Performance Optimization of High Specific Speed Pump-Turbines by Means of Numerical Flow Simulation (CFD) and Model Testing

Peter Kerschberger and Arno Gehrer
ANDRITZ HYDRO, Graz
A-8045 Graz, Reichsstrasse 68B, Austria
peter.kerschberger@andritz.com, arno.gehrer@andritz.com

Abstract

In recent years, the market has shown increasing interest in pump-turbines. The prompt availability of pumped storage plants and the benefits to the power system achieved by peak lopping, providing reserve capacity, and rapid response in frequency control are providing a growing advantage. In this context, there is a need to develop pump-turbines that can reliably withstand dynamic operation modes, fast changes of discharge rate by adjusting the variable diffuser vanes, as well as fast changes from pumping to turbine operation. In the first part of the present study, various flow patterns linked to operation of a pump-turbine system are discussed. In this context, pump and turbine modes are presented separately and different load cases are shown in each operating mode. In order to create modern, competitive pump-turbine designs, this study further explains what design challenges should be considered in defining the geometry of a pump-turbine impeller. The second part of the paper describes an innovative, staggered approach to impeller development, applied to a low head pump-turbine project. The first level of the process consists of optimization strategies based on evolutionary algorithms together with 3D in-viscid flow analysis. In the next stage, the hydraulic behavior of both pump mode and turbine mode is evaluated by solving the full 3D Navier-Stokes equations in combination with a robust turbulence model. Finally, the progress in hydraulic design is demonstrated by model test results that show a significant improvement in hydraulic performance compared to an existing reference design.

Keywords: pump-turbine, CFD, multi-objective optimizer, 3D-Euler, 3D-RANS, model testing

1. Introduction

A review of the hydraulic turbines markets shows that the demand for pump-turbines has increased in recent years. The main advantage of hydro-storage power plants lies in the option of providing electrical energy to the grid very quickly when it is needed by customers. In order to react to this demand at short notice, there is a need to develop pump-turbines that can reliably withstand dynamic operating modes. In this context, fast changes of discharge rate are required by adjusting the variable diffuser vanes, as well as fast changes from pump to turbine operation.

Furthermore, the overall operating range of a pump-turbine should be well balanced. In pump operation, the stability limits should be located quite far away from the normal operating range, and in turbine operation, it is important to guarantee fast synchronization to the grid, as well as a smooth power rise by opening the guide vanes.

As part of the present study, a runner blade profile is developed for a low head pump-turbine. Existing reference models form the basis [1] for selection of the main dimensions of the hydraulic shape. In the next step, initial runner blade designs are derived by applying a fully automated multi-objective optimizer in combination with a frictionless 3D-Euler CFD code. In order to meet all design criteria, these impellers are investigated, further improved and redesigned using a CAD-based blade design system if necessary. All relevant hydraulic features are evaluated and optimized during the design process by means of CFD.
2. Flow Phenomena in Reversible Pump-Turbines

The operation of a pump-turbine system is usually characterized by various load cases. Both operating modes, pump and turbine, form specific flow patterns that determine the hydraulic behavior of the machinery and should be taken into consideration during the actual design phase. For a better understanding of those hydraulic effects, it is useful to discuss pump and turbine operation separately although they have an impact on one another, as will be shown.

2.1 Targets during the design process

The efficiency level in pump and in turbine operation must be improved during the design process. In addition, the stability limits in both operating modes must be shifted in such a way that the overall operating range can be extended, and finally, adequate cavitation behavior has to be obtained (see Fig. 1). In terms of physics, these targets influence one another and are sometimes contradictory.

As shown in Fig.1, the optimum efficiency point in pump operation is usually located at lower heads than the corresponding point in turbine operation. Considering hydraulic losses at the low pressure and high-pressure sides of the plant, the pump mode operates at higher heads than the turbine mode. Combining these two aspects, the turbine mode of a pump-turbine is usually operated quite far away from its efficiency optimum.

In order to improve turbine performance, the operating range in turbine mode must be positioned closer to its actual optimum at higher heads, which is equivalent to a shift of $H_{\text{max,PU}}$ to higher heads. This procedure is clearly limited by the onset of instabilities occurring at low discharge rates in pump operation.

![Fig. 1 Targets for pump-turbine design](image)

In pump mode, the three-dimensional shape of the runner’s leading edge is responsible for cavitation occurring. In turbine operation, the same edge has substantial influence on the exit swirl leaving the impeller. As source of the draft tube vortex, this exit swirl affects dynamic pulsations and thus limits the normal operating range of the turbine. Thus, widening of the overall operating range of the pump-turbine by shifting the pump operating range to higher heads, for example, and at the same time requiring higher power output in turbine operation would be a clearly contradictory target.

