Unsteady numerical analysis of the rotating stall in pump-turbine geometry

U Ješe¹ and R Fortes-Patella²

¹Litostroj Power d.o.o., Ljubljana, Slovenia (previously at Univ. Grenoble Alpes, LEGI)
²Univ. Grenoble Alpes, LEGI, F-38000, Grenoble, France

E-mail: uros.jese@litostrojpower.eu

Abstract. Main challenges in energy sector nowadays are storing and recovering of a large amount of energy in a short time. Pumped Storage Power Plants (PSP), using reversible pump-turbines are among the most cost-efficient solution to answer these needs. To provide a rapid adjustment to the electrical grid, pump-turbines are subjects of quick switching between pumping and generating modes and to extended operation under off-design conditions. To maintain the stability of the grid, the continuous operating area of reversible pump-turbines must be free of hydraulic instabilities. One of the main sources of pumping mode instabilities is the presence of the rotating stall that occurs at the part load. It can be observed as periodic occurrence and decay of recirculation zones in the distributor regions. Consequently, the machine can be exposed to uncontrollable shift between the operating points with the significant discharge modification and the drop of the efficiency. The phenomenon is very complex, three-dimensional and demanding for the investigation. The paper presents cost-efficient numerical methodology that enables the accurate prediction and analysis of the rotating stall. The investigations were made on a reduced-scaled high head pump-turbine design. Unsteady numerical calculations were performed using code FINE/Turbo™ and URANS equations. Local flow study was done to describe in details the governing mechanisms of the rotating stall. The analyses enable the investigations of the rotating stall frequencies, the number of stalled cells and the intensity of the rotating stall. Moreover, the unsteady calculations give very good prediction of the pump-turbine performance for both, stable and unstable operating regions. Numerical results show very good qualitative and quantitative agreement with the available experimental data.

1. Introduction

In order to maintain the stability of the electrical grid, pump-turbines are subject of switching between the pumping and the generating mode several times per day (even up to 10 and more). The continuous operating area should be free of hydraulic instabilities. Nevertheless, some instability can limit the operating range. The main features of unstable behavior areas can be divided into generating and pumping modes. Two main reported instabilities in the pumping mode are the cavitation occurrence and the phenomenon of the rotating stall that causes humped-shaped performance curve, which is the origin of the instabilities. Both phenomena have been addressed during PhD studies [1] at laboratory LEGI in Grenoble. Presented paper focuses on unsteady CFD analysis of the rotating stall for various discharges.
1.1. Pumping mode instabilities

Rotating stall is usually described as the occurrence and the decay of stalled cells inside the distributor. It has been initially investigated for axial and centrifugal compressors [2-4]. Studies on the centrifugal pump geometry have been performed much later. In case of the pump-turbines geometry, the amount of studies of the pumping mode instabilities is still relatively low. Pump-turbines are subject of operating away from the BEP (best efficiency point), especially in the part load region. The problem can be partly, but not completely, avoided by some new control techniques, such as variable speed pump-turbine technology [5].

The appearance of the rotating stall phenomena, seen as a hump zone on the performance curve, significantly increases the losses in the distributor and, therefore, causes a positive differential slope on the performance curve as presented on the figure 1. The green and the red lines represent the characteristics of the system. The normal, stable operating range for the pump-turbine is between the green lines. In a stable area, the pump-turbine provides one corresponding flow rate for every given net head of the system. On the contrary, in the area of the unstable characteristics due to the phenomenon of the rotating stall, the pump-turbine can provide three different flow rates for the given system characteristic, as shown by the red curve. In reality, it means that the flow rate through the pump-turbine varies uncontrollably. Behavior like this drastically increases the losses. Moreover, it also causes very strong vibrations that can damage and in the worst case even destroy the machine.

