Numerical analysis of rotating stall instabilities of a pump-turbine in pump mode

L S Xia, Y G Cheng, XX Zhang and J D Yang

State Key Laboratory of Water Resources and Hydropower Engineering Science, Wuhan University, Wuhan 430072, China

E-mail: xialinsheng_123@sina.com

Abstract. Rotating stall may occur at part load flow of a pump-turbine in pump mode. Unstable flow structures developing under stall condition can lead to a sudden drop of efficiency, high dynamic load and even cavitation. CFD simulations on a pump-turbine model in pump mode were carried out to reveal the onset and developed mechanisms of these unstable flow phenomena at part load. The simulation results of energy-discharge and efficiency characteristics are in good agreement with those obtained by experiments. The more deviate from design conditions with decreasing flow rate, the more flow separations within the vanes. Under specific conditions, four stationary separation zones begin to progress on the circumference, rotating at a fraction of the impeller rotation rate. Rotating stalls lead to the flow in the vane diffuser channels alternating between outward jet flow and blockage. Strong jets impact the spiral casing wall causing high pressure pulsations. Severe separations of the stall cells disturb the flow inducing periodical large amplitude pressure fluctuations, of which the intensity at different span wise of the guide vanes is different. The enforced rotating non-uniform pressure distributions on the circumference lead to dynamic uniform forces on the impeller and guide vanes. The results show that the CFD simulations are capable to gain the complicated flow structure information for analysing the unstable characteristics of the pump mode at part load.

1. Introduction

Reversible pump-turbines in pumped-storage plants deliver peak energy in turbine mode to meet the demands of power grids and utilize excess energy available in grids in pump mode. Modern pump storage plants need pump-turbine higher flexibility and reliability in operation under off-design conditions, especially in the pump mode, [1]. When a pump-turbine operates at part load in pump mode, flow separations within the flow passages form unstable flow structures such as stationary vortex and rotating stall. Rotating stall is a highly dynamic phenomenon for substantial flow fluctuations propagating at a low frequency along the circumference in the vane diffuser or impeller. The rotating flow on the circumference could exert large dynamic radial forces and torque variations on the impeller which might lead to rotor-dynamic instabilities. Due to the low frequency characteristic of the rotating stall, the perturbation caused by it may resonate with water conveyance system of the pumped-storage plants, which may threat the safety operation of the plants [2].

Rotating stall may occur in any turbo-machinery, and this phenomenon is most frequently studied in centrifugal compressors. Nevertheless, the research and publications regarding hydraulic machinery,
especially about pump-turbine, remain still limited. Krause et al. [4] and Sinha et al. [5] used Particle Image Velocimetry (PIV) to analyze the jet and reverse flow patterns in the vane diffuser of the centrifugal pump under rotating stall condition. Berten et al. [6] performed pressure fluctuation measurements in a high-energy centrifugal pump and they found that the stall may be stationary or rotating with different rotational speeds. Hasmatuchi et al. [7] injected air bubbles into the vaneless gap between the impeller and the guide vanes, and used a special image processing technique to visualize the flow patterns of the rotating stall.

At present, study methodologies on the rotating stall problem are mainly through the analysis the time signals of pressure or velocity by experimental measurements, with transducers mounted flush with the wall. Although the measurements are reliable in detection of the stall, they lack the ability to visualize the complicated flow structure evolution mechanism in the whole flow space directly. Three-dimensional (3-D) CFD simulations not only can offer all flow variables in time and space, while measurements are often difficult or even impossible, but also can track the development and movement of 3-D stall cells in space and time. Lucius A et al. [8] used CFD numerical simulation to investigate the stall phenomenon in a pump, and mainly investigated the effectiveness of different turbulence models and mesh scales in simulating this complex flow patterns. Widmer C et al. [9] conducted CFD simulations and test rig measurements of a pump-turbine model in the turbine brake operation. The results showed that the stationary vortex formation and rotating stall within the turbine and vane-less space both increase the total pressure difference over the machine. The increased pressure can be the cause for the “S-shape” characteristic which is responsible for hydraulic system oscillations in the hydropower plants.