As a design strategy for the current study, blades to which high pressure loading is applied are considered. By increasing the blade loading, the maximum head for pumping operation $H_{\text{max,PU}}$ is shifted to higher values, whereas the correlations between efficiency and head in the turbine characteristics should be retained. Based on pump operating experience, blades with high pressure loading may be unstable at high heads and show a greater tendency towards cavitation. Of course, it is important to bear in mind that blades with high pressure loading result in smaller machine sizing and lower costs.

In summary, we can say that the hydraulic development of pump-turbines is a multi-objective optimizing task. The dynamic behavior of the complete system, as well as the hydraulic performance of each component, must all be taken into consideration. As it forms the hydraulic core of the plant, the present study focused on the runner design.

2.2 Pump operation

In order to define a unique, stable operating point, the pump-turbine should feature a head-capacity curve where only one single intersection point with the head-capacity curve of the hydraulic system is defined. A pump-turbine is usually regulated by setting the variable diffuser guide vanes. In general, each guide vane setting features one head-capacity curve. At a certain discharge rate, the guide vane setting with the highest efficiency level is normally used for operation. Within the normal operating range, most of the head-capacity curves form a single, so-called "envelope" curve.
Instabilities at part-load: As the flow rate is reduced to values below normal operating range, pump operation may become unstable. This manifests itself in a sudden drop in the head-capacity curve. As shown in previous studies [2, 3], the peak of the head-capacity curves mostly coincides with the onset of rotating stall in the diffuser vanes. This is related to the guide vane setting angle (flow rate) and is always accompanied by a fall-off in the rising stage pressure curve. The rotating stall cells rotate in circumferential direction at a speed of 15 to 20% of the impeller tip-speed [2]. These flow separations can be observed at the diffuser vanes (s. Fig. 3, operating point C).

Independently of the rotating stall phenomenon, flow separation occurs on the hub side of the impeller outlet. By further reducing the flow rate, this separation shifts from the hub side to the shroud side of the channel. If the flow rate is throttled further, pre-rotation can be observed at the inlet to the impeller. While the peak of the head-capacity curves mostly coincides with the onset of rotating stall, the trough of these curves is related to the onset of pre-rotation [2]. The magnitude of recirculation increases continuously as the flow is reduced. Unlike flow separation at the diffuser vanes, pre-rotation does not depend on the guide vane setting angle. It is linked to the suction diameter and to the shape of the impeller inlet. The above mentioned types of instability are summarized in Fig. 2.

Cavitation: Cavitation usually appears at load cases outside the normal operating range. The areas at risk are zones with low static pressure, usually on the suction side of the pump-turbine. Runner blade cavitation is usually caused by the flow angle being misaligned to the blade profiles, leading to zones where the local static pressure drops below the vapor pressure of the fluid. With respect to the discharge rate, two different types of cavitation can be distinguished: Suction-side cavitation at part load and pressure-side cavitation at high load. If the flow rate is reduced, the suction-side cavitation increases gradually, whereas pressure-side cavitation grows much faster once triggered.

Using CFD, the beginning of cavitation can be determined by considering a one-phase fluid model and analyzing the static pressure field of the impeller. In order to simulate the influence of cavitation on the efficiency, usually in combination with highly cavitating flow, a two-phase fluid model is required. [4].

In Figure 3, operating point A shows suction-side cavitation, while operating point B shows pressure-side cavitation, both predicted by CFD. The blue areas represent zones where the static pressure is equal to the vapor pressure.

2.3 Turbine operation

Synchronization: As already mentioned, rapid availability of electrical energy is one of the main benefits of pump-storage plants. Thus, there may be frequent cases where the turbine is running at synchronous speed in a no-load condition, with the generator not connected to the grid. As shown in Fig. 4, operating point D, this mode of operation is an unstable condition, with
pressure pulsations and complex vortex structures. In [5], it is shown that the rotor-stator interaction still dominates the flow field in this mode of operation. Furthermore, it is widely agreed that the S-shape of the four-quadrant characteristics of a pump-turbine is mainly responsible for oscillations in no-load operation [5, 6].

