EXPERIMENTAL AND NUMERICAL RESEARCH OF THE EFFECTS OF A PERTURBED INFLOW OF A CENTRIFUGAL PUMP

Dipl. Ing. M. Roth
Chair of Turbomachinery and Fluid Power
Darmstadt University of Technology, Germany
roth@tfMaschinenbau.tu-darmstadt.de

Prof. Dr. Ing. B. Stoffel
Chair of Turbomachinery and Fluid Power
Darmstadt University of Technology, Germany
stoffel@tfMaschinenbau.tu-darmstadt.de

Dr. Ing. G. Ludwig
Chair of Turbomachinery and Fluid Power
Darmstadt University of Technology, Germany
ludwig@tfMaschinenbau.tu-darmstadt.de

ABSTRACT

The influence of a disturbed inflow condition on a single-stage, end suction volute casing pump, as they are frequently found e.g. in water supply plants, was analyzed experimentally in a test bench and numerically using CFD simulation.

The analyzed pump operates with water and has a specific speed (n_s) of 42 (metric units), an outside impeller diameter (D_a) of 188 mm and an operating speed (n) of 2300 rpm. The measurements were carried out in a closed test loop where the operating conditions and the inlet piping can be varied to analyze the influence of bend and valve over the pump performance.

For the 3-D numerical analysis two configurations were simulated. First the combination bend and butterfly valve was calculated and second the complete pump. Different coupling methods between fixed and rotating parts and some turbulence models were analyzed.

For the comparison between experiment and simulation the characteristic curves of the pump are used. Also the calculated flow distribution directly upstream of the suction branch of the pump was compared with the velocity profile measured non-intrusively with a Laser Doppler Velocimeter (LDV).

The aim of the study is to determine the effects of a disturbed flow at the inlet of a pump on its performance and to analyze the causes of these effects using CFD.

INTRODUCTION

It is well known that the upstream piping of a centrifugal pump has an influence on the operating conditions of the machine. These effects depend on the pump type (they usually increase with the specific speed) and on the installed configuration, which generates the flow disturbance at the pump inlet. Many investigations of the influence of disturbed inflows at different pumps were already done e.g. [3, 5], but it is hardly possible to analyze all combinations experimentally.

The manufacturer of pumps normally determines the characteristic curves of the pumps using a long straight pipe before the inlet of the pump, which guarantees optimal velocity inflow conditions. In practice this is often not possible to hold; bends, T-pieces and/or fittings like valves are installed sometimes just upstream the pump. A very common disturbance, which can often be found in water supply plants, is the combination of a 90° bend and a butterfly valve, like a study of 346 pump installations made by the German Technical and Scientific Organization on Gas and Water (DVGW) shows [4]. This study also testifies that most of the installed pumps belong to the radial multistage (37%), the single stage radial (28%) and the semi axial pump type (20%). Experimental investigations of these three pump types and numerical simulations for the semi axial pump were done at the Chair of Turbomachinery and Fluid Power (TFM).

The knowledge of the influences of a disturbed inflow could help the pump users to take considerations over the operating conditions and prevent possible damages of the machine.

Keywords: centrifugal pumps, disturbed inflow, CFD analysis.
The application of the CFD opens large possibilities in this field. In relatively short time and normally more economical than experimental investigations, the influences of various disturbances as well as other parameters, which affects the pump performance, can be analyzed.

The CFD applied in turbomachinery has experienced a great progress in the last years due to an improvement of the numerical methods as well as computer resources. The results of CFD simulations and experiments show good agreements and have made an important contribution to understand the flow within turbomachines.

This paper presents some interesting experimental results of the influence of disturbed inflow conditions on pump operation. The measurement data were compared with numerical simulations and it could be shown how CFD serves as a suitable method to gain more information of the flow and helps to analyze the causes of these influences.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>b.e.p.</td>
<td>best efficiency point</td>
</tr>
<tr>
<td>( c_m )</td>
<td>mean velocity</td>
</tr>
<tr>
<td>D</td>
<td>piping diameter</td>
</tr>
<tr>
<td>DN</td>
<td>nominal diameter</td>
</tr>
<tr>
<td>L</td>
<td>length of the straight link</td>
</tr>
<tr>
<td>( n_s )</td>
<td>specific speed</td>
</tr>
<tr>
<td>R</td>
<td>curvature radius of the bend</td>
</tr>
<tr>
<td>q</td>
<td>flow ratio ( V/V_{b.e.p.} )</td>
</tr>
</tbody>
</table>

TEST FACILITIES

The experimental investigations were carried out in a closed water test rig, which is equipped with all necessary components to control the operating point of the pump (figure 1). The tested pump is a single-stage, end-suction volute casing pump having a specific speed of \( n_s = 42.5 \text{ min}^{-1} \). It has an axial inlet with a flow straightener rib in the suction branch to limit the upstream extent of part load recirculation. The spiral casing of the pump is of a double volute type with a tangential discharge nozzle.

