Investigation of the 4-Quadrant behaviour of a mixed flow diffuser pump with CFD-methods and test rig evaluation

S. Höller¹, H. Benigni¹ and H. Jaberg¹

¹ Graz University of Technology, Kopernikusgasse 24/IV, 8010 Graz, Austria

e-mail: hoeller-litzlhammer@tugraz.at

Abstract. The complete pump characteristics including its 4-quadrant behaviour are of essential interest for off-design operations such as a pump trip. At this exceptional load case the pump enters the dissipation mode and moves further into the turbine mode while the direction of rotation and the flow direction will change. The time-consuming and expensive experimental investigation of the 4-quadrant behaviour requires a specific test rig, allowing the flow direction as well as the rotational direction of the investigated pump to be reverted. By measuring the pump performance (head and efficiency) at variable positive and negative discharge and rotation the complete pump characteristics are evaluated. Nowadays CFD-analysis allows for the reliable prediction of the hydraulic performance of a pump near the design point. However, abnormal operating conditions lead to complex and unsteady flow phenomena inside the pump. Besides steady-state calculations in the normal operating conditions quite comprehensive transient CFD-investigations are required to simulate the whole pump characteristics accurately. The present study focuses on the comparison of the results obtained on the test rig and by numerical methods and shows a remarkably good agreement between them. It can be shown that it is possible to reliably simulate the 4-quadrant behaviour of a mixed flow diffuser pump based on CFD-methods. Furthermore an exemplary waterhammer calculation shows the successful application of the numerically calculated 4-quadrant behaviour.

1. Introduction

The hydraulic behaviour of centrifugal pumps under normal operating conditions is well known. In this normal operating mode the pumped fluid passes the pump from suction side to the pressure side while the pump rotates in pump direction. Typically this behaviour is of interest when operating a pump in any system, whether in a closed or an open loop. Almost every pump manufacturer offers head and efficiency curves for their pumps to calculate a stationary duty point as the intersection of the pump head curve and the system resistance curve. Another important information is the power consumption related to this operation point. The prediction of the hydraulic behaviour by means of computational fluid dynamics (CFD) has been shown in recent years to be very reliable. Especially for operating points near the design or best efficiency point the correlation between simulation and experiment is very good [1], [2]. Performance prediction of pumps in off-design and especially in abnormal operation modes is more complicated since unsteady phenomena then dominate the flow field.

The abnormal hydraulic performance of a pump gets into focus in case of off-design operating conditions where the direction of flow or rotation may change. An example application is the reverse
operation of a pump where the unit is used as turbine. In this specific application it is necessary to have information on the relationship of turbine head and turbine flowrate for given speed and the efficiency or power output at this special operation. A fundamental investigation of reverse operating pumps has already been presented by Laux [3] in 1982. A review of research and investigations related to pumps as turbines has been carried out by Nautiyal et al. [4]. Another important abnormal operation mode, where the knowledge of the full pump characteristics is essential is the pump trip. Only if full information about the pump behaviour in every possible operation quadrant is known, the transient behaviour of the machine can be predicted. This information is crucial to calculate reliable system transients which provide necessary data for the pipeline design like the maximum or minimum resulting internal pressure for example.

Since extensive testing is required to get all necessary information concerning the full 4-quadrant behaviour of centrifugal pumps there is limited information available in the open literature. The publications of Wylie and Streeter [5] or Stepanoff [6] are well known in the community and their information is often referenced for transient system analysis if no other information is available. This approach leads to inevitable inaccuracies in the prediction of the hydraulic pressure transients since the real pump characteristics are not known.

A reliable numerical calculation of the 4-quadrant pump characteristics is a very challenging task as the abnormal operating conditions lead to complex and unsteady flow phenomena inside the pump. Gros [7] and Couzinet [8] investigated the 4-quadrant pump performance by means of numerical simulation and experimental investigations. The results of their unsteady numerical simulations showed good accuracy when comparing them to their experimental results.

The present paper outlines a new approach to obtain a reliable prediction of the complete pump characteristics by means of CFD-calculation.

