Effect of the flow upstream the impeller inlet on flow instability of a centrifugal pump

W X Ye\textsuperscript{1}, X J Li\textsuperscript{2}, Z C Zhu\textsuperscript{2} and X W Luo\textsuperscript{1}

\textsuperscript{1} State Key Laboratory of Hydroscience and Engineering, Department of Energy and Power Engineering, Tsinghua University, Beijing 100084, China
\textsuperscript{2} Key Laboratory of Fluid Transmission Technology of Zhejiang Province, Zhejiang Sci-Tech University, Hangzhou 310018, China

E-mail: luoxw@tsinghua.edu.cn

Abstract. Under part-load conditions, flow instability phenomena in a hydraulic machinery could lead to strong pressure fluctuations and vibrations, posing a great threat to the safety of the units. This paper analysed unsteady flow in a centrifugal pump (specific speed of 39 \text{m}^{1/3}\text{m}^{3}\text{s}^{-1}) with different inlet geometries under part-load conditions. An advanced turbulence model, PANS (partially averaged Navier-Stokes), acting as a bridge from RANS to DNS, is applied for numerical simulations. The results indicated the pump with a straight inlet pipe reached higher values of pump head and efficiency near the design point than the test pump with a non-straight inlet pipe. According to the characteristic curves, hump phenomenon was observed at 0.75\(\phi_{\text{bep}}\) for the pump having a straight inlet pipe, while it occurred at 0.5\(\phi_{\text{bep}}\) for the test pump. Under the condition of a low flow rate i.e. \(\phi=0.5\phi_{\text{bep}}\), there is strong pre-swirling flow upstream impeller inlet for the test pump, which contributes to the propagation of rotating stall cells; But energy loss in the impeller inlet of the pump with straight inlet pipe is slightly higher than that in the test pump due to the presence of the stationary stall cells, which causes large energy loss in the impeller and the passage upstream impeller inlet. Further, rotating stall cells are captured in the impeller in both pumps under the condition of \(\phi=0.75\phi_{\text{bep}}\).

1. Introduction
Flow instability related to turbulent swirling flow is a quite common phenomenon, which attracts much researchers’ attention over a century \[1-3\]. In many industries applications, centrifugal pumps are widely applied. However, under part-load conditions, several adverse phenomena, such as strong noises,
vibrations and strong pressure fluctuations, etc., posing a great threat to the stability and safety to the units. Thus, it is of great significance to study flow instability in a centrifugal pump.

The flow instability analysis has been investigated by means of experimental methods and numerical calculations. Yoshida [4] observed rotating stall cells in the vaned diffuser of a centrifugal pump by experiment to clarify the interaction effect between impeller and diffuser, and the result depicted that the clearance between impeller and vaned diffusers had great effects on rotating stall in the diffuser. Further, Sinha et al [5] investigated the onset and development of rotating stall cells in a centrifugal pump with a vaned diffuser by using Particle Image Velocimetry (PIV), and the cross-spectra of pressure fluctuations showed that the propagation rate of stall cells was 0.93Hz, 6.2% of the impeller rotation speed. Owing to the complexity of turbulent flow, it is still inadequate to carry out some internal flow researches through traditional measurement technology. With the development of computational fluid dynamics, numerical calculations of turbulent flows have been approved to be a valuable tool for instability analysis [6-10]. Some useful turbulence modelling methods have been developed for different turbulent swirling flow cases, such as realizable k-ε model in analysis of flow in a double volute centrifugal pump [7], SST k-ω model for swirling flows analysis [8], LES applied in cavitation analysis [9], and Partially-averaged Navier–Stokes (PANS) model used in flow simulations in a draft tube of a Francis turbine [10].

Recently, PANS is regarded as an advanced turbulence model proposed by Girimaji [11], which bridges a connection between RANS model and DNS by two parameters, i.e., the unresolved-to-total ratios of kinetic energy (f_k) and dissipation (f_ω). Several cases utilizing PANS model, such as cylinder flows [12], cavitating flows around a hydrofoil [13] and internal flows in a low specific speed centrifugal pump [14], have proved its applicability in separated flow prediction compared with the traditional RANS models.