![Figure 1. Pumping mode unstable characteristic due to distributor hump.](image-url)

Obviously, the pump-turbine should not operate in the area of the distributor hump. However, since the pump-turbine needs to change the regime several times per day and must be as adaptable as possible, there is no possibility to completely avoid the problematic unstable area. Extensive study of the hump zone phenomena has been performed by [6]. Numerically, some steady and unsteady 3-dimensional simulations were performed using RANS equations. Detailed numerical study and analysis of the phenomena have been made by [7], who used the Large Eddy Simulation approach. Global rotating stall behavior in one operating point for one opening has been caught quite well, including the area and the frequency of the rotating stall. Nevertheless, those kinds of calculations are very computationally demanding. On contrary, subject of present paper is a cost-efficient approach that enables rotating stall simulations for wide range of operating points.
2. Physical and numerical modelling

CFD simulations have been carried out in LEGI, using FINE/Turbo™ code. The applied code solves 3D steady and unsteady Reynolds-averaged Navier-Stokes (URANS) equations. Turbulence model $k-\varepsilon$ (Extended wall functions), proposed by [8] has been used for modelling turbulence. It is important to stress that described wall functions turn on at the $y+$ value below 12. Therefore, for precise prediction of the flow separation in the distributor regions, mesh in that region should be refined to $y+$ lower than 12.

The solver applies central spatial discretization scheme. Initial solutions for unsteady simulations are calculated from steady results. Local time stepping is used for the steady simulations and dual time stepping for the unsteady simulations (described in [9-10]). Chosen time step for the unsteady simulations is 2° of the impeller revolution, what corresponds to 0,00033 s for a rotation speed of 1000 rpm. The number of internal iterations is limited to 35 for each time step. Around 15 revolutions of the machine have been simulated for one operating point. Computational time for one impeller revolution has been 270 CPU hours. All simulations have been performed on max 12 processors, which is nowadays widely accessible computational power.

Numerical geometry for the presented paper has been provided by Alstom Hydro together with some available experimental data. The investigated machine is a reduced-scaled high head pump-turbine model ($n_q=30$) with the rotational speed 1000 rpm. The domain (figure 2) consists of a draft tube cone, an impeller with 9 blades, 20 guide vanes, 20 stay vanes and a convergent expansion instead of a spiral case. Some more detailed information about the geometry can be found in [1], [11], others are not available due to confidential aspects. All the presented calculations have been performed in a pumping mode regime.

![Figure 2. Numerical pump-turbine domain; left: 3D view, right: meridional view.](image)

Commercial software AutoGRID 5 has been used to create a fully structured mesh that consists of only hexahedral cells. Several feasibility studies and optimizations have been made to find an adequate mesh. The mesh in our case has to be refined in distributor region (guide vanes, stay vanes and diffuser region) to give accurate results in zone of hump-shaped performance curve and rotating stall. Mesh characteristics in table 1 refer to the fully meshed domain mesh for guide vanes opening 14°.
Table 1. Mesh characteristics.

| Component   | Number of nodes | Min. Skewness | Max. expansion ratio | $y_{mean}$ |
|-------------|-----------------|---------------|----------------------|------------|
| Impeller    | 2,88 M          | 35°           | 1.76                 | 7.4        |
| Distributor | 10,4M           | 27°           | 1.52                 | 8.7        |
| Draft tube  | 0.27M           | 41°           | 1.06                 | 6.0        |
| Total       | 13,55 M         | 27°           | 1.76                 | 7.5        |

Boundary conditions are very important for the stability of the simulations. Moreover, in some cases they have also significant influence on the obtained results, especially at the non-optimal flow conditions, such as part load. In our case, constant mass flow rate $Q$ has always been set at the inlet of the domain and at the outlet of the domain, static pressure $p_s$ has been imposed. No-slip condition has been applied on the solid walls.

