Investigation of the Flow Field and Performances of a Centrifugal Pump at Part Load

R Prunières, Y Inoue and T Nagahara
Industrial Products Company, Hitachi Ltd., Japan

Email: romain.prunieres.gx@hitachi.com

Abstract. Centrifugal pump performance curve instability, characterized by a local dent at part load, can be the consequence of flow instabilities in rotating or stationary parts. Such flow instabilities often result in abnormal operating conditions which can damage both the pump and the system. In order for the pump to have reliable operation over a wide flow rate range, it is necessary to achieve a design free of instability. The present paper focuses on performance curve instability of a centrifugal pump of mid specific speed ($\omega_s = 0.65$) for which instability was observed at part load during tests. The geometry used for this research consist of the first stage of a multi-stage centrifugal pump and is composed of a suction bend, a closed-type impeller, a vaned diffuser and return guide vanes. In order to analyse the instability phenomenon, PIV and CFD analysis were performed. Both methods qualitatively agree relatively well. It appears that the main difference before and after head drop is an increase of reverse flow rate at the diffuser passage inlet on the hub side. This reverse flow decreases the flow passing area at the diffuser passage inlet, disallowing effective flow deceleration and impairing static pressure recovery.

1. Introduction

Centrifugal pump of mid specific speed equipped with vaned diffuser sometimes show performance curve instability. Such performance instability, characterized by a local dent (head drop with decreasing flow rate), can result in abnormal operating conditions which can damage both pump and system. In order for a pump to have reliable operation over a wide applicable range, it is then necessary to understand the onset and the mechanism of the phenomenon resulting in performance curve instability.

For centrifugal pump with vaned diffuser, performance curve instability typically appears around 60 % of best efficiency flow rate ($Q_{BEP}$) and is often attributed to instabilities occurring in the diffuser such as stall. Hergt and Starke [1] connected the head drop to the development of span-wise non-uniformity caused by diffuser inlet reverse flow. An important change of the axial thrust was also noticed. By means of experiments and computations, Sano et al. [2][3] analysed the influence of the clearance between impeller and diffuser on the development of the different types of diffuser stalls: forward and backward rotating stall, alternate blade stall and asymmetric stall. One of their conclusions being that the clearance plays an important role in the diffuser stall.

As far as the authors are aware, the first use of Particle Image Velocimetry (PIV) to analyse stall and performance curve instability was conducted by Sinha et al. [4]. They highlighted the influence of the flow in the clearance between impeller and diffuser on the onset of the stall mechanism.
More recently, using Large-eddy Simulation, Pacot [5] analysed in detail the propagation mechanism of rotating stall in a water pump-turbine, driven by the growth and the shrinking of the stall cells. Using Reynolds-Averaged Navier-Stokes (RANS) approach, Imai et al. [6] and Prunières et al. [7] analysed the effect of impeller outlet blade geometry on the head drop at very low flow rate (30% of \( Q_{BEP} \)). Finally, Cooper et al. [8] analysed for a double suction pump, the effect of diffuser design and impeller trimming geometry on curve stability, as part of a pump performance optimisation.

Present analysis focuses on the performance curve instability of a multi-stage centrifugal pump of universal specific speed of 0.65 and aims at investigating the head drop process both qualitatively and quantitatively. Experimental tests, using a PIV device, and Computational Fluid Dynamics (CFD), using an unsteady RANS approach, were performed to analyse the pump flow field and performances.

2. Experimental Analysis

2.1. Test Apparatus

The pump analysed in this paper is the first stage of a multi-stage centrifugal pump, and is composed of a suction bend, a closed-type impeller, a vaned diffuser and return guide vanes to the next stage (not included for tests). The pump universal specific speed is \( \omega_s = 0.65 \).

The experimental apparatus is presented on Figure 1. The PIV system is composed of a camera, a laser and small particles of Nylon added to the pump working fluid (industrial water). The Nylon particles are small enough (50 μm) not to disturb the flow, whilst their specific gravity (1.03) is slightly higher than the working fluid.

The PIV measurement technic consists in highlighting a thin slice of the flow with the laser at two instant \( t = t_0 \) and \( t = t_0 + \Delta t \) (with \( \Delta t \to 0 \)). At each instant, a picture of the flow is recorded by the camera. Using a correlation algorithm between the two pictures, it is then possible to define the velocity vectors of each particle and subsequent the velocity contour.

