Multistage canned motor pump – getting deep insights with CFD – simulation, optimization and validation

M Mosshammer¹, H Benigni¹, H Jaberg¹, S Krämer² and C Dahlke²

¹ Institute of Hydraulic Fluid Machinery, University of Technology, Kopernikusgasse 24/IV, 8010 Graz, Austria
² HERMETIC-Pumpen GmbH, Gewerbestraße 51, 79194 Gundelfingen, Germany
mosshammer@tugraz.at

Abstract. This paper describes the detailed analysis of a multi-stage canned motor pump by means of CFD (computational fluid dynamics) simulations to find the main losses and show optimization potential of the hydraulic parts. In a first step, the numerical model was successively generated including all details like impellers, guide vanes (GV), return vanes (RV), gaps and pressure relief holes as well as the hydraulically coupled rotor of the drive. Simulations were carried out with the commercial CFD package ANSYS CFX 17.1 in stationary and transient way where mainly structured grids were used with a final model of the existing pump consisting of more than 35 mio. nodes. The behavior of the components was analyzed in detail and additionally the CFD-simulations were validated with model tests. Based on the loss analysis, a 2-stage optimization procedure was performed by combining manual engineering with automated optimization to establish the desired objectives and validated with the full numerical model. The simulations predict a huge efficiency increase from part load up to the best efficiency point (BEP) with respect to all given limitations (identical head curve, suction behavior and dimensions) and were then validated on the test rig where they have proven their high accuracy and reliability.

1. Introduction

If an outstanding safety of a pump is required, hermetic pump concepts like a canned motor or a magnetic coupling, which require higher energy consumption, are commonly used. In the past, poor efficiency of special pumps like the investigated one was a minor problem as their main purpose was to be a reliable powerful workhorse. Nowadays, companies are pushed by law to increase the efficiency of their products like the Energy Efficiency Directive of the European Union or the Paris Agreement. As pumps almost require 20% of the world electric power consumption [1], they offer a huge potential of saving energy.

As the investigated pump is designed for high head and small flowrates, like many process pumps, the specific speed \( n_q \) as defined in equation (1) is relatively low (stage specific speed is around 30 (rpm, m³/s, m)) and therefore the hydraulic efficiency of the pump is rather moderate. This is mainly due to physical causes like disc friction losses of the runner, gap losses and due to the design as a multistage pump which requires guide and return vanes which also generate losses. Another limiting factor is the quite low flowrate of around 0.003 m³/s in BEP and the usage of a canned motor as pump drive. Taking all these points into account, available literature like [2], [3], [4], [5] predict a feasible
hydraulic efficiency of around 50 - 60 %, although none of the mentioned sources include losses from pressure relief holes nor the usage of a canned motor.

\[ n_q = n \cdot \frac{Q^{1/2}}{H^{1/4}} \]  

(1)

As there are no accessible publications of numerical simulations for a multistage canned motor pump so far, this paper presents an approach for detailed flow and loss analysis and an optional optimization.

2. Design and basic functionality of a multistage canned motor pump

The investigated multistage canned motor pump is shown in figure 1 and consists of 3 stages, whereas the first stage is especially designed for an excellent suction behavior, and the canned motor. The following components are taken into account:

• Impellers
• Canned motor
• Suction piece of the first stage at the inlet and pressure housing at the last stage
• Pressure relief holes
• Return channel consisting of guide and return vanes
• Hub and shroud cavities with gaps along the shaft
• Shaft (hollow) with return of motor side stream in suction area of stage 2

![Figure 1. Cross section of a 3-stage canned motor pump with modelled components and flow paths.](image)

To a large extent the investigated pump complies with a “normal” multistage pump as fluid passes through all stages. The main difference is the required motor side stream for heat removal of the hydraulically separated rotor and stator of the canned motor. Therefore, the motor side stream needs to cover large surfaces for best possible cooling performance and then be returned in the regular flow. As already mentioned, hub and shroud cavities, pressure relief holes and gaps were modelled for best possible accuracy which leads to quite complex flow situations inside the pump as depicted in figure 2. Though flow rate at inlet and outlet of the pump are identical, each stage and even each impeller of
the pump has to handle different flow rates. Additionally, the inflow of all stages is different, as the first stage mainly depends on the inflow conditions, the second stage on return channel and return of the motor side stream and the third stage on the return channel. To show the influence of the various components on head and efficiency, different models were used as will be shown in chapter 4.

