Prediction of the dynamic forces acting on a shaft

Simulation of the flow in a pump sump

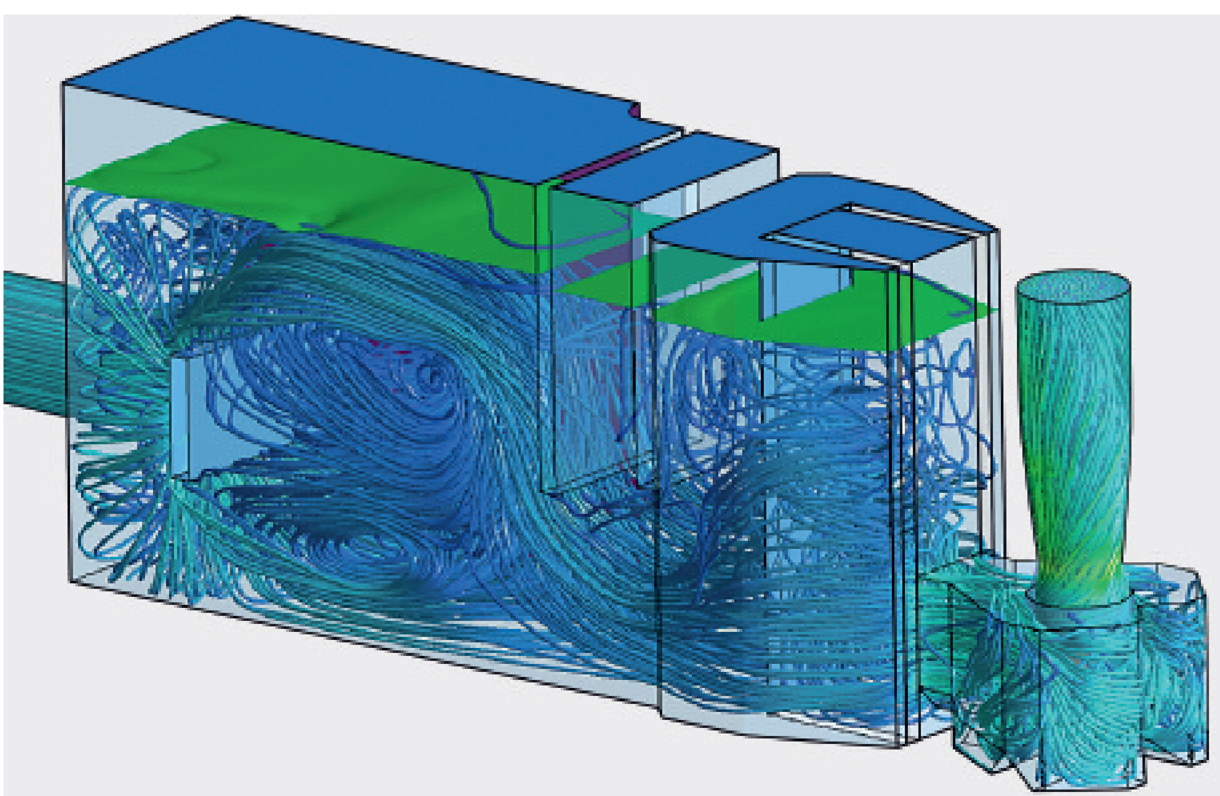
By means of flow simulation, engineers from Sulzer Pumps and Sulzer Innotec have been able to show that a modified pump sump geometry can lead to significantly smoother and uniform inflow at the pump inlet. The radial forces acting on the shaft can therefore be reduced and mainly centered around the zero point, allowing a significant increase in the lifetime of the bearings.

Computational fluid dynamics (CFD) has been successfully used for many years in the development and optimization of pumps. CFD is widely used for the prediction of the pump head, power, and efficiency of pump stages. However, exact knowledge of the inflow conditions is necessary in order to be able to correctly calculate these quantities. It is therefore also of interest to correctly predict through CFD the flow conditions in the sump, from which the vertical pump draws in the water.

The continuous development of numerical models, a steady increase in the available computing power, and careful validation based on experimental data with clearly defined boundary conditions are important factors for the successful application of CFD. The latter is necessary in order to be able to reliably design pumps and their hydraulic components through numerical flow simulations.