1.1. The 4-Quadrant behaviour of a pump and its representation

There are several ways to describe the hydraulic behaviour of a pump in normal as well as abnormal conditions. The representation of the 4-quadrant behaviour of a pump with the head coefficient $\psi$ and the flow coefficient $\phi$ (eq. (1) and eq. (2)) is not preferable since the direction of rotation is not captured. Another disadvantage of this notation appears with blocked impeller, where a singularity in $\phi$ and $\psi$ occurs.

The IEC 60193 [9] suggests the illustration of the overall pump behaviour with the discharge factor $Q_{ED}$ (eq. (3)), the speed factor $n_{ED}$ (eq. (4)) and the torque factor $T_{ED}$ (eq. (5)). A principle representation of the 4-quadrant behaviour of a semi-axial pump is shown in Figure 1 where different operation modes of a pump are marked. The sign convention in the IEC 60193 standard for rotation and flowrate is specified with positive signs for turbine operation. Point A marks the pump operation

![Figure 1: Extended operating range of a semi axial pump according to the IEC 60193 standard](image)
mode with negative flow and rotation. Point B identifies the zero discharge condition or shut off point in pump direction of rotation. The pump brake mode C is characterized by a negative rotation (pump direction of rotation) and a positive flowrate (from pressure side to suction side). The operation of the pump with blocked impeller and turbine discharge is marked with point D. The pump operating as turbine with positive flow and rotation direction is identified with mode E where the unit delivers torque to the machine shaft. The zero torque point F or runaway point in turbine operation is of essential interest in case of a pump trip to calculate reliably the runaway speed and runaway discharge in case of an emergency due to an electrical failure. Mode G represents the turbine brake condition were torque must be provided to the unit to rotate in turbine direction with turbine discharge. The curve below the zero head line (H=0) is of minor interest since this operation may only occur under laboratory conditions. Point H on the negative head curve (-H) identifies the pump runaway point.

\[ \psi = \frac{2gH}{u^2} \]  
\[ \varphi = \frac{cm}{u} = \frac{Q}{A u} \]  
\[ Q_{ED} = \frac{Q}{D^2 \sqrt{g H}} \]  
\[ n_{ED} = \frac{nD}{\sqrt{g H}} \]  
\[ T_{ED} = \frac{T}{D^3 \rho g H} \]

1.2. The investigated pump with variable pitch

The pump presented in this paper was designed for a specific speed of \( n_q = 95 \) (rpm, m³/s, m) with variable pitch. The mechanical concept of a variable pitch pump is quite complicated as far as the adjustment of the blade is concerned.

The benefit is clearly visible in Figure 2 where the envelope of the efficiency as well as the best efficiency points of each head curve for different blade positions are shown. Such pumps are very often used for applications with horizontal system curves. For a variable pitch pump the blade can be opened for higher flow rates and thus the head does not drop. Towards part load this appears vice versa. The shut off head is lower for a more closed position, and so also the power consumption is lower.

![Figure 2 Head curves and relative efficiencies for a variable pitch pump](image-url)
2. Numerical Setup

2.1. Numerical model

The numerical simulation was carried out with the commercial CFD-code ANSYS CFX14.5. Many different models were generated and analysed in course of the numerical analysis. The final simulations, which led to the results stated below, were carried out on a full 360° model. The numerical setup (see Figure 3) consists of the suction bell domain (green – stationary – including an inlet extension), the impeller domain (red – rotating) and the stator domain (blue – stationary – including an outlet extension). The impeller domain consists of 6 impeller passages and the diffuser of 7 stator passages. Structured hexagonal meshes were generated for each domain. 1 on 1 mesh matching has been applied to the periodic faces of the impeller and diffusor domains. Therefore no additional grid interface is necessary inside one domain. The impeller mesh also includes the tip gap of the semi open impeller for this variable pitch geometry. The mesh generation for diffuser and impeller was done with Turbogrid®, whereas the mesh for the inlet and suction bell region was generated with ICEM®. Near wall inflation on all meshes was applied to resolve the boundary layer of the viscous flow. The averaged y+ value for all meshes at a flowrate of the best efficiency point (BEP) of the pump is equal to 20. The mesh statistics are summarized in Table 1.