Although swirling flow downstream a runner or in a draft tube have been widely investigated for instability analysis, few studies investigated the flow upstream the impeller inlet in a centrifugal pump. Ref. [15] investigated the effect of non-uniform inflow on performance breakdown of a water-jet pump and presented the disturbance of the flow field at the impeller shroud. Ref. [16] analysed the non-uniform inflow characteristics in a canned nuclear coolant pump and pointed out that the swirling flow in a non-uniform channel would destroy the circular symmetry of the flow field and reduce the head and efficiency of the pump. However, the mechanism of non-uniform upstream impeller inlet has not been sufficiently explained in those researches.

Inspired by the previous study, this paper aims to clarify the effect for the flow upstream the impeller inlet on flow instability in a centrifugal pump by numerical simulation using the PANS model. The internal flows are compared for a test pump and its comparative pump with a straight inlet pipe. For better understanding, energy loss analysis is conducted to obtain a clear explanation of the results.

2. Governing equation for PANS model
The incompressible Navier-Stokes equations for the unsteady flow are described in the equation (1) and equation (2).
\[
\frac{\partial u_j}{\partial x_j} = 0
\]

\[
\frac{\partial u_j}{\partial t} + \frac{\partial}{\partial x_j}(u_iu_j) = -\frac{1}{\rho} \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}\left((\nu + \nu_t)\left(\frac{\partial u_j}{\partial x_j} + \frac{\partial u_i}{\partial x_i}\right)\right)
\]

where \(\rho\) is the density, \(p\) is the pressure, \(u_i\) is the velocity in the \(i\)-th direction, while \(u_j\) is the velocity in the \(j\)-th direction. \(\nu\) and \(\nu_t\) represent the laminar and turbulent kinematic viscosity, respectively.

For closure of the Reynolds stress in the RANS model, there are several turbulence models such as \(k-\varepsilon\) model, \(k-\omega\) model, RSM model, ASM model, etc. As for the standard \(k-\varepsilon\) model, two equations, i.e., \(k\) and \(\varepsilon\) equations are shown in the equation (3) and equation (4).

\[
\frac{\partial k}{\partial t} + V_j \frac{\partial k}{\partial x_j} = P - \varepsilon + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial k}{\partial x_j}\right)
\]

\[
\frac{\partial \varepsilon}{\partial t} + V_j \frac{\partial \varepsilon}{\partial x_j} = C_{\varepsilon_1} P \varepsilon - C_{\varepsilon_2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial \varepsilon}{\partial x_j}\right)
\]

where \(P\), \(k\) and \(\varepsilon\) stand for the production of turbulence kinetic energy, turbulence kinetic energy and turbulent dissipation, respectively. Some model constants are: \(C_{\varepsilon_1}=0.9; C_{\varepsilon_2}=1.44; C_{\varepsilon}^e=1.92; \sigma_k=1.0; \sigma_\varepsilon=1.3\).

For the PANS model with standard \(k-\varepsilon\) model as a parent model, the corresponding two equations of the turbulence closure are [11]:

\[
\frac{\partial k_u}{\partial t} + V_j \frac{\partial k_u}{\partial x_j} = P_u - \varepsilon_u + \frac{\partial}{\partial x_j}\left(\nu_u \frac{\partial k_u}{\partial x_j}\right)
\]

\[
\frac{\partial \varepsilon_u}{\partial t} + V_j \frac{\partial \varepsilon_u}{\partial x_j} = C_{\varepsilon_1} P_u \varepsilon_u - C_{\varepsilon_2} \frac{\varepsilon_u^2}{k_u} + \frac{\partial}{\partial x_j}\left(\nu_u \frac{\partial \varepsilon_u}{\partial x_j}\right)
\]

where \(P_u\) is the unresolved production term. \(\nu_u\) is the eddy kinematic viscosity formulated by \(\nu_u = C_{\nu_u} k_u^2 / \varepsilon_u\). \(k_u\) represents the unresolved turbulent kinetic energy and \(\varepsilon_u\) is the turbulent dissipation rate. Some other coefficients are:

\[
f_k = \frac{k_u}{k}, f_\varepsilon = \frac{\varepsilon_u}{\varepsilon}
\]

\[
\sigma_{k_u} = \sigma_k f_k^2; \sigma_{\varepsilon_u} = \sigma_\varepsilon f_\varepsilon^2; C_{\varepsilon_2} = C_{\varepsilon_1} + \frac{f_k}{f_\varepsilon} (C_{\varepsilon_2} - C_{\varepsilon_1})
\]

where \(f_k\) and \(f_\varepsilon\) are the unresolved-to-total ratios of kinetic energy and dissipation respectively.