In the present paper, 3-D CFD simulations were carried out to analysis the rotating stall phenomena within the guide vanes of a pump-turbine model in pump mode. The analysis is focused on the onset and development mechanism of the flow instabilities, the relations between outward velocity distribution of the impeller and pressure pulsations intensity distribution in space and time, and the influence of dynamic load on the impeller and guide vanes under rotating stall condition.

2. Simulation Model and numerical methods

2.1. Simulation model

A low specific speed pump-turbine model is studied, featuring \( n_{st} = 20 \) stay vanes (including tongue), \( n_{gv} = 20 \) guide vanes and \( Z_b = 9 \) impeller blades. The diameter of the impeller inlet, \( D_2 \), is 140.9mm and that of the impeller outlet, \( D_1 \), is 280mm. The computational domain is composed of spiral case, vane diffuser, impeller, and draft tube. The impeller rotational speed, \( \omega \), is 1200rpm, and the guide vanes opening is 24 degree.

2.2. Turbulence model and boundary conditions

The numerical simulations were performed with FLUENT code. Since the rotating stall is a strong turbulence phenomenon, a turbulence resolving model is expected to gain this unstable stall cells superiorly in spatial and temporal. Based on a comparative analysis, the scale-adaptive simulation (SAS) model is adopted, which introducing the von Karman length-scale into the turbulence scale equation. The information provided by the von Karman length-scale allows SAS models to dynamically adjust to resolved structures in the unsteady Reynolds Averaged Navier-Stokes (U-RANS) simulation, which results in a LES-like behaviour in unsteady regions of the flow field. At the same time, the model provides standard RANS capabilities in stable flow regions.

For the pressure interpolation, PRESTO! Scheme, which is highly recommended for a pressure-based solver, was used. For both the steady and transient flow in this paper, SIMPLEC algorithm was chosen to achieve the coupling solution for the velocity and pressure equations.

The boundary conditions were defined for the present simulations as follows: mass flow rate was defined at the inlet of the draft tube, and pressure out boundary with static pressure was applied at the outlet of the spiral case. The remaining boundary conditions are imposed by the no-slip wall boundary.
2.3. Grid and time step

Non-uniform grids were generated at different domain of the pump-turbine with approximately \(8.2 \times 10^6\) elements. The tetrahedral grids were used in the spiral case, wedge grids were used in the vane diffuser, and the structured grids were applied in the impeller and draft tube. Special refinement was applied in the impeller (3.2 million elements) and in the vane diffuser domain (2.8 million elements) to ensure the grid resolution for capturing the severe separation structure and the distance of the first grid points to the wall.

In the process of numerical simulation, the results of steady-state flow calculation were used as the initial flow field for the unsteady calculation. For the unsteady simulation, the time-step was adjusted to 0.00015625s, corresponding to 320 time steps per revolution. The convergence rate for each time-step is set to \(10^{-5}\), and the maximum number of iterations per time-step is set to 40.

A mesh sensitivity analysis revealed that the deviates of the pump head obtained between the above computational grid and the finer meshes were less than 1%.

2.4. Schematic diagram of monitor points

In order to analyze the distribution and propagation of the pressure waves within the vane diffuser channels under rotating stall condition, twenty lines with equal angular intervals (18 degree) are set at three circular sections, with different radius of \(r_1\), \(r_2\) and \(r_3\). According to the radius of the circular from small to large \((r_1 < r_2 < r_3)\), the three sections are named as n, z and w. Every line has three monitoring points at different height along the vanes, with the span-wise=0.1, 0.5, and 0.9, respectively. As shown in Fig.2, the three monitoring points are located near the hub (upper annular disc), at the mid span plane, and near the shroud (lower annular disc), respectively. Their locations are named as s, z and d, respectively. Specific name rule of the monitoring points is demonstrated in Fig.1 and Fig.2.