3. Results

The results in this chapter have been obtained by performing the unsteady simulations on fully meshed domain for a non-cavitating regime. From 2 to 3 revolutions have been required to stabilize the calculations, after that around 10 - 15 revolutions have been performed to analyze the rotating stall phenomenon. Presented results have been obtained for guide vanes opening 14°. Additional openings are presented in [1]. Unsteady flow behaviors such as rotating stall frequency and number of rotating stall cells have been investigated and analyzed. The detailed analysis of the time dependent local flow has been made for the discharge $\Phi/\Phi_{opt}=0.86$ in order to analyze the governing instability mechanisms. Global unsteady results have been compared to the experimental data, as seen on the figure 3. Flow rate coefficient and specific energy coefficient are defined as:

$$\phi = \frac{Q}{\pi \omega R^3} ,$$

(1)

$$\psi = \frac{2gH}{\omega^2 R^2} .$$

(2)

Unsteady results show significant improvement in the prediction of the performance curve, comparing to the steady simulation that have been presented in previous papers [11-12]. Moreover, the comparison between the experimental data and the unsteady simulations shows very good agreement in both stable and unstable operating regions. Flow distribution analyses have been made on the mid-span cutting plane by comparing the meridional velocity patterns. Figure 3a shows symmetrically distributed flow for the discharge $\Phi/\Phi_{opt}=0.95$. Similar behavior has been observed for the operating point $\Phi/\Phi_{opt}=0.89$ and according to the other results, we can assume that the same symmetrical distribution could be observed for all discharges higher than the hump zone. In the hump zone (figure 3b; $\Phi/\Phi_{opt}=0.86$), 4 rotating cells of increased and decreased flow rate have been observed. The rotating frequency of the cells is around 3.5% of the operating frequency of the pump-turbine. The discharge $\Phi/\Phi_{opt}=0.86$ has been chosen for a detailed local flow analysis of the rotating stall that will be presented. Specific energy fluctuations for this discharge reach around $\pm 0.5\%$ and indicate an unstable, potentially dangerous operating point. By additional lowering of the discharge, we can observe 8 rotating stall cells at $\Phi/\Phi_{opt}=0.81$. Rotating stall frequency is lower, comparing to $\Phi/\Phi_{opt}=0.86$ and reaches around 2% of the operational frequency (figure 3c). Moreover, the rotating cells also seem to be less intensive for this case and there are no performance fluctuations. The flow becomes even more unstable at the very low partial load at $\Phi/\Phi_{opt}=0.72$. Three rotating cells of the strongly increased and decreased flows can be observed (figure 3d). The frequency of the rotating stall cells reaches 5.5% of the operational frequency. The fluctuations of the specific energy coefficient are around $\pm 0.5\%$, which is another indicator of a very unstable operating point.
3.1. Time-depending local flow analysis

Local flow analysis between the guide vanes channels has been made for several time steps. Flow separation on the guide vanes (mid-span cutting plane) together with the velocity vectors have been compared to the local flow rate fluctuations through the guide vane channel (figure 4). Moreover, static pressure fields have been analyzed for the same time steps and on the same mid-span cutting plane (figure 5). Figure 4a shows the flow rate fluctuations through one guide vane channel. It can be seen that the flow rate values fluctuate between 45% and 130% of the mean value. The reference plane for flow rate fluctuations has been put at the constant radius, close to the leading edge of the guide vanes. τ represents the time needed for one revolution of the pump-turbine. One cycle of increased and decreased flow rate needs about 7,2 revolutions of the pump-turbine impeller, which corresponds to a rotating stall frequency around 3,5% of the nominal frequency of the machine.
Figure 4. a) Flow rate fluctuations through one guide vanes channel b)-j) Mid-span vorticity with velocity vectors inside the guide vane channel for various time steps
Figure 5. a) Flow rate fluctuations through one guide vanes channel b)-j) Mid-span static pressure distribution inside the guide vane channel for various time steps
Figure 4'b'-'j' shows the vorticity and velocity vectors on the mid-span cutting plane. Flow rate through the channel is the highest at the time step 'c'. At this time, the separation occurs only at the trailing edge of the guide vane. Time steps 'd' - 'g' show the growing region of the flow recirculation inside the guide vanes channel. Velocity vectors show the deviation of the flow to the downstream channel. At the time step 'c', we can observe increased separation and flow deviation on the pressure side of the stay vane, which leads the flow back to the guide vanes regions and consequently causes increased backflow close to the guide vane leading edge. Flow recirculation on the mid-span seems to be the highest in the time steps 'f' and 'g'. However, the flow rate continues to reduce due to unfavourable conditions.