The selected test installation for generating a disturbance is schematically shown in the figure 2. Directly upstream of the pump inlet a short transparent pipe section is built in, which can be used for pressure measurements, optical analyses (by means of an endoscope), as well as the determination of the velocity distribution with the aid of a Laser Doppler Velocimeter.

![Figure 2: Test disturbance installation](image)

The length \( L \) of the intermediate distance can be varied by the installation of appropriate pipes. The configurations \( L = 0D, 2D \) and \( 5D \) are tested. Upstream the intermediate distance pipe is connected to a 90°-bend with \( R/D = 1.5 \) and a simple butterfly valve. All parts are standard components and commercially available. All fittings at the suction side have a nominal diameter of DN=100 mm.

The influence of the valve is investigated only in terms of a shut-off valve i.e., measurements are only done for the completely opened position of the butterfly valve. The angle of attack of the valve disc was 89° (fully opened). As an additional variation, the valve is installed with the disc horizontally (case “K”) or perpendicularly (case “S”) to the curvature plane of the bend.

![Figure 3: Disturbances: case “K” (left), case “S” (right)](image)

EXPERIMENTAL INVESTIGATION

In this section results of the experimental investigations are presented. The nomenclature for the different disturbances is given by the length of the distance pipe (e.g. \( 5D \rightarrow L = 5DN \)) and the orientation of the valve. If the valve is installed with the disc perpendicular to the plane of curvature of the bend, this is indicated by “S”, if it is installed with the disc in-plane with the bend curvature plane this is indicated by “K”.

![Diagram](image)
1) Characteristic curves.

The head (H) is calculated by means of the differential total pressure between pump inlet and outlet and the measured flowrate (V), the shaft power (Psh) by means of the torque (T) and the speed of rotation (n). Using equation (1) the pump efficiency can be calculated. The density \( \rho \) of the water is determined in the measurements considering the water temperature.

\[
\eta = \frac{\rho \cdot g \cdot H \cdot V}{T \cdot 2 \cdot \pi \cdot n}
\]  

(1)

The effects on the characteristic curves caused by the different disturbances are summarized in figure 4. The values are referred to the measurements carried out with the straight pipe for the flowrates \( q = 0.8, 1.0 \) and 1.2.

It is to recognize that all "K"-cases result in a head and a power loss. Differing from this behaviour, the perpendicularly installed valve ("S"-cases) leads to results showing a clear increase of power. In general each disturbance produces a decrease of efficiency; in the worst case (0DK) up to 1.4 % in the b.e.p. This tendency of changes is recognized also in other operating points, whereas the values are lower at partial load and higher at overload (\( \Delta \text{Head}_{\text{max}} = -1.6 \% \)).

![Figure 4: Changes of characteristic values](image)

From the above shown results it can be supposed that probably a swirl, acting in the direction of pump rotation, produces a drop of the shaft power and an incorrect incidence flow at the blade leading edges in the "K"-cases. An increase of shaft power as found for the "S"-cases seems to be caused by a swirl contrary to the pump rotation resulting from the combination of valve and bend.

2) LDV Measurements.

The velocity profiles at the inlet of the pump were measured with the non intrusive method of Laser Doppler Velocimetry. The equipment to measure the velocity components is mounted directly upstream of the suction branch of the pump (figure 5). It is possible to rotate the positioning device of the laser probe for 360° around the pipe axis in order to measure a complete profile of the inlet velocity.

![Figure 5: Laser measurements](image)

The axial and tangential velocities for the configuration with \( L = 0 \), that means without any pipe between bend and measuring window, is shown in figure 6. The measured velocities are related to the mass averaged value of the mean flow velocity \( (c_m = V / A) \).

In both cases the axial velocity shows a distortion with bigger magnitudes at the outside of the bend and smaller ones at the inside. The axial disturbance in both cases smoothes out after 5D. The combination with the valve installed in the curvature plane of the bend produces a strong non axisymmetrical swirl which does not disappear after a distance of 5D. When the valve is perpendicular to the bend curvature plane a smooth swirl contrary to the direction of pump rotation occurs which disappears almost completely after 5D. These measurements confirm the assumption that a swirl, rotating in the same direction as the pump impeller, produces the head and power loss, which could be determined for the "K"-cases.