In order to allow PIV analysis of the flow field in the diffuser, a straight pipe was set between the suction bend and the impeller inlet, while a transparent casing was mounted at the diffuser tip side. The transparent casing allowing highlight of two consecutive diffuser passages, and three different span positions were investigated: near hub (15% of diffuser width), mid-span (50% of diffuser width) and near vanes tip (85% of diffuser width, impeller’s front shroud side).

2.2. Pump Performances

The investigated pump performances in term of pressure coefficient versus flow rate are presented on Figure 2. A clear head drop can be observed from 61 to 59% of pump best efficiency flow rate (\( Q_{BEP} \)). The head drop corresponds to approximately 2.9% head decrease.
2.3. PIV Analysis

PIV analysis was performed at 64 and 59 % of $Q_{BEP}$ in order to investigate the flow field in the diffuser before and after head drop. Two types of analysis were performed, firstly an instantaneous analysis and secondly a phase-averaged analysis at different impeller – diffuser relative positions.

A sample instantaneous image of the flow field is presented for both flow rates and the three span positions on Figure 3. On this figure, the velocity has been normalised by the impeller outlet peripheral velocity $U_2$.

![Sample Instantaneous PIV Measurements with Zoom on the Guide Vanes Leading Edge, a: 64 %, b: 59 %](image)

From Figure 3, it can be observed that the diffuser passage suffer from low velocity area on both suction and pressure side, for both flow rates and the three span positions. However, the instantaneous flow is not two dimensional in the span direction. At 64 % of $Q_{BEP}$ (before head drop), in the diffuser passage, the tip side shows lower velocity than mid-span and hub side. The analysis of the phase-averaged data hereafter will allow concluding that this phenomenon is unsteady. At 59 % of $Q_{BEP}$ (head drop), the hub side shows lower velocity than mid-span and tip side.

At 59 % on hub side, reverse flow from diffuser inlet throat is visible and the flow angle does not match the guide vanes angle. At other span positions, the flow is fairly well oriented and matches the guide vanes angle. This can be explained as follows, considering that diffuser reverse flow impacts the impeller outlet velocity profile: because the flow passing area is reduced by the existence of reverse flow from diffuser inlet throat, the flow at impeller outlet is accelerated at other span positions leading to an increase of the flow angle. In other words, at span positions with no low reverse flow from diffuser inlet throat, the flow configuration is locally similar to higher flow rate conditions, and then almost matches the guide vanes angle which was set for best efficiency flow rate.
During this instant analysis over several impeller revolutions, no propagation process of the low velocity area has been clearly identified. This makes a straightforward interpretation impossible for the existence of rotating stall in this present case, before or after head drop.

The phase-averaged data, with velocity normalised by $U_2$, are presented on Figure 4. For both flow rates and the three span positions, one can noticed that the impeller blade position has limited impact on the flow field. Moreover, it can be observed that at tip side and mid-span the flow patterns are very similar for both flow rates. At 64 % and tip side position, the flow pattern does not show any low velocity near the diffuser passage inlet throat as observed with the instantaneous data. It can then be concluded that this phenomenon is unsteady.

**Figure 4: Phase Averaged PIV Measurements, a: 64 %, b: 59 %**
At hub side however, the flow is completely different from 64 to 59 % of $Q_{BEP}$. At 59 %, strong reverse flow appears at the diffuser passage inlet throat and the fluid flows back to the clearance between the impeller and the diffuser guide vanes (Figure 4b). This was already observed with instant data on Figure 3b, and it can then be concluded that this phenomenon is steady.

Such reverse flow at the diffuser passage inlet throat when head drop has been pointed out by Hergt and Starke [1]. In that case, the reverse flow was occurring at the tip side, and was caused by the span-wise non-uniformity of the flow at the impeller outlet. In this present case it occurs at hub side and one can consider that the flow at the impeller outlet is not uniform in the span-wise direction.

3. CFD Analysis

3.1. CFD Settings

In order to investigate in detail the head drop mechanism, unsteady CFD analysis was performed. The flow was solved using incompressible Reynolds-averaged Navier-Stokes equations (referred to as “RANS”) and the Reynolds-stress tensor closure is ensured by $k-\omega$ SST turbulence model. The geometry used is exactly the same as the one for the experiments, and for example, the straight pipe between suction elbow and impeller inlet (recall Figure 1) was also included. Moreover, leakage flows were considered at wear ring and stage bush.

The mesh is full hexahedral and the total size is approximately 19M elements. Pressure was imposed at the domain inlet and flow rate at outlet. The time step was set so one impeller revolution consists of 120 time steps. The results presented herein were obtained after 14 impeller revolutions.