![Figure 3 left: Computational domains, right: numerical mesh of impeller and diffusor](image)

| Domain   | Elements (in 1000) | Nodes (in 1000) | Max. aspect ratio | Min. orthogonal element angle [deg] |
|----------|--------------------|-----------------|------------------|------------------------------------|
| Inflow   | 183                | 191             | 1600             | 20                                 |
| Impeller | 4438               | 4652            | 1300             | 25                                 |
| Diffuser | 2547               | 2701            | 3500             | 27                                 |
| Global   | 7168               | 7544            | 3500             | 20                                 |

For the stationary calculations the frozen rotor interfaces type was chosen between stationary and rotating domains. In the transient CFD-simulations the rotor position is updated at every timestep during the simulation according to the rotors rotational speed. For the inlet boundary condition (defined at a distance of \( L = 5 \cdot D_{\text{inlet}} \) away from the pump inlet) the mass flow rate was specified whereas at the outlet (defined at a distance of \( L = 5 \cdot D_{\text{outlet}} \) away from the stator outlet) an average static pressure was applied as boundary condition. As turbulence model the SST model developed by Menter et al. [10] was applied to the stationary calculations. This two-equation approach based on an eddy-viscosity concept is commonly used for hydraulic turbomachinery. The transient analyses were
carried out with the SAS-SST turbulence model [11]. The concept of the SAS-turbulence model is based on the introduction of the von Karman length scale into the turbulence scale equation. So, the model dynamically adjusts to resolved vortex structures in the URANS (Unsteady Reynolds Averaged Navier Stokes) method, which results in a LES-like behaviour in unsteady regions of the flow field.

### Table 2 General CFD settings

| Option         | Stationary CFD                  | Transient CFD                  |
|----------------|---------------------------------|---------------------------------|
| **Inlet**      | Mass Flow Rate                  | Mass Flow Rate                  |
| **Outlet**     | Average Static Pressure         | Average Static Pressure         |
| **Turbulence Model** | SST                            | SAS-SST                         |
| **Timestep**   | Iteration 1 to 25: 1/(ω * z_r * z_d) | revolution 1: 12/360 * 2 π / ω |
|                | Iteration 26 to 100: 1/(ω * z_r) | revolution 2 to 6: 1/360 * 2 π / ω |
|                | Iteration 101 to 500: 1/(ω)     | -                              |
| **Coefficient loops** | -                              | 10                             |

An adaptive timestep was used both in stationary as well as in transient simulations to speed up convergence. In the stationary calculations 500 iterations were calculated and to determine the timestep combinations of the runner speed ω and the blade numbers of the runner (z_r) and the diffuser (z_d) were used. For the transient simulations a timestep corresponding to a resolution of 12° for the first impeller revolution and 1°for the following revolutions was chosen. The general CFD settings are summarized in Table 2.

### 2.2. Post processing

The evaluation of the hydraulic performance is evaluated by means of the key figures as mentioned in the following. In general, the net head is the difference between total pressure head at the outlet and total pressure head at the inlet. According to ISO 9906 standard [12], the net head represents the difference between the static pressure plus the mean kinetic energy head at outlet and inlet (geodetic head difference neglected, see eq. (6)). The pressure on the test rig is measured on 4 pressure measuring taps which are being positioned around the pipe circumference with an angle of 90° between them. The locations were set 2D away from the flanges (Figure 4). The post-processing of the CFD results was then carried out in a similar way as given in eq. (7).

\[
H_{\text{meas}} = \frac{1}{\rho g} \left[ \left( \frac{1}{4} \sum_{i=1}^{4} p_i + \frac{\rho}{2} \left( \frac{Q}{A} \right)^2 \right) \right]_{\text{Outlet}} - \left( \frac{1}{4} \sum_{i=1}^{4} p_i + \frac{\rho}{2} \left( \frac{Q}{A} \right)^2 \right)_{\text{Inlet}} \quad (6)
\]

\[
H_{\text{CFD}} = \frac{1}{\rho g} \left[ \frac{1}{A_{\text{Outlet}}} \left( \int p_{\text{stat}} \cdot dA \right)_{\text{Outlet}} - \frac{1}{A_{\text{Inlet}}} \left( \int p_{\text{stat}} \cdot dA \right)_{\text{Inlet}} \right] + \frac{(q_{\text{Outlet}})^2 - (q_{\text{Inlet}})^2}{2g} \quad (7)
\]