For the flows with high Reynolds number, the small scales of dissipation is hardly resolved, \(f_\varepsilon\) is generally set to 1.0. As for the parameter of \(f_k\), Girimaji [11] proposed a spatially varying function as shown in equation (9), which is determined upon the local grid size and turbulence length scale.
\[
f_k(x) = \min \left\{ 1, \frac{1}{\sqrt{C_\mu}} \left( \frac{\Delta}{\Lambda} \right)^{\frac{2}{3}} \right\}
\]  

(9)

where \( C_\mu \) is a model coefficient. \( \Lambda \) is the Taylor turbulence length scale, which is defined as \( \Lambda=\frac{k^{1.5}}{\varepsilon} \). \( \Delta \) stands for the local grid size, defined as \( \Delta=(\Delta_x*\Delta_y*\Delta_z)^{1/3} \).

3. Problem description and setups

To investigate the effect of the flow upstream the impeller inlet on flow instability, two simulation models have been treated using the PANS model based on standard \( k-\varepsilon \) turbulence model as a parent model. One is the test model with a non-straight inlet pipe (figure 1(a)), which has been tested by experiment, and another is a comparative model with a straight inlet pipe (figure 1(b)). In addition, as depicted in the figure 1, computation domain includes the inlet pipe, impeller, gap, casing, outlet tube and extension pipe, which are established in UG NX 6.0. The only difference between these two pumps is the geometry of the inlet pipe, which is investigated individually. For the test pump, the specific speed is 39\( \text{min}^{-1}\cdot \text{m}^3\cdot \text{s}^{-1}\cdot \text{m} \), and the diameter at impeller exit is 142mm. Other relevant parameters have been shown in our previous research [1].

![Figure 1. Computational domain with (a) test pump and (b) comparative pump.](image)

In order to achieve high accuracy within a short time period, hexahedral meshes are applied in all components. A mesh independence test has been carried out in previous work [1]. The mesh consisting of 3.2 million hexahedral cells is adopted. The mesh is refined in areas of large variable gradients, i.e., near-wall. The minimum value of \( y^+ \) is 0.76 located near the interface between the trailing edge of the blade and the hub and the shroud.

In the current research, unsteady calculations have been conducted for the investigation using the commercial CFD code CFX. The time-dependent governing equations are discretized spatially and temporally. For boundary conditions, no-slip condition i.e., \( V_i^+=0 \) is imposed on the walls. Different
mass flow rates are prescribed at the inlet plane, and the static pressure based on the experimental data is assumed at the outlet plane. The time step is $3.3 \times 10^{-4}$ s, corresponding to a runner-rotating angle of $2^\circ$ per time step. In addition, the max residual convergence precision of the unsteady calculation at any flow rate is below $10^{-6}$.

4. Results and discussion

4.1 Characteristic curves

In order to validate the accuracy of the current PANS model, a comparison on the characteristic curves are carried out in the figure 2. In the characteristic curves, several non-dimensional parameters can be defined, i.e., head coefficient $\psi$, flow coefficient $\phi$, and hydraulic efficiency $\eta$, shown as equations (10)-(12).

$$\phi = \frac{Q}{\pi R_2 b_2 u_2}$$

(10)

$$\psi = \frac{H}{u_2^2/(2g)}$$

(11)

$$\eta = \frac{\rho g Q H}{P_{in}}$$

(12)

where $Q$ represents the flow rate, $H$ is the pump head. $P_{in}$ is the power input to the pump shaft; $R_2$ is the radius at the impeller exit, $b_2$ is the blade width at the impeller exit; and $u_2$ is the peripheral velocity at the impeller exit.

![Figure 2](image-url) Characteristic curves of the test pump and comparative pump.