3.2. CFD Performances

CFD performances for pressure coefficient as function of the flow rate are presented on Figure 5. It can be observed that the head drop position is not exactly well predicted: in the case of CFD, the head drop occurs from 64 to 61 %, while in the experiments it was from 61 to 59 %. As the difference of flow rate is small, it was concluded here that the present CFD analysis was able to reproduce the experimental results and can be used in order to investigate the head drop phenomenon.

![Figure 5: CFD Performances Compared to Experimental Performances](image-url)

The magnitude of the head drop agrees well with the experiments: 3.2 % in the case of CFD and 2.9 % in the case of experiments.
3.3. CFD Impeller and Diffuser Analysis

The meridional velocity vectors, mass flow averaged in the pitch direction and time averaged over one impeller revolution are presented on Figure 6.

![Figure 6: Meridional Velocity Vectors](image)

At 79 % of $Q_{BEP}$, one can observe that the impeller inlet recirculation starts to develop. When decreasing the flow rate to 64 % and lower, the inlet area showing recirculation increases. However no major difference can be observed between 64, 61 and 59 %.

At impeller outlet, for 64, 61 and 59 %, the meridional velocity vectors near the hub are clearly smaller than at shroud. This is not observed at 79 %. This can be attributed to an outlet reverse flow on the hub side. This effect seems smaller at 61 %, however, due to the pitch-averaged process used here, it is difficult to quantify if this recirculation expands or vanishes from 64 to 59 %.

In order to quantify this reverse flow, the reverse flow rate at impeller outlet and diffuser inlet throat is presented on Figure 7 as percentage of the flow rate going through the impeller $Q_i$ (i.e. including leakage flow rate). On one hand, the reverse flow rate at impeller outlet is based on the impeller outlet area where the fluid flows back into the impeller (i.e. radially inward). On the other hand, the reverse flow rate at the diffuser inlet throat is based on the inlet throat area where the fluid flows back into the clearance between the impeller and the diffuser guide vanes.

![Figure 7: Impeller Outlet and Diffuser Throat Inlet Reverse Flow Rate](image)
On this figure, it can be observed that the impeller reverse flow rate decreases when the CFD capture the head drop (from 64 to 61 %) before increasing again. This confirms the analysis made on Figure 6. Conversely, the diffuser throat inlet reverse flow rate strongly increases from 64 to 61 %, before decreasing from 61 to 59 % when the pump head is increasing again in CFD. It can then be conclude that when there is head drop, the additional reverse flow rate from the diffuser throat inlet does not flow back to the impeller but is mainly conveyed in the clearance between the impeller and the diffuser guide vanes. It is also important to notice that the diffuser reverse flow is occurring on the hub side, as observed by PIV on Figure 4.

Figure 8 presents the velocity vectors and contours in two diffuser passages for comparison with experimental results obtained by phase-averaged PIV (recall Figure 4). These data were averaged over one impeller revolution and the span positions are identical to the ones for the experiments. Like for PIV, flow velocity has been normalised by $U_2$.

![Figure 8: Velocity Contours and Vectors in Diffuser, Averaged Over One Impeller Revolution](image)

At tip side and mid-span, CFD results qualitatively agree well with PIV results (recall Figure 4). The flow angle fairly well matches the guide vanes angle at tip side and mid-span, but not at the hub side.

At hub side, the reverse flow at diffuser inlet appears clearly for the three flow rate. However, for CFD, the difference is less obvious before and after head drop (64 to 61 % in CFD analysis) as it was observed for PIV phase-averaged data (from 64 to 59 %, recall Figure 4).

The consequences of the modification of diffuser reverse flow rate observed on Figure 7, can be seen on Figure 8. At the tip side: the velocity downstream of the inlet throat is clearly higher at 61 % than other flow rates. This is a direct consequence of the reverse flow occurring at the hub side which decreases the flow passing area at diffuser inlet throat. Because the flow cannot enter the diffuser passage at the hub side, the velocity increases on the tip side.

In order to quantitatively analyse the effect of the diffuser reverse flow rate on pump performances, Figure 9 presents the evolution of the static pressure at different streamwise positions, averaged over one impeller revolution. At 61 %, as the velocity cannot be decelerated as it should be due to the important reverse flow rate at the hub side of the diffuser, a lack of pressure recovery is observed from impeller outlet to diffuser throat inlet. Whilst downstream the diffuser inlet throat (to the pump outlet),
the increase of static pressure is almost equivalent to between 64 and 61%. The decrease of static pressure recovery between the impeller outlet and the diffuser inlet throat results in loss increases which lead to the head drop.