Calm flow for smooth pump operation

Large vertical pumps, such as those used for water supply, for process plants, and in cooling processes of thermal power



1 Pump sump with a vertical pump: view of the water surface (green) and the streamlines, colored by velocity (increasing velocity: blue, green, red).

plants, are submerged in a water basin, the so-called pump sump, where they draw in water 1. These semi-axial flow pumps must be reliable for a wide range of operating conditions, from low partload to overload. The behavior of these vertical pumps is strongly influenced by the flow conditions at the pump inlet.

The formation of vortices and high-velocity gradients at the inlet of the pump can negatively affect the performance and may lead to shaft vibrations due to the dynamic radial forces. These vibrations can damage the shaft and the bearings and, in the worst case, lead to the failure of these components. Therefore, the industrial standards of the Hydraulic Institute[1] define the vortex structures and velocity gradients that are permissible at the pump inlet.

Up to now, model tests have been performed to demonstrate to the customer that no impermissible flow phenomena occur upstream of the pump. As these tests are very time consuming and expensive, it is advantageous to complement or replace these experimental tests by numerical flow simulations. Often, different pump sump configurations have to be investigated before a favorable flow into the pump can be achieved. The investigations of these variants can be carried out much more efficiently using numerical simulations, which reduce development time and costs.

Modeling of the flow conditions for a pump sump on the computer

In order to assess the possibilities and limits of numerical flow simulation, the operational behavior of a typical pump sump with a vertical pump has been modeled on the computer. This numerical model has been compared with the results from experiments using a physical model. Engineers have analyzed two pump sump designs to ensure that the numerical methods also provide the correct forecasts under different flow conditions 2:

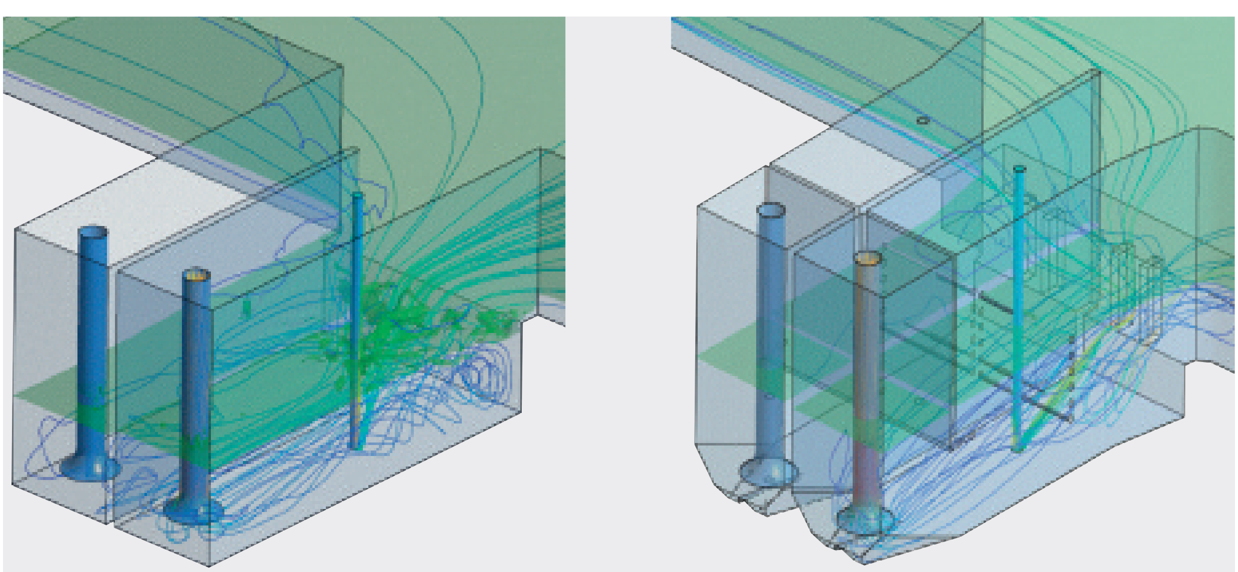
- The basic geometry, in which the vortices that occur are impermissible according to the industrial standards.
- Improved geometry, in which no vortices occur or only such vortices occur that are permissible according to the industrial standards.