![Figure 2. Definition of system limits and flow paths inside of the investigated pump.](image)

3. Methodology of numerical simulations

3.1. Preparation of the geometry and mesh generation

In a first step, the real pump had to be transferred in a 3D-CAD model. For that, drawings and dimensions of the pump were used in addition to a 3D-Laserscan of the impeller and return channel. Based on the 3D-CAD model, the geometry was prepared for meshing. As an example, the impeller is mentioned as it consists of 6 identical blades (1 blade equals to a section of 60°), where only one was used for meshing. To generate a suitable, structured mesh, ANSYS TURBOGrid© was used. The final mesh for the impeller is shown in figure 3 which shows the applied near wall inflation to resolve the boundary layer of the viscous flow. Numerical meshes for the remaining parts were mostly generated with ANSYS Meshing©, combining structured and unstructured grids depending on the complexity of the geometry. The averaged $y+$ value for all meshes at a flowrate around BEP of the pump is around 20 to 40.

Besides a correct modelling of the impeller, return vanes and canned motor, it is vital to mention, that the resulting head and efficiency of a pump with quite low specific speed, and especially of a multistage pump, relies heavily on the existing gaps. The modelled gaps between impeller and housing as well as between the rotor and stator of the canned motor are between 0.2 and 0.35 mm. To get an impression of the size of the gaps, it is vital to mention that they equal to 0.17 % respectively 0.3 % of the impeller diameter. Those gaps in addition to the hub and shroud cavities and the stages itself are of essential importance for reliable simulation results.

To guarantee high accuracy, a mesh study for a single stage was performed to show the influence of different refined numerical meshes on pump head and efficiency. The results are shown in figure 4 which yield at least a fine mesh resolution to eliminate mesh influence on numerical results. Given that refinement of the mesh, the full model of the pump consists of 35 mio. nodes. (informative: medium/fine mesh: 22 mio. nodes; very fine mesh: 51 mio. nodes)
3.2. Numerical model of the pump

Based on numerous successfully performed projects in the field of hydraulic fluid machinery at the institute within the last decade [6], [7], [8], a wealth of experience was gained. The simulations (carried out with ANSYS CFX 17.1) of the investigated pump are primarily based on this knowledge – especially the definitions of the numerical model and the solver (turbulence modeling, boundary conditions, interfaces, iterations…), and approaches of modeling hub and shroud cavities as well as pressure relief holes and gaps.

As stationary simulations mostly give a good first impression on the flow situation it is vital for getting the most accurate results, to perform also transient simulations. The used turbulence model for stationary simulations (RANS) was k-ω-SST-turbulence model from [9], combining the 2-equations-turbulence-models k-ε- and k-ω. This turbulence-model requires an adequate mesh for modeling near wall (k-ω) and wall distant (k-ε) flow regimes. Though the SST-turbulence model could be used for transient simulations (URANS), experience has shown that it does not always provide satisfying results, even if the grid and time step resolution would be sufficient for that purpose. That is why the scale resolving turbulence model SAS-SST from [10] was utilized. It combines the advantages of the SST-model in steady zones but enables to switch to SRS mode in flows with large and unstable separation zones which are likely to be present between impeller and return channel.

For the stationary calculations, a mixing plane approach was chosen for the multiple frames of reference interface 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 \( L = 10 \times D_{\text{Inlet}} \) away from the pump inlet) an average static pressure was applied whereas at the outlet (defined at a distance of \( L = 20 \times D_{\text{Outlet}} \) away from the pump outlet) the mass flow rate was specified as boundary condition.

During stationary simulations, between 600 and 2000 iterations were required depending on the convergence of each operating point. The evaluation of the convergence was based on two factors. First on the RMS-weighted residuals and second on the behavior of the defined monitor points (e.g. pump head, efficiency, motor side stream). When residuals were sufficiently small and the fluctuations of the monitor points negligible, the simulations were stopped. For the calculation of the results, the arithmetic average of the last 100 iterations was used.
For transient simulations, which used the corresponding stationary result for initial values, the Transient-Rotor-Stator approach with an adaptive timestep was used. The first revolution used a timestep which equals for 12° and the second revolution used 2°-steps. They were mainly used as settling time for the simulation. Afterwards the timestep was reduced to get a resolution of 1° for the next two revolutions, whereas only the last one was used for calculating the results. A summary of the used numerical settings for stationary and transient simulations is given in Table 1.