3. Results and Discussions

To verify the accuracy of the calculation, a comparison between the CFD simulation and experimental performance curves for the pump-turbine in pump mode is shown in Fig. 3. The experimental data was provided by HARBIN ELECTRIC CORPORATION. In the figure, the flow coefficient \(\psi\) and total head coefficient \(\phi\) are defined as \(\psi = \frac{2gH}{\omega^2 R_2^2}\) and \(\phi = \frac{Q}{\pi \omega R_2^3}\), respectively. Where \(R_2\) is the radius of the impeller inlet, \(H\) is the head of pump-turbine. The characteristics curves of the CFD simulation are both based on the unsteady calculation results. Fig.3 shows that predicted pump characteristic curves by the CFD agrees well with the experimental results.
At best efficiency operating point with $\phi = 0.362$, the operating condition is named as OP1. At part load condition, $Q = 0.8Q_{BEP}$, namely $\phi = 0.290$, rotating stall occurs within the vane diffuser channels.

Figure 3 shows that the numerical predicted efficiencies are higher than that of the experiments. This is because the numerical predicted efficiency only considered the torque within the rotating impeller while ignored the mechanical and leakage losses arise in the actual pump-turbine model.

3.1. Pressure pulsations and flow patterns at OP1

3.1.1. Analysis of pressure pulsations in time and frequency domains

At the best operating point, OP1, the pressure pulsations at the periphery of the vaneless space between the impeller and guide vanes are mainly induced by the rotor-stator interaction (RSI) between the impeller blades and guide vanes, seeing Fig.4. The black arrow line shown in Fig.4 illustrates the angular travelling passage of the pressure peaks in time. In addition, the arrowhead direction demonstrates that one pressure wave propagates in the opposite direction of impeller revolution, with an angular velocity equalling to the impeller rotational velocity $\omega$.

The frequency spectra of the pressure pulsations on section $n$ at different span-wise of guide vanes are presented in Figs. 5(a)-(c). The frequency spectra distributions show that the dominant frequency of the pressure pulsations is the blade passing frequency ($f = 9f_n = 180 \text{ Hz}$), whereas the amplitudes of pressure pulsations in the spanwise direction are significant different. The pressure pulsation amplitude at the mid span plane is higher than that of the monitoring points near the hub and shroud, and also the pulsation amplitude near the hub is lowest. The frequency spectra of the pressure pulsations also indicate the effect of tongue which is the closing part of spiral case. The effect is distinct at the mid span-wise where the amplitude of the pressure pulsations is higher near the tongue,
and decreases along the circumferential direction which is opposite to the impeller rotation direction, seeing Fig.5 (b).

### 3.1.2. Flow patterns analysis

The flow patterns at the best efficiency operating point are smooth and regular. Nevertheless, there are different separation zones at the tailing edge at different span plane of the guide vanes, as shown in Fig.7. In order to analyse the cause of the difference, velocity variations of three motoring points on one line at section n are presented in Fig.6. The velocity field fluctuates regularly and periodically due to the RSI between impeller and guide vanes, but the velocity amplitude and the outflow angle of the impeller are different. At the mid span, the angle $\theta$ between the tangential velocity and absolute velocity is the smallest, which is due to that the radial velocity is the smallest, and the tangential component is the largest. Smaller angle $\theta$ means a higher inflow incidence angle at the guide vanes, resulting in the flow separations at the mid span occurring more easily at the suction side of the guide vanes, as can be seen in Fig.6 (b) and Fig. (7). As shown in Fig.6(c), the amplitude of the velocity fluctuation at the mid span is the largest and the variation rate is the fastest. Most rapid changes in flow field at the mid span lead to the strongest pressure pulsations. Contrary to the flow patterns at the mid span, the smallest amplitude of velocity fluctuation and the slowest velocity variation rate near the hub lead to the lowest amplitude of pressure pulsations. The above analysis about the correlation between the velocity field and intensity of pressure pulsations at different span-wise is coincide with the spectra analysis of pressure pulsations shown in Fig.5. The impeller outlet velocity distribution may be related to the geometry of the impeller, which is beyond the scope of this paper.