**Part-load vortex**: After synchronization to the grid, the flow rate and the power output of the turbine are increased by opening the guide vanes. Still at part load, the flow in the draft tube develops an unsteady rotating flow pattern, the so-called part-load vortex, which leads to pressure pulsations. In Fig. 4, operating point C, the part-load vortex is visualized by means of CFD and compared to a corresponding figure captured during model testing [7]. With respect to the rotational frequency of the impeller, the pressure pulsations occurring may be split into an asynchronous and a synchronous component. The asynchronous frequency originates from the precession of the vortex and is mainly responsible for the loud noise. The synchronous component is a through-flow pulsation that can cause severe damage if the frequency is close to the natural frequency of the system [7].

**Normal operating range**: Within the normal operating range, the flow field is uniform within the blade channels and through the entire machine, there is no flow detachment, and no vortices are generated. Figure 4, operating point A, illustrates the flow near the optimum efficiency point of the turbine at the highest head \( H_{\text{max}} \).

**Full-load vortex**: At high loads, the helical vortex shape of the part-load vortex disappears. Usually, the full-load vortex has a rotationally symmetric shape and does not show any vortex precession (see Fig. 4, operating point D). Thus, there are no asynchronous pressure pulsations [8]. In order to obtain the amplitudes and frequencies of the pressure pulsations caused by a full-load vortex, the analytical-empirical 1D assumptions as shown in [8] can be used.

In general, it is very difficult to capture all aspects related to transient operating conditions by means of CFD. Regarding vortex structures in the draft tube, cavitation behavior is not negligible [7]. Thus, CFD calculations considering a two-phase fluid model have to be performed. These simulations are still very time-consuming. As a result, the respective methods are still not a practicable solution nowadays in a standard design process.

### 3. Hydraulic Design

As shown in Fig. 5, the hydraulic design is made in a staggered mode.

Starting with an inviscid 3D-Euler optimization, the best designs are then evaluated by means of 3D-Navier Stokes simulations.

Finally, suitable geometries are released for model testing. Possible uncertainties regarding the predictive quality of numerical flow calculation are compensated by consequent interaction between model test and numerical simulation.

---

![Fig. 4 Flow phenomena in turbine operation: A BEP at \( H_{\text{max, TU}} \), B full-load vortex, C part-load vortex, D synchronization](image)

---

![Fig. 5 Hydraulic Design Process](image)
3.1 Blade design and 3D-Euler optimization

The geometry of the pump-turbine under consideration is defined by means of a CAD-based parametric blade design system [1]. With this tool, it is possible to modify the three-dimensional shape of the blade in terms of feasible geometric parameters (e.g. blade angles, wrap angles, meridional contour …).

Based on an initial blade design, the hydraulic analysis is performed using a frictionless 3D-Euler code. This CFD code is a powerful tool for obtaining information very quickly on the pressure and velocity field inside the blade channels. The code was derived from a fully compressible, high-speed Navier-Stokes code for transonic flows in turbo machinery [10]. The incompressible version solves the governing equations by means of an artificial density method. As it does not consider friction terms, no direct conclusion can be drawn on hydraulic losses and efficiency.

Nevertheless, all relevant flow patterns are present, and tendencies can be derived relating to pressure gradient-based secondary flows or flow separations. In order to forecast cavitation behavior (e.g. expressed by the THOMA coefficient σ), a statistical histogram evaluation of the pressure field, as described in Section 3.3, is used.

Regarding hydraulic optimization, two different methods have been applied within the scope of the present work: On the one hand, blade geometries were tuned “by hand” and, in addition, an automated optimizer has been applied. With the first approach, experienced blade designers reshape the three-dimensional geometry manually. This procedure requires a thorough understanding of the flow phenomena discussed in the previous chapters.

As outlined in ref. [14], an optimizer based on evolutionary algorithms is coupled with the 3D-Euler code and with the blade generator. All characteristic values, such as objectives and constraints, are extracted during post-processing. Since the optimizer supports multi-objective optimization, the following objectives are defined:

- **Pump mode objectives**: The goal is to achieve minimum values for the Thoma coefficient (σ) at two operating points (typically at part load and close to full load). The respective (σ) values (as defined in Section 3.3), are extracted from the pressure distribution.
- **Turbine mode objective**: The desired objective is a minimum of deviation between the velocity profile of the flow leaving the runner domain and a prescribed velocity distribution (close to the optimum efficiency point). The target profile is an evenly distributed axial velocity profile and a minimum of swirl.

All objectives, the “sigma values” and the “profile values”, are multiplied by weighting factors and collected for the operating points considered. The optimizer then analyzes these values, selects “parents” with evolutionary strategies, and mutates them to new individuals. Each individual consists of geometry parameters defining a specific blade design. An entire optimization cycle represents one generation, typically consisting of approximately 100 individuals.