When the pump head increases again, from 61 to 59%, the pressure recovery from impeller outlet to diffuser inlet throat increases again. This is explained by the decrease of the diffuser reverse flow rate (recall Figure 7) and consequently the flow deceleration effectiveness increases as can be also observed by comparing Figure 8b and c at tip side.

4. Conclusions
The purpose of this paper was to investigate the flow field and performances of a centrifugal pump at part load when performance curve instability occurs. For that purpose, both experimental and computational analyses were performed.

Main observations can be summarised as follow:
- CFD was able to capture relatively well the head drop. Despite a small difference of flow rate where the head drop occurs, the amount of head drop was well predicted;
- Whilst at head drop, a small change of flow pattern at impeller outlet was observed by CFD, as the impeller outlet reverse flow rate at hub side decreases;
- By PIV, it was observed that the main difference of flow pattern in the diffuser before and after head drop consists in the development of strong reverse flow at diffuser inlet throat on the hub side. This phenomenon was quantitatively well captured by CFD, showing a strong increase of diffuser reverse flow rate when there is head drop. However in CFD, the velocity contours at hub side do not show a clear difference before and after head drop. For other span locations, CFD and PIV agree well;
- Finally, CFD allowed understanding of the consequence of the diffuser reverse flow on the pump performances. As the reverse flow rate strongly increases when there is head drop, pressure recovery in the diffuser is impaired due to a lack of flow deceleration. When the head increases again, the reverse flow rate decreases and the flow deceleration effectiveness increases, resulting in more pressure recovery. By CFD, the head drop can then be attributed to modification of the flow from the impeller outlet to the diffuser inlet throat.
Nomenclature

\( A \)  impeller outlet meridional area
\( C \)  velocity
\( D_2 \)  impeller outlet diameter
\( g \)  acceleration due to gravity
\( H \)  total head
\( H_s \)  static head
\( P \)  pump absorbed power
\( Q \)  pump flow rate
\( Q_{\text{BEP}} \)  pump best efficiency flow rate
\( Q_i \)  impeller flow rate
\( U_2 \)  impeller outlet peripheral velocity, \( U_2 = \omega \frac{D_2}{2} \)
\( \rho \)  density of working fluid
\( \psi \)  pressure coefficient, \( \psi = \frac{2gH}{U_2^2} \)
\( \psi_s \)  static pressure coefficient, \( \psi_s = \frac{2gH_s}{U_2^2} \)
\( \omega \)  pump angular velocity
\( \omega_s \)  universal specific speed, \( \omega_s = \frac{\omega Q^{0.5}}{(gH)^{0.75}} \)

References

[1]  Hergt P and Starke J 1985 Flow Patterns Causing Instabilities in the Performance Curves of Centrifugal Pumps With Vaned Diffusers Proc. of the 2nd Int. Pump User Symp. (Houston)
[2]  Sano T, Nakamura Y, Yoshida Y and Tsujimoto Y 2002 Alternate Blade Stall and Rotating Stall in a Vaned Diffuser JSME Int. J. Serie B 45 810–19
[3]  Sano T, Yoshida Y, Tsujimoto Y, Nakamura Y and Matsushima T 2002 Numerical Study of Rotating Stall in a Pump Vaned Diffuser J. of Fluids Eng. 124 363–70
[4]  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. of Fluids Eng. 123 490–99
[5]  Pacot O 2014 Large Scale Computation of the Rotating Stall in a Pump-turbine Using an Overset Finite Element Large Eddy Simulation Numerical Code Ph. D Thesis (École Polytechnique Fédérale de Lausanne)
[6]  Imai N, Inoue Y, Okihara T and Sato T 2014 Study of Performance Curve Instability in a Centrifugal Pump by Using CFD Analysis (in Japanese) 71st Turbomachinery Association Lecture Meeting (Turbomachinery Society of Japan)
[7]  Prunière R, Imai N, Inoue Y, Okihara T and Nagahara T 2015 Study of The Flow Field and Performances of a Centrifugal Pump During Unstable Operating Conditions by CFD Proc. of the ASME/JSME/KSME Joint Fluids Eng. Conf. (Seoul) pp 2373–79
[8]  Cooper P, Schiavello B, Hosangadi A and McGuire T 2015 CFD-aided Hydraulic Development of a Steam Generator Feed Pump Proc. of the ASME/JSME/KSME Joint Fluids Eng. Conf. (Seoul) pp 2315–28

Acknowledgments

Authors would like to express their sincere gratitude to Dr. Sato and Ms. Suzuki of Hitachi, Ltd., who strongly contributed to the CFD settings and analysis.