In the figure 2(a), the experimental data of $\psi-\phi$ with the numerical results of the test pump and comparative pump are compared. In the figure 2(b), the results of $\eta-\phi$ are compared. As shown in the figure 2(a), the predicted head coefficient tendency presents a fairly good agreement with the experimental data at flow rate coefficients ranging from $\phi=0.0138$ to $\phi=0.0225$, especially near the design point condition ($\phi=0.0225$). For hump prediction, the test pump matches the experimental data
well. Two hump regions have been predicted: one ranges from $\phi=0.0100$ to $\phi=0.113$; other ranges from $\phi=0.0138$ to $\phi=0.163$, which coincide with the experimental data. The maximum prediction error is 3.98% located at $\phi=0.0100$. As for the results of the comparative pump, the head coefficient is slightly higher than the results of the test pump near the design point condition. In addition, an obvious hump hysteresis has been observed in the comparative model ranging from $\phi=0.015$ to $\phi=0.175$. Figure 2(b) denotes that the efficiency of the test pump. In general, overestimation in the test model is observed because the mechanical loss of all solid walls is neglected in the current study. The maximum prediction error is 7.16% located at $\phi=0.0113$, which is acceptable for this simulation. Similarly, the efficiency of the comparative pump is slightly higher than the test pump near the design point condition. While, under part-load conditions, the efficiency of the comparative pump is close to the test pump, even lower than the test pump under condition of $\phi_1=0.5\phi_{bep}$. The discrepancies attribute to the energy loss in inlet pipe and impeller.

4.2 Energy loss before impeller inlet

Having discussed the characteristic curves with two different pump models, some relevant analysis on the energy loss are conducted before impeller inlet.

Previously, Yan [7] has presented the one of the factors effecting the efficiency of hydraulic machinery, which is regarded as the entropy creation. This parameter has been utilized in turbomachinery. Yan [7] defined the entropy generation rate per unit volume, as shown in the equation (13), which represents the dissipation of mechanical energy into entropy.

$$\sigma = \frac{1}{T} \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\mu}{\rho} \frac{\partial \tilde{u}_j}{\partial x_i} \right)$$

(13)

where $\bar{T}$ is the average temperature. In the current study, the temperature is set as a constant. $\tilde{u}_i$ is a viscous stress tensor.

Figure 3. Contours of $\sigma$ on a cross section plane and other four planes (Pl.1~Pl.4).

Figure 3 shows the contours of the entropy generation rate per unit volume on a cross-section plane and other four planes (Pl.1~Pl.4) in two pump inlet pipes operated at $\phi_1=0.5\phi_{bep}$ and $\phi_2=0.75\phi_{bep}$. As
shown in the figure 3(a), under the low flow rate condition, i.e., $\phi_1=0.5\phi_{bep}$, one high entropy generation rate region is observed at the impeller inlet (Pl.2) of the comparative pump. This loss comes from the rotor-stator interaction which is also captured in the test pump. Another high value region is located at the entrance of the impeller. Whereas, in the test pump, the entropy generation rate in this region is relative lower. This discrepancy is associated with the internal flow in the impeller, which will be discussed further. Similarly, under the condition of $\phi_2=0.75\phi_{bep}$, the distribution of high entropy generation rate is similar in these two pump inlet pipes. A remarkable energy loss is observed at the corner of the non-straight inlet pipe, where a higher value of $\sigma$ is captured.

Figure 3. Helicity on the cross section of inlet pipe.

Figure 4 shows the distributions of relative helicity in different inlet pipes under the conditions of $\phi_1=0.5\phi_{bep}$ and $\phi_2=0.75\phi_{bep}$. The flow instability is associated with a relative helicity value. If the relative helicity is positive, the vortex will be promoted. Versus, the vortex will be alleviated. It is demonstrated in figure 4 that the relative helicity value near the solid wall is almost negative, which means that the shear flow will be alleviated. For comparison, the relative helicity in the straight inlet pipe is similar with the non-straight inlet pipe overall. However, obvious discrepancies have been captured at the impeller inlet, where the value of the relative helicity in the straight inlet pipe is slightly lower. The results match the figure 3 well.

As discussed above, the geometry of the inlet pipe is an important source of loss generation, especially at the corner of the inlet pipe where the inflow onsets. However, under the low flow rate condition, such as $\phi_1=0.5\phi_{bep}$, the loss in the straight inlet pipe is higher than that in the non-straight inlet. Under part-load condition, such as $\phi_2=0.75\phi_{bep}$, a slight discrepancy is observed. For explanation, further attention should be paid to the internal information in the impeller.