3.2 Reynolds-averaged Navier-Stokes equations

For the most promising designs coming from the in-viscid optimization, a more detailed flow analysis is carried out as a next step, including friction and turbulence. This analysis provides results on losses and hydraulic efficiencies. If the results contradict the 3D-Euler optimization, the in-viscid optimization loop has to be repeated, this time taking account of the results of the viscous simulations.

In this study, the Reynolds-averaged Navier-Stokes (RANS) equations in conjunction with a two-equation turbulence model are solved by the commercial flow solver ANSYS CFX5 V.12. Furthermore, the standard k-ω model with scalable, logarithmic wall-functions is used [11].

Considering the normal operating range, computations are performed in steady-state mode. In turbine mode, one guide vane channel, one runner blade channel, and the draft tube are considered, whereas in pump mode, the draft tube is disregarded. The guide vane channels and also the runner-blade channels are linked in circumferential direction by means of periodical interfaces. A mixing plane approach (stage interface) is applied between the rotating and stationary components.

The stay vane, the guide vane, and the runner blade channels are meshed by means of an in-house parametric grid generator. The mesh of the draft tube is generated by using the commercial mesh tool ICEM CFD. The mesh size for the computational domain is about 2.15 million nodes for simulation of turbine mode and 0.8 million nodes for pump mode. Structured multi-block meshes are used for the runner and the guide vane channels, and an unstructured mesh is applied for the draft tube.

The simulation process is performed fully automatically. The setup is performed by means of ANSYS CFX5 V.12 macro files, considering the corresponding meshes, domain characteristics, boundary conditions, and solver settings. The actual simulation is then run on a powerful LINUX Cluster.

3.3 Cavitation analysis

It is convenient to characterize the potential for cavitation by means of a cavitation number σ that characterizes the difference between the suction head and the vapor pressure given by the actual operating conditions (p,T). Thus, the tendency of a pump-turbine to suffer cavitation is dictated by the Thoma cavitation coefficient σ, defined as

\[
\sigma = \frac{p_{\text{tot, suction-side}} - P_v}{\rho_L g H}
\]

Here, \(p_{\text{tot, suction-side}}\) is the total pressure at the low-pressure side of the pump-turbine, \(g\) the constant of gravity, \(P_v\) the vapor pressure, and \(\rho_L\) the density of the liquid. \(H\) corresponds to the hydraulic head defined by the difference in total pressure between
the machine inlet and outlet. In order to forecast cavitation behavior by means of CFD, a corresponding \( \sigma_{\text{CFD}} \) value derived from a statistical histogram analysis is used [12]. \( \sigma_{\text{required,CFD}} \) is defined as:

\[
\sigma_{\text{required,CFD}} = \frac{p_{\text{rot, suction} - \text{side}} - p_{\text{histo@blade}}}{\rho g H}
\]

(2)

4. Case Study, Numerical Results

For the current development project, a suitable reference model with high specific speed \( n_{sq,\text{PU-opt}} \approx 260 \text{[rpm]} \) (see e.g. [13]) is selected. As commonly practiced, the following relation is used to define specific speed

\[
n_{sq} = 3.65 \cdot n \cdot \frac{\sqrt{Q}}{\sqrt{H^3}}
\]

(3)

In a first step, the normal operating range of this reference model was simulated numerically and the corresponding results were compared to the values measured. The broken black lines in Fig. 6 show the experimental data of the reference model, while the red lines depict the corresponding CFD results. In view of these results in pump mode, the head-capacity curve fits very well, while the efficiency curve shows some discrepancies at full load, but indicates the onset of instabilities. In terms of cavitation, a reasonable level of accuracy could be achieved by means of the histogram analysis.

In turbine mode, the general shape of the propeller curve is captured, however, there are some small discrepancies at lower discharge rates, but the location of the optimum efficiency point, as well as the efficiency at full load, is captured.

In order to address the requested design features described in Section 2, a thorough design optimization by means of CFD was performed. To increase stability at high heads in pump mode, the flow field within the blade channels at the operating range shown was investigated and design changes were applied so that flow separation is avoided. The histogram analysis has been used extensively to improve cavitation protection by flattening the \( \sigma \) curve. In turbine operation, the vortex-free operating zone was widened and flat propeller curves were obtained by creating uniform exit velocity distributions. Due to the high load on the blades, the overall machinery size is reduced by 4.5 percent.