The head was analysed with the head coefficient ψ, the flowrate is expressed by the flow coefficient φ. The efficiency η is described by eq. (8) for pump operation.

\[
\eta = \frac{H \cdot Q \cdot \rho \cdot g}{T \cdot \phi} \quad (8)
\]

### 3. Experimental setup

The measurements were carried out on the 4-quadrant test rig of the Institute of Hydraulic Fluid Machinery at Graz University of Technology (see Figure 4). As the full size prototype pump with a mechanical power of more than 1500 kW would exceed the performance of the test facility a model scale of 1:3.6 (model : prototype) was chosen based on the requirements for acceptance tests according to the IEC60193 standard [9]. The speed of the model pump was chosen to ensure a Reynolds number higher than 4 x 10^6 for the measurements in pump operation mode. The model pump (see Figure 4)
was manufactured and installed on the closed loop 4-quadrant test rig. The speed-regulated model pump, consisting of a runner with 6 blades and a diffuser with 7 guide vanes, was equipped with a 90 kW speed controlled motor/generator. The impeller consists of a turned and milled hub and 6 separate, turned and milled impeller blades. After the assembly of the impeller blades the tip contour was turned spherically and the impeller was balanced. The experimental setup and the measurement instruments were based on the IEC60193 standard [9] providing a total measurement uncertainty of ±0.2%.

4. Results

4.1. Pump operation
In Figure 5 the head curves (head coefficient $\psi$) in pump operation for different blade positions are shown versus the flow rate (flow coefficient $\phi$) for both CFD and test rig results. While the CFD simulations were carried out with the prototype size and speed of the pump, the measurements were performed with above mentioned model size. To allow for a direct comparison of measurement and simulation an efficiency scale up according to IEC60193 standard [9] was used. As it can be seen a good correlation over the whole operation range could be detected.
Zone A in Figure 5 indicates the main instability zone of the pump for different blade positions. Usually the pump normally can only be operated on the right side of this zone which is generally indicated as operation limit in pump data sheets. The second zone B indicates the diffuser instability and is of minor importance. This zone is not identified as an unstable head curve since the sign of the slope of the curve does not change and so a continuously increasing head with decreasing flow rate is observed. This zone could be eliminated by diffuser modifications (eg. reduction of diffuser blade numbers and altered diffuser inlet and outlet angles) but would shift the best efficiency point to lower flow rates. The dotted lines indicate the results of CFD calculations in stationary mode. Especially around the optimum there is a good prediction of the pump head. Zone A is also well predicted whereas zone B could not be found in the stationary CFD simulations.

**Figure 6** Head and efficiency history of unsteady computations; left at 100% \(Q_{\text{BEP}}\), right at 60% \(Q_{\text{BEP}}\)

In addition to the stationary calculations transient calculations for several operating points in pump operation were carried out. These calculations are marked with diamonds in Figure 5 for the -0.5° blade position. Especially the head drop in zone A is captured well by these transient calculations. Figure 6 shows the head and efficiency convergence history of the unsteady simulations for two different operating points at best efficiency flowrate \(Q_{\text{BEP}}\) (left side) and at part load at 0.60*\(Q_{\text{BEP}}\) (right side). It can be seen very clearly that both, head and efficiency, increase at the beginning of the transient calculations. This is true for both discussed operating points. The increase of head and efficiency continues for some revolutions until more or less stable oscillations around an average value occur. At least 6 impeller revolutions were simulated in course of the unsteady calculations. The transient results of the last three revolutions were finally used to calculate a representative average value of head and efficiency marked with diamonds in Figure 5.

Furthermore it turned out that the results of average head and efficiency calculated based on transient simulations are slightly higher than compared to the stationary simulation. Thus, the complete head and efficiency curves originally calculated by means of stationary CFD were shifted by the found deviations. By applying this approach an increase in accuracy and reliability of the stationary numerical results compared to measurements is obtained.