The flow rate is the lowest at the time step 'h', even though the recirculation zone already starts to reduce on the mid-span cutting plane. It means that the flow patterns are very complex and three-dimensional. At the time steps 'i' and 'j', the recirculation zone in the guide vane channel continues to reduce and also the flow rate inside the channel starts to increase again. As expected, it seems that the flow rate through the guide vane channel depends on the size of the flow separation and recirculation zone between the two consecutive guide vanes. However, the dependence is not completely linear. It seems that there is a delay between the size of the recirculation zone and the flow rate, which is caused by the flow behavior closer to the hub and the shroud sides of the machine, as seen on figure 6.

**Figure 6.** a) Flow rate fluctuations through one guide vanes channel b)-i) Radial velocity distribution through guide vane channel for various time steps

Under normal, stable operating points, static pressure distribution along the guide vanes channel should not change in time. However, at the partial load, when the rotating stall is present, the pressure distribution can vary significantly and is related to the flow rate inside the channel. Figure 5 represents the static pressure fields on the mid-span cutting plane inside the guide vanes channels for various
time steps. Time steps and cutting plane are the same as on the figure 4, where the vorticity is presented and can therefore be directly compared. Time step ‘c’ represents the highest flow rate in the guide vane channel. The static pressure value in the channel is very low, which is to be expected, according to Bernoulli’s principle. A small static pressure peak can be seen close to the trailing edge of the guide vane at the time steps ‘b’, ‘c’ and ‘d’. It corresponds to the small recirculation zone that has been observed also on the figure 4. At the time steps from ‘d’ to ‘f’, the flow rate in the channel reduces, the recirculation zone increases and consequently the static pressure increases. Increasing of the static pressure in the channel causes strong adverse pressure gradient around the downstream guide vane leading edge that even increases the flow deviation to the downstream channel as seen at the time step ‘f’. Consequently, the flow rate continues to reduce even more. Slowly, adverse gradient starts to neutralize and with some delay (time step ‘h’ and ‘i’), the flow rate starts to increase again. The stalled cell meanwhile moves to the downstream channel as it can be seen on the right side of figures 4j and figure 5j. Additional flow analysis is available in [1].

3.2. Evolution of the rotating stall

Evolution of the rotating stall can be described in a simplified way as an occurrence and a decay of the stalled cells.

- **Phase 1.** Flow increases in the channel due to favourable conditions, such as low pressure in the middle of the channel (figure 5b) and angle of attack adapted to the guide vanes (figure4b). Slowly, it reaches maximum value of around 130% of the nominal flow through the channel.

- **Phase 2.** The angle of attack at the entrance of the channel (seen from velocity vectors) starts to turn slowly to the downstream channel (figure 4d). Meanwhile, static pressure in the channel starts to increase and causes strong adverse pressure gradient between the middle region of the channel and the region close to the leading edge of the downstream guide vane on the pressure side of the vane (figure 5d, e, f). It leads into strong flow separation in the channel, growth of the recirculation zone and deviation of the flow towards the downstream channel (figure 4d, e, f, g). Flow rate reduction starts firstly in the middle of the channel and after that, it reduces also at the hub and shroud sides of the channel, which leads into additional flow rate reduction. It reaches minimum value of around 45% of the nominal flow rate.

- **Phase 3.** Adverse pressure gradients start to reduce, together with the size of the recirculation zone and cause gradually more favourable conditions that lead more flow rate into the channel. The stalled cell moves to the downstream channel, which causes additional fast flow rate increase in the channel.