Following this detailed optimization process, the final runner, corresponding to the blue lines in Fig. 6, was released for model testing. In pump operation, this runner shows a steeper slope in the head-capacity curve and indicates a slight shift of instabilities to higher heads compared to the reference model. Under \( \sigma_{\text{plant}} \), the cavitation-free zone is widened by shifting the onset of suction side cavitation to lower flow rates. As far as the overall efficiency level is concerned, a significant improvement can be noted. In particular, the highly weighted points in turbine operation at full loads (high power output) are improved.

5. Case Study, Model Test Results

As shown in Fig.7, the model test results of the released runner confirm the CFD tendencies illustrated in Fig. 6. A significantly higher efficiency level is achieved for the overall operating range. In slight contradiction to CFD, the efficiency level at full load in pump mode does not drop slightly as predicted. For the head-capacity curve, only a slightly steeper slope is obtained. The pump stability limit is improved, the onset of instabilities is shifted to higher heads, and the fall-off in the head-capacity curve
is less drastic. In turbine operation, the propeller curve is even flatter than expected, in particular with no efficiency losses at part load.

![Diagram showing model test results, reference and optimized design](image)

**Fig. 7** Model Test Results, Reference and Optimized Design

### 6. Conclusion

It has been explained that the overall operating range of pump-turbines is determined by various different flow conditions. In pump mode, instabilities impacting the head-capacity curve in combination with cavitation occurrences limit the normal operating range. In turbine mode, draft tube vortices linked with pressure fluctuations are the main determinants for smooth operation. During synchronization, as well as during operation at extremely low flow rates, the flow field becomes entirely separated and unstable. Considering the combination of these effects, the discussion demonstrated that designing a pump-turbine is a multi-objective and sometimes contradictory task.

The computational methods mainly used in the present study were described. It was stated that a multi-objective optimizer was used to generate blade designs. The hydraulic analysis was then performed in a staggered mode: Firstly, a frictionless investigation was carried out by means of a 3D-Euler code, followed by a fully 3D-Navier Stokes simulation, considering friction and turbulence.

With respect to the normal operating range, steady-state computations were conducted for an adequate reference design and for an optimized design. Compared to the experimental data, the CFD results showed a feasible level of accuracy.

For future projects, it would be useful to capture transient effects routinely in an early design phase by means of unsteady CFD.
Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
<td></td>
</tr>
<tr>
<td>g</td>
<td>Constant of gravity [m/s²]</td>
<td></td>
</tr>
<tr>
<td>H, H_p</td>
<td>Hydraulic head [m]</td>
<td></td>
</tr>
<tr>
<td>n</td>
<td>Speed of rotation [rpm]</td>
<td></td>
</tr>
<tr>
<td>n_sq</td>
<td>Specific speed of rotation [rpm]</td>
<td></td>
</tr>
<tr>
<td>p</td>
<td>Local static pressure [Pa]</td>
<td></td>
</tr>
<tr>
<td>p_v</td>
<td>Vapor pressure [Pa]</td>
<td></td>
</tr>
<tr>
<td>p_tot</td>
<td>Total pressure [Pa]</td>
<td></td>
</tr>
<tr>
<td>Q</td>
<td>Flow rate [m³/s]</td>
<td></td>
</tr>
<tr>
<td>T</td>
<td>Local temperature [K]</td>
<td></td>
</tr>
<tr>
<td>α_GV</td>
<td>Guide vane setting angle [°]</td>
<td></td>
</tr>
<tr>
<td>η</td>
<td>Efficiency [-]</td>
<td></td>
</tr>
<tr>
<td>ρ_t</td>
<td>Fluid density [kg/m³]</td>
<td></td>
</tr>
<tr>
<td>σ</td>
<td>Cavitation coefficient [-]</td>
<td></td>
</tr>
<tr>
<td>r_Cu</td>
<td>Swirl [m²/s]</td>
<td></td>
</tr>
</tbody>
</table>

References


Peter Kerschberger was born in 1983 in Graz, Austria. He studied mechanical engineering with focus on energy and environment at Graz Technical University. After graduating in 2008, he joined ASTROE, a laboratory for hydraulic machines within ANDRITZ HYDRO. He is mainly involved in the hydraulic development of pumps and reversible pump-turbines as a CFD and research engineer.

Arno Gehrer has been a research engineer at ANDRITZ HYDRO since 2001. He obtained his Ph.D. in mechanical engineering from the University of Graz in Austria, where he worked as research assistant and assistant professor at the institute for Thermal Turbomachinery and Machine Dynamics. At present, he heads the group for hydraulic development at ANDRITZ AG in Graz, and his main focus is the design of pumps and turbines, both with CFD and model testing.