4.2. Experimental and numerical results of the pump behaviour for the full operating range

The experimental results of the overall pump performance are shown as solid lines in Figure 7 for the 0° pitch angle and in Figure 8 for the -9° pitch angle. An instability in the hydraulic behaviour in pump operation is present for both pitch angles shown. These instabilities are indicated by multiple \(Q_{\text{ED}}\) values corresponding to one \(n_{\text{ED}}\) value. A stable pump operation near these instability regions is not possible. The turbine characteristic for the 0° pitch angle additionally shows a strong instability region in turbine operation near the runaway point.

The results representing the CFD data are marked with circles in Figure 7 and Figure 8 and were obtained by stationary calculations adding the head and efficiency shift stated in paragraph 4.1. The numerical prediction of the hydraulic behaviour in the pump quadrant shows an excellent correlation.
between the measurements and the numerical data. Also the pump operation mode with negative head as well as the runaway point in pump mode could be predicted by the numerical calculations. Since the pump brake quadrant is of minor importance, only a few operating points were simulated in this region. The numerical results show an acceptable agreement with the experimental data in this quadrant for both pitch angels. The operating point at blocked impeller \( n_{ED} = 0 \) was also captured with satisfactory accuracy by the stationary numerical approach.

**Figure 7** 4-quadrant curves - numerical results compared to experimental data; 0° pitch angle

In the turbine quadrant there is also an adequate prediction of the pump behaviour by stationary CFD calculations. For the -9° blade position also the runaway point \( (T_{ED} = 0) \) was captured with sufficient accuracy (see right picture in Figure 8). As for the 0° pitch angle an instability region could be identified by the numerical approach. The CFD calculated turbine characteristic shows an incorrect location of the predicted instability compared to the experimental data as can be seen in Figure 7. This results in an inadequate prediction of the turbine runaway point for this blade position. Towards the runaway point in the turbine brake region the accuracy of the CFD-results compared to the experimental data deteriorates for both investigated blade angles. It seems that the stationary numerical approach is no longer capable to simulate the flow inside the pump in this operating region as strong secondary flow effects occur inside the machine. These complex flow phenomena require a transient numerical approach to get precise information of the pump behaviour.

**Figure 8** 4-quadrant curves - numerical results compared to experimental data; -9° pitch angle

5. **Evaluation of the 4 Quadrant behaviour by means of a 1D transient system analysis**

Finally an exemplary numerical analysis (water hammer analysis) of a simple hydraulic system according to Figure 9 was carried out. The idea is to investigate the impact of the discrepancies between experimental and the numerically calculated 4-quadrant curves in case of a pump trip. The
hydraulic system consists of two reservoirs with a static head difference of 22 m. The mixed flow pump operates at 507 rpm, has a BEP flow rate of 5 m³/s, produces a head of 27 m at this flow rate and is located directly at the lower reservoir. A pipe of 2000 m length and 1.5 m in diameter connects the pump and the upper reservoir. To model the pipe friction an absolute wall roughness of 0.025 mm was used. No valves or other equipment is installed in this simple exemplary system. The method of characteristics (see [5]) is used to solve the pipe transients. The wave propagation velocity inside the pipe is set to 1000 m/s and the inertia of the pump including the drive is 1000 kgm². The transient system calculations were carried out with the commercial 1D-CFD software Flowmaster V7 with our own extensions for hydropower systems [13].

![Figure 9](image)

**Figure 9** Numerical model for the system analysis

In Figure 10 the results for a calculated pump trip after 5s of stationary pump operation with 0° pitch angle (left) and with -9° pitch angle (right) are shown. Both diagrams give a comparison of the system transients for flowrate (Q), pump rotational speed (n) and the pressure at the pump outlet (p) obtained with the experimental 4-quadrant curve and with the 4-quadrant behaviour resulting from the CFD calculations.

![Figure 10](image)

**Figure 10** Results of the transient system analysis; left: 0° pitch angle, right: -9° pitch angle

The decreasing rotational speed after the pump trip results in an immediate pressure drop on the pressure side of the pump. Due to the inertia of the fluid mass inside the pipe the flow rate decreases slowly compared to the rapid pressure drop after the pump trip. As it can be seen there is hardly any difference between the results gained with the experimental input data and the numerically calculated 4-quadrant curves for both investigated pitch angels. After the pump changes its direction of rotation and also the direction of flow changed a steady state operation in turbine runaway condition occurs. A slight difference in the runaway speed and flow rate is calculated with the CFD input data compared to the experimental input data. This discrepancy is an effect of the inaccurate prediction of the pump behaviour in the CFD calculations near the turbine runaway point.