![Fig.6 velocity of monitoring points on 1#line on section n at different span-wise](image)

![Fig.7 OP1: velocity vectors of different span-wise plane at t=0.2375s](image)

#### 3.2. Rotating stall within vane diffuser
3.2.1. Analysis of pressure pulsations

At part load operating condition, OP2, the pressure pulsations within the vane diffuser are dominated by a low frequency component with high fluctuation amplitude, which carries the frequency component of the RSI. The variations of pressure pulsations on the circumference in Fig.8 show a remarkable periodicity. The low-pass filtered signals of pressure pulsations demonstrate the high pressure instability source rotating with the impeller at sub-synchronous frequency. The characteristics of the pressure signals manifest the rotating stall within the vane diffuser channels.

![Original signals](image1)

![Low-pass filtered signals](image2)

**Fig.8** Circumferential distribution of pressure pulsations of monitoring points nd, at OP2

The rotation frequency \( f_s \) of rotating stall can be estimated by the slope of a line connecting the pressure peak of the filtered pressure signals, as seen in Fig.8 (b). For the angle \( \alpha \), the calculation is as follows:

\[
\frac{1}{\tan \alpha} = \frac{2\pi}{T_s} = f_s
\]

The results are \( T_s \approx 1.134s, \ f_s = 0.882 \text{ Hz} \approx 0.044 \ f_n, \) where \( T_s \) is the rotating stall period and \( f_s \) is the rotation frequency of impeller.

Frequency spectra in Fig.9 shows two obvious frequency peaks, which are the blade passing frequency at 180Hz and the low frequency caused by rotating stalls at 3.539Hz = \( 4 f_s = 0.176 \ f_n \), respectively. For the low frequency equalling to \( 4 f_s \), the spectra indicates that there are four stall cells within the vane diffuser simultaneously. The rotation frequency of rotating stall and the number of stall cells are in agreement with the results of experiment in a pump-turbine model by Braun [10].

As shown in Fig.9, the low frequency is the dominated frequency with larger amplitude comparing to that of the RSI, but the intensity difference of low frequency instability between different span-wise planes is obvious. The pressure amplitude caused by rotating stall at mid span is higher than that near hub and shroud, which is similar to the intensity distribution of pressure pulsations at OP1. The primary cause of the pressure intensity difference can be identified by the flow patterns of the stalls. As shown in Fig.12, the flow in the mid span is severely blocked due to intense flow separations on the suction side of the guide vane, while the flow near the hub and shroud is less disturbed. The velocity distributions of the impeller outflow at different span-wise are presented in Fig.11. It is obvious that the radial velocity at the mid span and the angle between the absolute velocity and tangential component is the smallest, which implies that the stall cell firstly occurs at the mid span of guide vane channel for a larger incidence angle. Larger incidence angle leads to higher intensity of separation, which causes higher intensity pressure fluctuations. The radial velocity of the impeller outflow behaves sudden intensive fluctuations (see the red frame marker in Fig.11 (a)) before reaching its minimum data, which indicates that the stall cells occur with strong reverse flow.

The distribution of pressure pulsations intensity along one guide vane channel at the mid span is shown in Fig.9 (b) and Figs.10 (a) - (b). The location of maximum amplitude of pressure fluctuations
caused by rotating stall is at the guide vane throat, namely at the position of the circular section z. The amplitude of the sub-synchronous frequency pressure pulsations attenuated towards the guide vanes out. The pressure pulsations intensity for RSI is relatively low and nearly vanished at the outlet of the guide vanes.