| Table 1. Settings for the numerical simulations. |
|-----------------------------------------------|
| Inlet                                       |
| Stationary Simulation                      |
| Average static pressure                     |
| Outlet                                      |
| Mass flow                                   |
| Turbulence model                            |
| SST                                         |
| Fluid                                       |
| Water (single phase), 20°C                  |
| Timestep                                    |
| Iteration 1 to 25: 1/ (10 * ω)              |
| Iteration 26 to 50: 1/(5 * ω)               |
| Iteration 51 to 100: 1/(2 * ω)              |
| From Iteration 101: 1/(ω)                   |
| „Coefficient loops“                         |
| RMS-Residuals < 1E-5                        |
| Convergence criteria                        |
| Criteria for single monitor points         |
| „Conservation Target“                        |
| Max. number of iterations depending on operation point |
| 0.005                                       |
| Transient Simulation                       |
| Average static pressure                     |
| Mass flow                                   |
| SAS-SST                                     |
| Water (single phase), 20°C                  |
| 1st revolution: 12/360 * 2 π / ω           |
| 2nd revolution:: 2/360 * 2 π / ω           |
| Revolutions 3 to 4: 1/360 * 2 π / ω        |
| -                                           |
| 10                                          |
| 0.005                                       |

3.3. Post-Processing

A major advantage of simulations compared to measurements is the infinite possibilities of the analysis of the results. Beside conventional measurement data like head curve, efficiency and shaft power it is possible to calculate quantities and data like:

- Velocity components (cm, cu,...) of the impeller
- Flow interaction between impeller and guide vanes as well as between guide and return vanes
- Flow situation of all components to detect possible flow separation and backflow zones
- Leakage flow in pressure relief holes and gaps

Given that kind of information, it is possible to get deeper insights in the functionality and behavior of the pump and its components. The whole post-processing was carried out with ANSYS CFX Post 17.1 and some quantities are described in detail below.

Net head: In general it is defined as the difference between total pressure head at the outlet and total pressure head at the inlet of a pump. According to ISO 9906 standard, the net head represents the difference between the static pressure plus the mean kinetic energy head at outlet and inlet. As the pressure on the test rig was measured on 4 pressure measuring taps (which are being positioned circumferentially around the pipe with an angle of 90° between them) with their locations being set 2D away from inlet and outlet according to ISO 9906, the post-processing of the CFD results was then carried out in a similar way as given in equation (2).

\[
H = \frac{1}{\rho g} \left[ \frac{1}{A_{\text{Outlet}}} \left( \int p_{\text{stat}} \cdot dA \right) |_{\text{Outlet}} - \frac{1}{A_{\text{Inlet}}} \left( \int p_{\text{stat}} \cdot dA \right) |_{\text{Inlet}} \right] + \frac{Q_{\text{Outlet}}}{2g A_{\text{Outlet}}} \cdot \frac{Q_{\text{Inlet}}}{2g A_{\text{Inlet}}} \tag{2}
\]
Hydraulic efficiency: Is defined as ratio of hydraulic power to required shaft power of the pump and described by equation (3).

\[ \eta = \frac{H \cdot Q \cdot \rho \cdot g}{T \cdot \omega} \]  

(3)

4. Results

4.1. Comparison of different model depth

To show the influence of the components, five different numerical models as shown in figure 5 (bottom) were generated. Each model includes all three stages and the fine mesh settings.

Model A: 6 mio. nodes – including straight pipes as inflow (10xD_{Inlet}) and outflow (20xD_{Outlet}) regions, impellers as 1/6 model and return vanes as full model (360°).

Model B: 8 mio. nodes – based on model A, including the canned motor with hollow shaft for returning the motor side stream.

Model C: 25 mio. nodes – based on model B (still only 1/6 model of the impellers) but including all gaps and hub and shroud cavities. As gaps are quite narrow, the required mesh size to model them accurately increased by a factor of 3.

Model D: 30 mio. nodes – based on model C, but impellers are fully modelled (6/6).

Model E: 35 mio. nodes – based on model D, including pressure relief holes.

The canned motor has a significant influence on head and efficiency which can be seen in figure 5 by comparing model A and B. The reduction of the pump head is mainly due to the increased discharge of stage 3 but also due to the diversion of the motor side stream which causes a more disturbed flow situation between outflow of the impeller and outlet of the pump. Those factors are also the main reason, beside the friction losses of the rotor of the canned motor, for the reduced efficiency of approximately 4 %.

The pump and efficiency characteristics are also strongly changing when gaps and hub and shroud cavities are introduced as the comparison of model B and C in figure 5 shows. The reduced pump head is caused by the additional leakage flow through front and hub cavities and therefore the possible head...
is lower. An even stronger influence can be observed at the efficiency which drops nearly by 7% around BEP and is caused by the impeller friction losses which are of high significance when pumps with low specific speeds and small size are investigated. Therefore the simulated efficiency decrease matches quite good with research findings from [11], [12] who predict impeller friction losses of 6 to 8% for a single stage pump with a similar specific speed.