6. Conclusion

This paper presents an approach to investigate the 4-quadrant behaviour of a centrifugal pump by means of CFD-calculation. With the help of stationary numerical simulations good correlation between the numerically obtained pump characteristics and experimental data was found. Together
with transient simulations the accuracy of the numerical data could be increased further and the main instability in pump mode could be calculated in an even better way. The abnormal pump operation like pump brake or turbine operation could also be predicted correctly with the help of stationary CFD calculations. Further transient calculations in this operating region would yield even higher accuracy in this very special pump operation zone that becomes interesting in case of an emergency due to pump trip. An exemplary transient system calculation proves that the numerically calculated 4-quadrant behaviour leads to equally reliable results compared to an analysis with experimental 4-quadrant input data.

**Nomenclature**

\[
\begin{align*}
Q & \quad \text{discharge} \left[ \frac{m^3}{s} \right] \\
Q_{\text{ED}} & \quad \text{discharge factor} [\text{-}] \\
H & \quad \text{head} [\text{m}] \\
T_{\text{ED}} & \quad \text{torque factor} [\text{-}] \\
g & \quad \text{acceleration due to gravity} \left[ \frac{m}{s^2} \right] \\
n & \quad \text{rotational speed} [\text{rpm}] \\
q_{\text{ED}} & \quad \text{speed factor} [\text{-}] \\
q_s & \quad \text{specific speed} \left[ \frac{\text{rpm}}{\sqrt{m^3/m}}, \text{m} \right] \\
p & \quad \text{pressure} [\text{bar}] \\
u & \quad \text{circumferential velocity at reference diameter} \left[ \frac{m}{s} \right]
\end{align*}
\]

**References**

[1] Gehrer, A., Benigni, H. and Penninger, G. 2004 Dimensioning and Simulation of Process Pumps Karlsruhe Pump Users Technical Forum Preprints 15-4, VDMA

[2] Muggli, F. A., Holbein, P. and Dupont, P. 2002 CFD calculation of a mixed flow pump characteristic from shutoff to maximum flow ASME Journal of Fluids Engineering Vol. 124 pp. 798–802

[3] Laux, C.H. 1982 Reverse-running pumps as energy recovery turbines Sulzer Tech. Rev Vol. 2 1982 pp. 23-27

[4] Nautiyal, H and Varun A. K. 2010 Reverse running pumps analytical, experimental and computational study: A review Renewable and Sustainable Energy Reviews Vol. 14 pp. 2059-2067

[5] Wylie, E. B. and Streeter, V. L. 1993 Fluid Transients in Systems Englewood Cliffs, New Jersey, Prentice-Hall.

[6] Stepanoff, A. J. 1959 Radial- und Axialpumpen Berlin, Springer

[7] Gros L., Couzinnet A., Pierrat D. and Landry L. 2011 Complete pump characteristics and 4-Quadrant representation investigated by experimental an numerical approaches Proceedings of the ASME-JSME-KSME 2011 Joint Fluids Engineering Conference

[8] Couzinnet A., Gros L. and Pierrat D. 2013 Characteristics of Centrifugal Pumps Working in Direct or Reverse Mode: Focus on the Unsteady Radial Thrust International Journal of Rotating Machinery

[9] IEC 60193 1999 Hydraulic turbines, storage pumps and pump-turbines – Model acceptance tests, Second edition 11-1999

[10] Menter, F. R., 1994, “Two-equation eddy-viscosity turbulence models for engineering applications”, AIAA journal, vol. 32, pp.1598–1605.

[11] Menter, F. R., Egorov, Y., 2005, “A Scale-Adaptive Simulation Model Using Two-Equation Models”, AIAA journal, pp. 271–283.

[12] ISO 9906 2012 Rotodynamic pumps – Hydraulic performance acceptance tests – Grades 1, 2 and 3

[13] Hoeller, S., Jaberg, H. 2013 Ein Beitrag zur Druckstoßberechnung von Pumpspeichieranlagen Wasserwirtschaft Vol. 103(1) pp. 78-84