![Figure 6: Axial velocity (top) / tangential velocity (bottom)](image)
NUMERICAL SIMULATIONS

The 3-D numerical simulations were carried out with the commercial CFD software Fluent 6.2.12. A 3-dimensional model of a bend and a butterfly valve was generated and meshed with the preprocessor Gambit. The fluid volume to calculate was modeled with hexahedral cells on the pipe and with non-structured cells in the valve region due to the complexity of the geometry. The inclination of the valve disc was 1° to the inner side of the bend in the case “S” and to the upper side of the pipe in the case “K”. At the inlet a turbulent full develop velocity profile was given as boundary condition and at the outlet there was set as static pressure condition. A long straight pipe was modeled at the outflow to avoid an influence from the boundary condition.

Figure 7 shows the modeled regions for the disturbance “S”. Planes have been plotted at the outflow of the bend to show the locations were the velocity profiles and the pressure were measured.

![Diagram](image)

Figure 7: Modeled fluid volume of the distortion.

The analyzed flow is a turbulent flow like the most flows in industrial applications. Therefore a study of various turbulence models has been performed. Turbulence is modeled using statistical approaches for the unknown Reynolds-stress-tensor in the Navier-Stokes equation. Models using two additional transport-equations for the turbulent quantities are the most common ones because they were tested in a wide range of applications and they are known to be a good compromise between precision and economy. The comparison parameters were the velocity measurements. The best agreements with regard to the velocity profiles were achieved by using “standard-kε” and “ko-σt” models. Due to the advantages of the “ko-σt” model, particularly for flow simulations within turbomachines showed in [1], this one was used for the presented simulations.

The most appropriate discretization scheme was the segregated implicit solver. The momentum, kinetic energy and dissipation rate equations are solved using a second-order-upwind-scheme; a standard-order-scheme is taken for the pressure correction and a SIMPLE-scheme for the pressure-velocity-coupling.

The convergence criteria of the calculation were based on the residuals and a mass-averaged velocity magnitude. For the first simulations the residuals falls under $10^{-4}$ only by setting the discretization on first order but the velocity field after the bend was calculated symmetric and didn’t agree with the LDV-measurements. Just by using second order discretization on the moment the tendency of the measurements could be confirmed, but with some convergence problems. Also results for a stationary calculation of the whole system (valve, bend and pump) were done and non satisfactory convergence criterions and accordingly velocity fields were obtained. Due to above mentioned experiences it could be supposed that for the existing flow configuration (butterfly valve directly upstream of a bend) no stationary solution can be achieved. Therefore it was decided to use the experimental measured velocity profiles as inlet condition for the pump.

The calculated velocity fields obtained by using the “ko-σt” model are shown in the figure 8.

![Images](image)

Figure 8: Calculated velocities: axial (top)/tang. (bottom)

Although the results were not satisfactory, the locations of the swirls determined by the LDV measurements could be validated. The velocity field downstream of the valve respectively upstream of the bend is shown in figure 9 (left). The figure shows a swirl, which is situated in the upper left sector of the pipe. It increases and develops into the swirl measured downstream of the bend showing a direction of rotation which is contrary to that of the pump impeller (figure 9 right).
One of the most important and time-consuming tasks in the process of a CFD-Simulation is the generation of the computational grid. The pipe and the impeller were created with hexahedral cells, the valve region, the inlet of the pump and the volute were meshed with tetrahedral cells, because of their complex geometries. The model was created without considering the impeller side spaces (i.e. only inlet, impeller and spiral casing) and had a grid size of approximately 900,000 cells (figure 11).

The model includes three fluid zones, one rotating (impeller) and two stationary (inlet zone and volute) ones. The interfaces between the regions are shown in figure 12 (right). The solving parameters and turbulence models used are the same as for the simulation of the valve and bend.

The measured velocity profiles for the seven disturbances variations were specified as boundary conditions at the inlet. At the outlet a static pressure condition was set. The wall parts of the rotating zone are treated as moving surfaces with a rotational speed of zero relative to the adjacent cell zone, which is rotating. Non-rotating walls, that are part of the rotating reference frame, are treated as moving walls with a rotational speed of zero in the absolute reference frame.

The CFD-simulation of real turbomachinery flows can be realised using time-dependent numerical methods, i.e. the rotation of the impeller relative to non-moving parts of the machine is modeled. Although enormous computer resources are needed for this type of modeling, it simulates real flow physics best of all, while steady or quasi-steady numerical approaches only approximate the real flow, because they neglect important effects of real physics. Nevertheless the steady models are quite popular in industrial use, mainly due to the relatively short computing time compared to transient calculations, which is needed to obtain numerical results. In this paper, a steady CFD-model for turbomachinery, widely known as "frozen-rotor"-model and transient simulation with the sliding mesh model, are used for the calculations.
The "frozen rotor" method considers only the movement of the fluid in the impeller, which means that the position of the impeller does not move relative to the volute. The "sliding mesh" method calculates for each time step a new position between impeller and stationary parts and therefore better reproduces the physics of the flow in the pump.