4. Conclusions

The results obtained by unsteady simulations, using URANS equations, are very good comparing to the results in the literature. Moreover, they prove that the developed CFD tools can be very exact in the prediction of the performance even under off-design conditions, such as partial load. Additionally, CFD simulations give us a detailed insight into the flow patterns inside the machine that are experimentally very demanding or even impossible to get. Rotating stall can take various forms and intensity. The number as well as the frequency of the rotating stall cells can vary. Even though it should be confirmed experimentally, it seems that the lower number of rotating cells indicates higher rotating stall frequency and higher intensity that is seen as larger cells of increased or decreased flow. Therefore, corresponding operating points seems to be more dangerous for the machine. Local flow analysis showed that the rotating stall can be described as a growth and decay of the stalled cells and rotation of these cells around the distributor. Most important parameters for the stall evolution are the size of the flow separation and recirculation zones that completely change the flow conditions at the leading edge of the guide vanes. Pressure fields, especially the direction and magnitude of adverse
pressure gradient, cause the deviation of the flow to the downstream channels and are responsible for the so-called rotation of the stalled cells. The occurrence of the rotating stall causes losses in the distributor regions [11] and consequently causes the positive slope on the performance curve, so called hump zone. Taking into account all mentioned rotating stall mechanisms, we can highlight several possible ways to modify the effects of the rotating stall:

- Changing guide vanes opening angles
- Changing guide vanes geometry
- Changing radial gap between the impeller and the distributor
- Changing impeller geometry

The main idea of the presented study was the investigation of the hump zone and related rotating stall in order to minimize its effects on the pump-turbine performance. At this moment, all the numerical tools to achieve this are available and can be used for any geometry that is facing the problem of the rotating stall. Additional study from industrial or academic point of view should be performed in order to propose practical solutions for reducing or eliminating the negative effects of the rotating stall on the pump-turbine performance and stability of the operating points.

5. Acknowledgments

The authors wish to express their gratitude to Alstom Hydro for supplying geometry and experimental results. The laboratory LEGI is part of the LabEx Tec 21 (Investissements d’Avenir – grand agreement n°ANR-11-LABX-0030). PhD study was financed by Slovene Human Resources Development and Scholarship Fund.

6. References

[1] Ješe U 2015 Numerical study of pump-turbine instabilities: Pumping mode off-design conditions PhD Thesis Univ. Grenoble Alpes, Grenoble
[2] Frigne P and Van den Braembussche R 1984 Distinction between different types of impeller and diffuser rotating stall in a centrifugal compressor with vaneless diffuser Trans of ASME Journal of Engineering for Gas Turbines and Power 106(2):468-474
[3] Haupt U, Seidel U, Abdel-Hamid A and Rautenberg M 1988 Unsteady flow in a centrifugal compressor with different types of vanes diffuser Trans of ASME Journal of Turbomachinery, 110(3):293-302
[4] McDoughal N M, Compsty N A and Hynes T P 1990 Stall inception in axial compressors Trans of ASME Journal of Turbomachinery 112(1):116-123
[5] Ciocan G D, Teller O and Czerwinski F 2012 Variable speed pump-turbines technology University “Politehnica” of Bucharest Scientific Bulletin, Series D: Mechanical engineering 74(1):33-42
[6] Braun O 2009 Part load flow in radial centrifugal pumps PhD Thesis EPFL, Lausanne
[7] Pacot O, Kato C, Guo Y, Yamade Y and Avellan F 2016 Large eddy simulation of the rotating stall in a pump-turbine operated in pumping mode at a part-load condition Journal of Fluids Engineering, ASME, doi: 10.1115/1.4033423
[8] Hakimi N, Hirsch C, and Pierret S 2000 Presentation and application of a new extended k–ε model with wall functions ECCOMAS Barcelona.
[9] NUMECA International 2011 Theoretical manual FINE™/Turbo v8.9.
[10] Pouffary B, Fortes-Patella R, Reboud J L and Lambert PA 2008 Numerical simulation of 3D cavitating flows: Analysis of cavitation head drop in turbomachinery Journal of Fluid Engineering vol. 130, 061301, pp. 1-10.
[11] Ješe U, Fortes-Patella R and Antheaume S 2014 High head pump-turbine: Pumping mode numerical simulations with a cavitation model for off-design conditions IOP Conference Series: Earth and Environmental Science 22, 032048
[12] Ješe U, Fortes-Patella R and Dular M 2015 Numerical study of pump-turbine instabilities under pumping mode off-design conditions ASME/JSME/KSME 2015 Joint Fluids Engineering Conference (pp. V001T33A018-V001T33A018)