![Frequency spectra of pressure pulsations on circular section n at different span plane](image)

**Fig.9** Frequency spectra of pressure pulsations on circular section n at different span plane

![Frequency spectra of pressure pulsations on circular sections z and w](image)

**Fig.10** Frequency spectra of pressure pulsations on circular sections z and w

![Variations of velocity and the angle between absolute velocity and its tangential component on l#line of section n at different span-wise at OP2](image)

**Fig.11** Variations of velocity and the angle between absolute velocity and its tangential component on l#line of section n at different span-wise at OP2

3.2.2. Flow patterns under rotating stall

At part load operating condition OP2, four rotating stall cells distribute equally around the circumference within the guide vane channels, seeing Fig13 (a). Due to the stall cells almost block the flow channel, more fluid has to flow through the next channel in rotational direction which leads to strong jets running into the spiral case. As shown in Fig.13 (b), the rotating high speed jets cause high pressure impingement on the spiral case wall periodically, which may adversely affect the stability of the structure.

In order to better understand the flow structures within the vane diffuser channels under rotating stall, $Q$-criterion [11] is used to identify the vortex structures, which is defined as the second invariant of the velocity gradient tensor. It is defined as $Q = 1/2(\Omega_y^2 - \Omega_y S_j S_j)$, where $\Omega_y$ is rate-of-rotation
tensor, $S_{ij}$ is rate-of-strain tensor. An instantaneous isosurface of the $Q$-criterion in the vane diffuser channels is shown in Fig.14. The usage of this identification can visualize the inception and evolution of the rotating stall in 3-D space. The vortex structures compose streamwise vortices, horseshoe vortices, and spanwise vortices. The vortices grow from the leading edge of the guide vane suction side at mid span, and are stretched into the elongated streamwise vortices and horseshoe-shaped vortices by flow field. The turbulent intensity of the vortex structures at the mid span range is larger relative to the forward flow field near hub and shroud, which also explain the cause that the amplitude of pressure pulsation in the vicinity of the mid span is higher than the positions near hub and shroud.

(a) Span-wise=0.1 (b) Span-wise=0.5 (c) Span-wise=0.9

**Fig.12** Instantaneous velocity vectors at different span-wise

(a) Streamlines (b) Pressure distribution

**Fig.13** Instantaneous streamlines and the corresponding pressure distribution at the mid span

**Fig. 14** Flow vortex structures within vane diffuser channels ($Q=700000\text{s}^{-2}$)
The instantaneous flow patterns in different vane diffuser channels demonstrate the evolution process of the rotating stall. The channel 2 between guide vane A and B is almost fully blocked with heavy separations from the leading edge of suction side to the tailing edge. The shedding vortices on the suction side of vane A can extend into the following channel 3, which may intensify the leading edge vortex separation of vane B and deviate the approaching flow to the next channel. In the preceding channel 1, the flow is re-established with the leading edge vortex separation disappearing. The vortices in the channel 1 will be flushed out for a decreasing incidence angle.

3.2.3. Radial force and torque

At part load operating condition, the non-uniform circumferential pressure distribution acting on the impeller can result in a large displacement of the rotor shaft. This displacement changes the shaft procession orbit which might lead to rotor-dynamic instabilities. At best efficiency operating point OP1, the fluctuations of radial force on the impeller are weak, and primarily caused by the RSI, seeing Fig. 15(a). Under rotating stall condition, a stronger sub-synchronous fluctuation superposes on the variations of radial force caused by RSI, which is due to enforced rotating non-uniform pressure distributions on the circumference within the vane diffuser. The amplitude of the force in y direction, (the direction is perpendicular to the outflow direction of the spiral case) is significantly larger than that in the x direction, as shown in Figs.15 (b)-(c). This considerable difference may relate to the spiral case shape.