A bit surprising was the outcome between model C (1/6 impeller blades) and D (6/6 impeller blades) as periodic effects are quite common in turbomachinery applications. With the same numerical settings, model D yields a noticeable higher pump characteristic (∼ 2 m). This is mainly caused by the interaction of impeller and guide and return vanes, as averaging of fluxes leads to unavoidable deviations when a different number of blades and vanes are used. This simplification in model C also causes an overestimated efficiency.

The results of model E, which represents the real hydraulic with best possible accuracy, show a reduced pump head of around 2-3 m and a reduced efficiency of around 2% due to the implementation of pressure relief holes. [13] have shown comparable values for a single stage pump with similar specific speed and leakage flows.

It is vital to mention that the presented results in figure 5 were computed in stationary mode and with hydraulically smooth walls.

4.2. Loss analysis in full model E

To optimize a certain problem, it is necessary to understand the causes which, in this case, lead to a loss analysis of the main components (described in detail in [6]) as shown in figure 6 (left):

- Impeller losses: including friction losses as well as shock and conversion losses in the impeller channel
- Hub cavity: including friction losses of the runner disc moving in the hub cavity as well as leakage losses (mainly due to pressure relief holes)
- Shroud cavity: including friction losses of the runner disc moving in the shroud cavity as well as leakage losses (mainly caused by the height of the gap between runner and casing)
- Guide and return vanes: losses between guide vane inlet and impeller inlet of the next stage
- Canned motor: including friction losses of the rotor and required pump head to maintain the motor side stream

In addition, the hub and shroud cavity losses were split up in friction and gap losses to specify them as accurate as possible and are depicted in figure 6 (right).

**Figure 6.** Different loss analyses for the investigated original 3-stage canned motor pump.
A comparison of the occurring losses with investigations of pumps with similar specific speed is shown in Table 2. Impeller and disc friction losses are of the same size as Kagawa [14] and Pfleiderer [11] have predicted and get smaller with increased pump size (Osterwalder [15]). As gap losses are directly connected to the height of the gaps, it is obvious that they offer a huge optimization potential. In this investigated case the gaps must not get smaller as the size is specified by API 685 and API 610 due to the field of application. Given that kind of limitations, the main optimization potential is within the guide and return vanes of the pump which account for almost one third of the overall losses (excluding the canned motor as it is mandatory).

Table 2. Breakdown of losses in a \( n_q \approx 20 \) rpm pump.

|                | Impeller and disc friction | Gaps | GV & RV |
|----------------|---------------------------|------|---------|
| Kagawa (small pump) | 15                        | 15   | 15      |
| Pfleiderer (small pump) | 14                        | 4    | -       |
| Osterwalder (large pump) | 9                         | 2    | 4       |
| CFD - Modell E (small pump) | 20                        | 15   | 15      |

4.3. Comparison of measurement data and numerical results

The comparison of the measurement data gained in the laboratory of the Institute of Hydraulic Fluidmachinery and the results of the transient simulations of model E show a nearly perfect match over the whole investigated range for pump head as well as efficiency as depicted in figure 7. Based on the proven accuracy of the numerical simulations, the optimization process according to figure 8 was performed.

Figure 7. Comparison of numerical and measurement data for pump head and efficiency.

Figure 8. Chosen strategy combining manual and automated optimization.

5. Optimization

Due to the high numerical effort of the full numerical model E of the presented multistage canned motor pump, this is not recommended for optimization tasks. Therefore a two-stage optimization process according to figure 8 was used. As shown on left hand side, the impeller and the pressure stage casing were optimized manually for what a simplified 1-stage model with no gaps and hub/shroud cavities were used to speed up the simulation time.
Given that the guide and return vane offer the largest optimization potential, a fully parametrized geometrical model was generated to conduct an automated multi objective optimization as depicted on the right hand side of figure 8. Therefore a simplified 1.5-stage model was generated which consists of one stage including impeller and return channel and a subsequent stage with only the impeller to analyze the outflow conditions of the RV on the impeller performance. For optimization tasks, the number blades of GV and RV were used as parameters as well as four additional parameters for guide vanes and 9 parameters for return vanes – adding up to an overall 15 parameters. To cover the resulting parameter domain, more than 300 geometry variations were generated with the “optimal space filling design” inside ANSYS Workbench and for each geometry, three operating points were simulated.