4.3 Energy loss analysis
In order to obtain the energy loss in the two pump models, and reveal the flow instability with different inflow upstream the impeller, energy loss analysis will be carried out further based on the energy equation. The energy equations are as follow:
\( P_{in} = P_e + P_l \)  \( (14) \)

\[
P_l = -\rho g \Delta H Q_v = -\iiint_V \frac{\partial(-u_i, \rho u_i)\mu}{\partial x_j} dV - \iiint_V \mu \frac{\partial(u_i D_{ij})}{\partial x_j} dV
+ \iiint_V (-\rho u_i u_j) \frac{\partial u_i}{\partial x_j} dV + \iiint_V \mu D_{ij} \frac{\partial u_i}{\partial x_j} dV \]
\( (15) \)

where \( P_e \) is the total pressure. \( P_l \) represents the effective power and \( P_l \) stands for the power loss. \( \Delta H \) represents the head loss. In addition, \( D_{ij} \) is a shear strain tensor. And the Reynolds stress is organized based on the Boussinesq approximation.

As depicted in equation (15), the energy loss can be treated as an integration of the transport term of total pressure, which can be expressed as a superposition of four terms of the right side of equation (15). The first term \( (P_l1) \) and the second term \( (P_l2) \) are the diffusion of the Reynolds stress and viscous stress, respectively, which means that the external flow with high speed would be transferred to the separated flow with low speed by means of Reynolds stress and viscous stress. The third term \( (P_l3) \) represents the turbulent kinetic energy production, which means the flow kinetic energy would be transferred to the turbulent kinetic energy. And the last term \( (P_l4) \) is the viscous dissipation of the mean kinetic energy. This term stands for the contribution of the viscous dissipation on the energy loss.

According to the equation (15), the energy loss distributions in different pump components are calculated. Results are shown in the figure 5.

The energy loss coefficient \( \lambda_i \) is defined in equation (16):

\[
\lambda_i = \frac{P_l}{\left(\rho \pi R_e^2 b_2 u_2^3\right)/2}
\]  \( (16) \)

**Figure 5.** Energy loss in different pump components under two part-load conditions.

As demonstrated in the figure 5, in a centrifugal pump, large energy losses occur in the impeller and in the casing. In different pump models, the energy loss in the casing almost remains the same under the
condition of $\phi_2=0.75\phi_{bep}$. However, the energy loss varies in the impeller remarkably, resulting in a discrepancy in the total energy loss. It is noted that the flow rate of $\phi_2=0.75\phi_{bep}$ locates at the hump region in the comparative pump instead of the test pump. Thus, it is notable that under this condition, the energy loss in the comparative pump is slightly higher than that in the test pump, which indicates that the energy loss under the hump condition is higher than that under non-hump condition. The results match with the previous discussion. Under a low flow rate condition, especially, the energy loss in the comparative pump is much higher than that in the test pump. The internal flow in the impeller accounts for this phenomenon.

Based on the previous research [1], the periodical evolution of rotating stall cells can be captured under the conditions of $\phi=0.5\phi_{bep}$ and $\phi=0.75\phi_{bep}$ in the test pump. Figure 6 shows the time-averaged contours of the turbulent kinetic energy production ($P_{\text{t}}$) in both pumps under two typical part-load conditions. Results show a relative high value of $P_{\text{t}}$ locates at the inlet of the impeller and at the outlet of the impeller in the test pump where the rotating stall cells grow and shed. This indicates that flow separations (low speed) at the inlet of the impeller and jet flow (high speed) at the exit of the impeller could induce high energy loss, transferring kinetic energy to turbulent kinetic energy. Although the distribution of energy loss in the comparative pump under the condition of $\phi=0.75\phi_{bep}$ is similar with the results of the test pump, the distribution of energy loss under the condition of $\phi=0.5\phi_{bep}$ is different from the test pump. Uneven and high energy loss is observed at three blade-to-blade passages, i.e., passage 3, passage 4 and passage 5, where the large-scale vortices occurs. It is can be also indicated in the figure 6 that the energy loss at exit of the impeller is higher in the blade passages, where the vortices are in the stationary state with very low speed. For a detailed analysis, streamlines distribution during 3 rotational period of the impeller ($T$) in the comparative pump under the condition of $\phi=0.5\phi_{bep}$ is displayed in the figure 7.