Fig.15 Variations of radial forces at different operating conditions

Figures.16 (a)-(b) present the frequency spectra of impeller blade torque at OP1 and OP2, respectively. At best operating point OP1, the dominated frequency is the harmonic frequency \((0.25n_{g}f_{s})\) of the guide vanes passing frequency. At part load condition OP2, the rotating stall has a significant influence on the impeller blade torque, leading to large amplitude fluctuations. The dominated frequency is \(76.47Hz = N \times (f_{s} - f_{p})\), where \(N=4\) is the number of stall cells.

The rotating stall changes the torque of the guide vanes periodically. At the stall zone, the flow patterns are different on both sides of a guide vane that on one side the flow is blocked while on the other side it is relative smooth. The flow patterns difference causes the pressure difference, which
changes the guide vanes torque periodically. The sub-synchronous frequency of guide vanes torque fluctuations caused by the rotating stall is coincide with the sub-synchronous frequency of the pressure pulsations, as shown in Fig.16(c).

4. Conclusions
The rotating stall of a pump-turbine in pump mode was simulated and analyzed by CFD in this paper. By investigating the results, the following conclusions could be drawn:

- The inflow is non-uniform in the span wise direction of the guide vanes. In addition, the rotating stall starts from the leading edge on the suction side at mid span, and then extends to the hub and shroud.
- The separation intensity is strongest at mid span in the spanwise direction, which leads to the highest amplitude of pressure pulsations. Moreover, the maximum amplitude of pressure fluctuations caused by rotating stall is at the guide vane throat along the guide vane channels.
- Under rotating stall condition, the enforced rotating non-uniform pressure distribution on the circumference acting on the impellers can result in a stronger sub-synchronous fluctuation of radial force, which leads to rotor-dynamic instabilities and causes strong vibrations. Moreover, the difference of pressure fluctuations on both sides of the guide vanes induce torque variations, which may have adversely influence on the machine operation.
- The rotating high speed jets could cause high pressure impingement on the spiral case wall periodically, which may lead to structural vibrations.

Acknowledgments
This work was supported by the National Natural Science Foundation of China (Grant No. 51039005).

References
[1] Anciger D, Jung A and Aschenbrenner T 2010 Prediction of rotating stall and cavitation inception in pump turbines 25th IAHR Symposium on Hydraulic Machinery and Systems (Timisoara, Romania)
[2] Brennen C E 2011 Hydrodynamics of pumps (Lodon:Cambridge University Press)
[3] Krause N Zähringer K and Pap E 2005 Time-resolved particle imaging velocimetry for the investigation of rotating stall in a radial pump Experiments in Fluids 39(2) 192-201
[4] Sinha M Pinarbası A and Katz J 2001The flow structure during onset and developed states of rotating stall within a vaned diffuser of a centrifugal pump J. of fluids engineering 123(3) 490-499
[5] Sano T Yoshida Y Nakamura Y 2002 Numerical study of rotating stall in a pump vaned diffuser J. of fluids engineering 124(2) 363-370
[6] Berten S Dupont P and Fabre L 2009 Experimental investigation of flow instabilities and rotating stall in a high-energy centrifugal pump stage ASME Fluids Engineering Division Summer Meeting (Colorado American)
[7] Hasmatuchi V Farhat M and Roth S 2011 Experimental evidence of rotating stall in a pump-turbine at off-design conditions in generating mode J. of Fluids Engineering 133(5) 051104.
[8] Lucius A Brenner G 2011 Numerical simulation and evaluation of velocity fluctuations during rotating stall of a centrifugal pump J. of Fluids Engineering 133(8) 081102
[9] Widmer C Staubli T and Ledergerber N 2011 Unstable characteristics and rotating stall in turbine brake operation of pump-turbines J. of fluids engineering 133(4) 041101.
[10] Braun O 2009 Part load flow in radial centrifugal pumps Ph.D. Thesis (Laboratory of Hydraulic Machinery, Swiss Federal Institute of Technology, Lausanne)
[11] Jeong J and Hussain F 1995 On the identification of a vortex J. of fluid mechanics 285 69-94