After all simulations were finished (which took more than 70 days on a high performance workstation with 24 cores and 96 GB RAM), a sensitivity analysis was conducted. This was used to generate a meta-model, which describes a theoretical relation between input and output parameters. Based on the defined goals (head, position of BEP and highest possible efficiency) a “Multi-Objective-Genetic-Algorithm” (MOGA) was used for optimization. Based on the outcome of the optimization, the best candidates were then validated in the full model, as the underlying data of the automated optimization was based on the simplified model. The results for GV and RV for the best candidate and a comparison with the original geometry are presented in figure 9.

The results of the optimized pump, consisting of new impellers, a new pressure stage casing and the new return channels are shown in figure 10. Up to a discharge of approximately 4.6 l/s, a significant efficiency increase in comparison to the original pump can be found. Additionally the position of BEP was shifted to a lower flowrate which was a mandatory requirement (besides retaining the number of stages and dimensions of the pump). As the pump is commonly used in part load, the efficiency increase of up to 8 % (absolute) can be taken full advantage of. The nearly identical head curve must be especially highlighted as this was another boundary condition of the project.

6. Conclusion
The presented numerical model of the investigated multistage canned motor pump has been proven high quality and reliability as both pumps, the original and optimized one, were simulated with high accuracy. With this model it is possible to get deep insights in the loss mechanics of the pump and is therefore perfectly suitable for optimization tasks. In the shown case, a lot of boundaries like identical head curve, number of stages, gap widths and overall dimensions had to be complied with. The chosen optimization approach yields a major efficiency increase as well as a shifted BEP towards a lower flowrate. A comparison at BEP of the original with the optimized pump and with proposed characteristics of [2], [3], [4], [5] in figure 11 shows a satisfying result as efficiency could be increased clearly though the specific speed of the pump was decreased due to shifting BEP towards part load by keeping the head curve identical. Though the resulting efficiency is below the suggested
characteristics, it is vital to mention that none of them contains pressure relief holes, canned motor or large gaps as are existent in the investigated pump.

![Figure 10. Comparison of numerical and measurement results for pump head and efficiency for original and optimized pump.](image)

![Figure 11. Comparison of original and optimized pump efficiency with reference values from various publications.](image)

**References**

[1] Almeida A T 2008 EuP Lot 11 Motors (Coimbra: Final report to the European Commission)
[2] Güllich J F 2013 Kreiselpumpen (Berlin: Springer)
[3] Karassik I J 2001 Pump Handbook (New York: McGraw Hill)
[4] Van Esch B P M and Kruyt N P 2001 Hydraulic Performance of a Mixed-Flow Pump: Unsteady Computations and Loss Models J. Fluids Eng. 123 256
[5] Neumaier R 1994 Hermetische Pumpen (Sulzbach: Faragallah Verlag)
[6] Mosshammer M 2018 Untersuchung und Optimierung einer mehrstufigen Spaltrohrmotorpumpe (Graz: PhD thesis)
[7] Höller S 2016 Cavitation optimization of a variable pitch mixed flow pump for cooling water by numerical methods and test rig verification (Düsseldorf: IREC Session 18-2)
[8] Benigni H Jaberg H 2012 Numerical Simulation of Low Specific Speed American Petroleum Institute Pumps in Part-Load Operation and Comparison With Test Rig Results J. Fluids Eng. 134 024501-1
[9] Menter F R 1994 Two-equation eddy-viscosity turbulence models for engineering applications AIAA 32 1598
[10] Menter F R Egorov Y 2010 Scale-Adaptive Simulation Method for Unsteady Flow Predictions. Part 1: Theory and Model Description J. Flow Turbul. Combust. 85:1 113
[11] Pfeifer P Petermann H 2004 Strömungsmaschinen (Berlin: Springer)
[12] Feng J Benra F Dohmen H J 2006 The Influence of Leakage Flow on the Main Flow in a Radial Pump Stage (Yokohama: 23rd IAHR Symposium)
[13] Will B 2011 Theoretical, Numerical and Experimental Investigation of the Flow in Rotor-Stator Cavities with Application to a Centrifugal Pump (Duisburg: PhD thesis)
[14] Kagawa S 2006 Loss Analysis and Performance Improvement of Very Low Specific Speed Centrifugal Pumps (Yokohama: 23rd IAHR Symposium)
[15] Osterwalder J Hippe L 1984 Guidelines for Efficiency Scaling Process of Hydraulic Turbomachines with different technical Roughnesses of Flow Pass J. Hydraul. Res. 22:2 77