![Figure 6](image_url)

**Figure 6.** Time-averaged energy loss contours on the mid-span section of the impeller.

As shown in figure 7, passage 3, 4 and 5 are totally blocked off by the vortices induced by separation flow at each instant. Note that in three rotation circles of the impeller, these vortices almost keep the same state. Two large-scale vortices, one is near the suction side of the blade, almost blocking off the
inlet passage of the impeller; another is near the outlet of the impeller, blocking the outlet passage. Different with the results under condition of $\phi = 0.75 \phi_{\text{bep}}$, i.e., an obvious periodical propagation of rotating stall cells can be obtained in both of the test pump and the comparative pump. Thus, we can conclude that the discrepancies in energy loss are due to the characteristic movement of the vortices. Higher energy loss occurs when the stall cells are in a stationary state, i.e., stationary stall cells.

The main reason for the difference of the flow characteristics in the impeller under small flow rate condition is the geometry of the inlet pipe. It is easy to observe the occurrence of strong pre-swirling upstream the impeller in the non-straight pipe as displayed in the figure 8. As demonstrated in the figure 8(a), the streamlines distribution and velocity vector on the $P_{1,2}$ in the non-straight pipe is extremely turbulent under this low flow rate condition, which generates strong pre-swirling upstream the impeller inlet. On the contrary, the flow in the straight inlet pipe is smoother.

**Figure 7.** Streamline distribution in the comparative pump under condition of $\phi_1 = 0.5 \phi_{\text{bep}}$.

**Figure 8.** The streamlines distribution and velocity vector on $P_{1,2}$ with different inlet pipes under
condition of $\phi=0.5\phi_{bep}$.

Further validation has been conducted through frequency spectra of pressure coefficient amplitude analysis on two monitoring points. Results are plotted in the figure 9.

![Figure 9. Pressure fluctuations in the impeller under two part-load conditions.](image)

The relevant location of test points under each condition has been plotted in the figure 7. Point 1 is located in the pressure side of the blade in the passage 2 and it is tested for the condition of $\phi=0.5\phi_{bep}$. Point 2 is located in the passage 5 of the impeller and it is tested for the condition of $\phi=0.75\phi_{bep}$. As plotted in the figure 9(a), a dominant, low-value frequency, i.e., rotating stall frequency ($f_{\text{stall}}$) has been captured in the test pump under condition of $\phi=0.5\phi_{bep}$, which is 17.58% of the rotational frequency of the impeller. Whereas, the frequency spectrum in the comparative pump denotes no low frequency signals. In figure 9(b), both pumps, predict a dominant, low-value frequency ($f_{\text{stall}}=8.76\%f_n$) under condition of $\phi=0.75\phi_{bep}$. This proves that the geometry of the inlet pipe hardly effects the characteristic of the internal flow under hump condition, such as $\phi=0.75\phi_{bep}$. Results in the figure 9 indicate the internal flow is much distinct under low flow rate condition, such as $\phi=0.5\phi_{bep}$. Under this condition, the pre-swirling flow in the non-straight inlet pipe plays an important role in the rotation of the stall cells.

5. Conclusions

In this paper, a partially-averaged Navier-Stokes (PANS) model based on the standard $k-\varepsilon$ model is applied to investigate the effect of the flow upstream the impeller inlet on the flow instability in a centrifugal pump. The main findings are listed as follows:

1) The unsteady turbulent flows simulated using PANS model are acceptable compared with the experiment data of the test pump. Thus, it is applicable to investigate flow instability in a centrifugal pump using the current numerical method, i.e., PANS model.

2) According to the characteristic curves, the hump phenomenon has been observed at $0.75\phi_{bep}$ for the comparative model, while it also occurs at $0.5\phi_{bep}$ for the test pump.

3) Under the condition of $\phi=0.5\phi_{bep}$, there is strong pre-swirling flow upstream impeller inlet for the
test pump with non-straight inlet pipe, which contributes to the propagation of rotating stall cells. On the contrary, in the comparative pump with straight inlet pipe, there are flow separation as well as stationary stall cells, resulting in high energy loss.