For the first method the flowrates of $q = 0.8, 1.0$ and $1.2$ were calculated. Therewith it was possible to reproduce a characteristic curve for the pump and compare it with the experimental measured one. In figure 13 the measurements as well as the simulation using the frozen rotor coupling method are shown for the case with straight pipe. A good agreement can be observed in the head curves. The power calculated is lower, due to the difference between the shaft moment measured in the pump and the hydraulic moment calculated by CFD. In the simulation the moment is determined integrating the static pressure across the impeller inner surfaces (without considering the hydraulic friction losses within the impeller side spaces) whereas in the experiments it is directly measured by use of a torque meter, which also includes the mechanical friction of the bearing and seals as well as the hydraulic friction within the impeller side spaces. This explains in combination with the negligence of the internal leakage flow the difference of approx. 6 % points in the efficiency curves.

Figure 13: Characteristic curves (measured and simulated)

The calculated head, power and pump performance affected by the different velocity profiles (relative to the reference case straight pipe) are shown in figure 14. These results can be compared with the measured values shown in figure 4.

In general the tendencies of the calculated values coincide very well with the measurements. The "K" cases of the measurement data show a little higher head drop than the calculations. The calculated power increases for the "S" cases are lower than the measured values. The numerical obtained efficiency ratios are approx. 0.5 to 1 % too high compared with the corresponding measurement values. This means that the calculated influence of the efficiency is very small compared with the measured one.

Figure 14: Simulated influence of the disturbance

The results of the instaneously simulations (sliding mesh) are shown in the figures 15 and 16. These simulations were done only for the b.e.p. ($q = 1.0$). The time step is chosen to have a calculation every 3°. In figure 14 the time dependent values for the head, moment and efficiency relative to the angle of rotation of the impeller are shown. Lower values for head and power can be seen for the "K" cases.

Figure 15: Characteristic curves (time depended simulation)
The mean values of the characteristic curves are made across 180°, i.e. three blades past the volute tongue. Also here the drop of head and power can be observed for the “K” cases. These values do not significantly vary from the values calculated with the “frozen rotor” method. The efficiency differences are approx. 0.2 % points compared with the results obtained with the “frozen rotor” coupling method. Compared with the measurements the efficiency differences are approx. 0.5 to 1 % higher.

![Graphs showing the influence of disturbances on head, power, and efficiency ratios.]

Figure 16: Influences of disturbances, b.e.p.

The head, power and efficiency ratios calculated from the simulation results with and without inlet disturbance are shown in figure 16 for the best efficiency point. Do to comparison the figure also presents the results of “frozen rotor” computations as well as the measurements.

SUMMARY AND OUTLOOK

A comparison between experimental results and CFD calculations with the purpose to analyze the influence of disturbed inflow condition on the performance of a centrifugal pump where shown. Although the investigated inlet disturbances caused only relatively small effects on the efficiency of the used pump (max. 1.4 % in b.e.p.), it has to be taken into account, that even such slight deteriorations can lead to remarkably higher operating costs particularly in respect to the live time of the pump installation.

To have a better understanding of the importance of the inflow conditions of a pump respectively its influences on the pump performance, the velocity profiles were measured directly upstream at the inlet of a centrifugal pump using the non-intrusive Laser Doppler Velocimetric (LDV). To analyze also the flow in regions where it is difficult or impossible to perform measurements CFD simulations were carried out. Due to the gained flow field information (experimental and numerical) flow distortions and swirl generation, which influences the pump performance, could be explained.

In the numerical investigations the whole system, including impeller and volute has been considered. The CFD results could reproduce the tendencies of the measurements and of the effects observed in the characteristics of the pump, whereas not all influences on power and efficiency could be predicted numerically. The different coupling methods used commonly in the CFD approaches do not affect significantly the results of the numerical investigation presented in this paper.

It could be shown that the mounting position of a butterfly valve relative to a bend should be taken into account since in several analyses the cases where the valve was installed with the disc in-plane to the bend’s curvature plane caused a significant deterioration of the pump characteristics. The development of this distortion could be explained with the CFD simulations.

The effect of a disturbed inflow in general diminishes with an increasing distance of the elements, which generate the disturbance, to the pump.

Instationary calculations for the whole system including valve, bend and pump will be made to study if a better agreement with the measured velocity profiles could be achieved.

REFERENCES


