, Zhu B, et al. Characteristic analysis on the pressure fluctuation in the impeller of a low specific speed mixed-flow pump. *IOP Conf Series: Mater Sci Eng* 2016; 129: 012035.

5. Engin T, Gu M and Scholz R. Effects of tip clearance and impeller geometry on the performance of semi-open ceramic centrifugal fan impellers at elevated temperatures. *Exp Therm Fluid Sci* 2006; 30: 565–577.

6. Wu H, Miorini R and Katz J. Measurements of the tip leakage vortex structures and turbulence in the meridional plane of an axial water-jet pump. *Exp Fluid* 2011; 50: 989–1003.

7. Qu W, Tan L, Cao S, et al. Numerical investigation of clocking effect on a centrifugal pump with inlet guide vanes. *Eng Comput* 2016; 33: 465–481.

8. Miorini R, Wu H and Katz J. The internal structure of the tip leakage vortex within the rotor of an axial water-jet pump. *J Turbomach* 2012; 134: 403–419.

9. Qu L, Wang F, Cong G, et al. Effect of volute tongue-impeller gaps on the unsteady flow in double-suction centrifugal pump. *Trans Chin Soc Agricult Mach* 2011; 42: 50–55.

10. Kim D, Min Y, Kim J, et al. A study of tip clearance effect for a mixed-flow pump on performance. In: *Proceedings of the ASME 2013 fluids engineering division summer meeting*, Incline Village, NV, 7–11 July 2013. New York: ASME.

11. Zhang D, Shi W, Pan D, et al. Hydraulic performance optimization and experiment of special mixed-flow pump based on numerical simulation. *J Mech Eng* 2014; 50: 177–184.

12. Miyabe M, Maeda H, Umecki I, et al. Unsteady head-flow characteristic generation mechanism of a low specific speed mixed-flow pump. *J Therm Sci* 2006; 15: 139–144.

13. Miyabe M, Furukawa A, Maeda H, et al. A behavior of the diffuser-rotating stall in a low specific speed mixed-flow pump and its suppression. *Int J Fluid Mach Syst* 2009; 2: 113–121.

14. Yamade Y, Kato C, Shimizu H, et al. Large eddy simulation of internal flow of a mixed-flow pump. In:
Proceedings of the ASME 2009 fluids engineering division summer meeting, vol. 1, Vail, CO, 2–6 August 2009, pp.407–416. New York: ASME.

15. Zhang K. Theory of fluid machinery, vol. 1. 1st ed. Beijing, China: China Machine Press, 2000, pp.76–80.
16. Zhang W, Yu Z, Zhu B, et al. Influence of tip clearance on pressure fluctuation in low specific speed mixed-flow pump passage. Energies 2017; 10: 148.
17. Cong G, Peng G, Yu Z, et al. Hydrodynamic design of rotodynamic pump impeller for multiphase pumping by combined approach of inverse design and CFD analysis. J Fluid Eng: T ASME 2005; 127: 330–338.
18. Feng J, Luo X, Guo P, et al. Influence of tip clearance on pressure fluctuations in an axial flow pump. J Mech Sci Tech 2016; 30: 1603–1610.
19. Wang Y, Tan L, Zhu B, et al. Numerical investigation of influence of inlet guide vanes on unsteady flow in a centrifugal pump. J Mech Eng Sci 2015; 229: 3405–3416.
20. Brennen C. Hydrodynamics of pumps. 1st ed. Cambridge: Cambridge University Press, 2011.

Appendix I

Notation

| Symbol | Description                           |
|--------|---------------------------------------|
| $D$    | impeller diameter (m)                 |
| $f_n$  | frequency of the impeller rotation (s$^{-1}$) |
| $F_2$  | blending function                     |
| $H_d$  | head (m)                              |
| $n$    | rotational speed (r/min)              |
| $n_s$  | specific speed (m$^{3/4}$/s$^{3/2}$)  |
| $P$    | shaft power (kW)                      |
| $p$    | static pressure (kPa)                 |
| $Q_d$  | design flow rate (m$^3$/min)          |
| $r$    | radial coordinate (dimensionless)     |
| $S$    | suction surface                       |
| $S$    | standard deviation of the fluctuation (dimensionless) |
| $w$    | relative velocity (m/s$^1$)           |
| $X_{sim}$ | performance value from simulation     |
| $X_{exp}$ | performance value from experiment     |
| $Z$    | blade number                          |
| $\rho$ | flow density (kg/m$^3$)               |
| $\tau$ | viscous stress tensor (Pa)            |
| $\delta$ | relative error (dimensionless)        |
| $\omega$ | rotational frequency of the impeller (s$^{-1}$) |
| $\kappa$ | turbulence kinetic energy (m$^2$/s$^2$) |
| $\omega$ | turbulence eddy frequency (s$^{-1}$)  |
| $\mu_t$ | turbulent viscosity (Pa·s)            |