Acknowledgment
This work was financially supported by National Key R&D Program of China (2018YFB0606101), the National Natural Science Foundation of China (Grant Nos 91852103 and 51776102), Beijing Natural Science Foundation (3182014), State Key Laboratory for Hydrosience and Engineering (Project No. sklhse-2017-E-02) and the Tsinghua National Laboratory for Information Science and Technology.

References
[1] Ye W X, Huang R F, Jiang Z W, Li X J, Zhu Z Z and Luo X W 2019 Instability analysis under part-load conditions in centrifugal pump J. Mech. Sci. Technol. Vol. 33 No. 1, pp. 269-278.
[2] Rayleigh 1917 On the dynamics of revolving fluids Proc. R. soc. Lond. Ser. A-Contain. Pap. Math. Phys. Character Vol. 93, No. 648, pp. 148-154.
[3] Batchelor G K 1964 Axial Flow in Trailing Line Vortices J. Fluid Mech. Vol. 20, No. 4, pp. 645-658.
[4] Yoshida Y, Murakami Y, Tsurusaki H and Tsujimoto Y 1991 Rotating Stalls in Centrifugal Impeller/Vaned Diffuser Systems: 1st Report, Experiment Proc. First ASME/JSME Joint Fluids Eng., Conf. Portland.
[5] Sinha M, Pinarbasi A and Katz J 2001 The flow structure during onset and developed states of rotating stall within a vaned diffuser of a centrifugal pump J. Fluids Eng.-Trans. ASME Vol. 123, No. 3, pp. 490-499.
[6] Xia J L, Yadigaroglu G, Liu Y S, Schmidli J and Smith B L 1998 Numerical and experimental study of swirling flow in a model combustor Int. J. Heat Mass Transf. Vol. 41, No. 11, pp. 1485-1491.
[7] Yan P, Chu N, Wu D Z, Cao L L, Yang S and Wu P 2017 Computational Fluid Dynamics-Based Pump Redesign to Improve Efficiency and Decrease Unsteady Radial Forces J. Fluids Eng.-Trans. ASME Vol. 139, No. 1, pp. 01101.
[8] Yaras M I and Grosvenor A D 2003 Evaluation of one- and two-equation low-Re turbulence models. Part I - Axisymmetric separating and swirling flows Int. J. Numer. Methods Fluids Vol. 42, No. 12, pp. 1293-1319.
[9] Gnanaskandan A and Mahesh K 2016 Large Eddy Simulation of the transition from sheet to cloud cavitation over a wedge Int. J. Multiph. Flow Vol. 83, pp. 86-102.
[10] Foroutan H and Yavuzkurt S 2014 A partially-averaged Navier-Stokes model for the simulation of turbulent swirling flow with vortex breakdown Int. J. Heat Fluid Flow Vol. 50, pp. 402-416.
[11] Girimaji S S 2006 Partially-averaged Navier-Stokes model for turbulence: A Reynolds-averaged Navier-Stokes to direct numerical simulation bridging method J. Appl. Mech.-Trans. ASME Vol. 73, No. 3, pp. 413-421.
[12] Lakshminarayanan T and Girimaji S S 2010 Partially Averaged Navier-Stokes (PANS) Method for Turbulence Simulations: Flow Past a Circular Cylinder J. Fluids Eng.-Trans. ASME Vol. 132, No. 12, pp. 121202.
[13] Kadivar E, Moctar O E and Javadi K 2018 Investigation of the effect of cavitation passive control on the dynamics of unsteady cloud cavitation *Appl. Math. Model* Vol. 64, pp. 333-356.

[14] Liu J T, Zuo Z G, Wu Y L, Zhuang B T and Wang L Q 2014 A nonlinear Partially-Averaged Navier-Stokes model for turbulence *Comput. Fluids* Vol. 102, pp. 32-40.

[15] Cao P Y, Wang Y, Xiang Z, Li G D and Yin G 2017 Concentrated separation vortex induced by non-uniform suction flow and its role in performance breakdown of a water-jet pump *Adv. Mech. Eng.* Vol. 9, No. 6, pp. 1-14.

[16] Wang Y H, Wang P F, Tan X H, Xu Z B and Ruan X D 2018 Research on the non-uniform inflow characteristics of the canned nuclear coolant pump *Ann. Nucl. Energy* Vol. 115, pp. 423-429.