The occurrence of these vortex structures should be demonstrated by means of unsteady flow simulations.

The real flow within a pump sump includes the free surface of the water that may also be deformed by velocity gradients or by the occurrence of vortices. The modeling of this free surface by means of CFD is possible but is more complex and thereby requires more computer power. This additional effort is required because a two-phase simulation has to be set up that takes the liquid and the gas phases into account, thus, also, the air above the water.

In some cases, simpler models can also reliably provide the desired information. Therefore, the free water surface is replaced by a fixed one. However, this simplification means that deformations at the water surface can no longer be reproduced.

In addition, to keep the computational costs as low as possible, engineers do not model the impeller of the pump—only the outer and inner contours of the pump housing are taken into account. It is assumed that the impeller has no



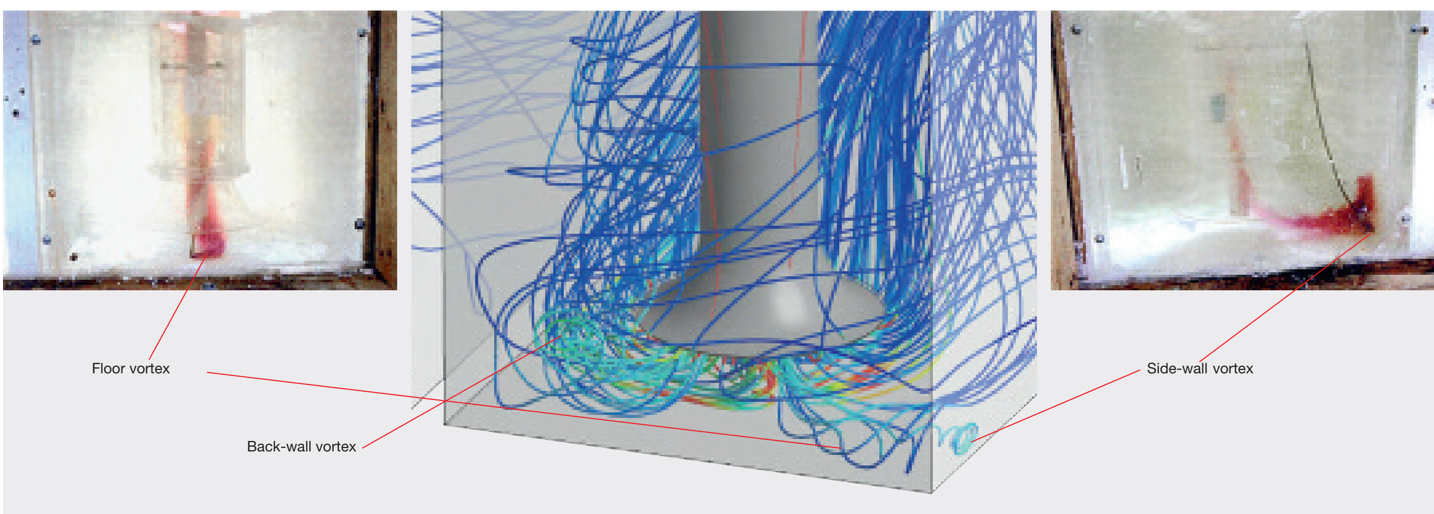
2 Inflow to the vertical pumps. On the left, the original sump geometry with distinct vortex structures; on the right, the modified sump geometry with uniform flow.

significant influence on the flow in the pump sump.

Figure 3 compares simulation and experiment for the basic geometry. Five vortex structures can be seen in the model experiments. Two of these can be seen in the photo from the experiments, and they are also indicated in the streamlines from the simulation: the "floor vortex" and the "side-wall vortex". The vortices are submerged and permanent (type 2 according HI). The more complex simulation of this geometry with a free surface could predict four of these. With the simpler simulation using a fixed water surface, on the other hand, it was not possible to clearly identify the intermittent vortex described as the "back-wall vortex".

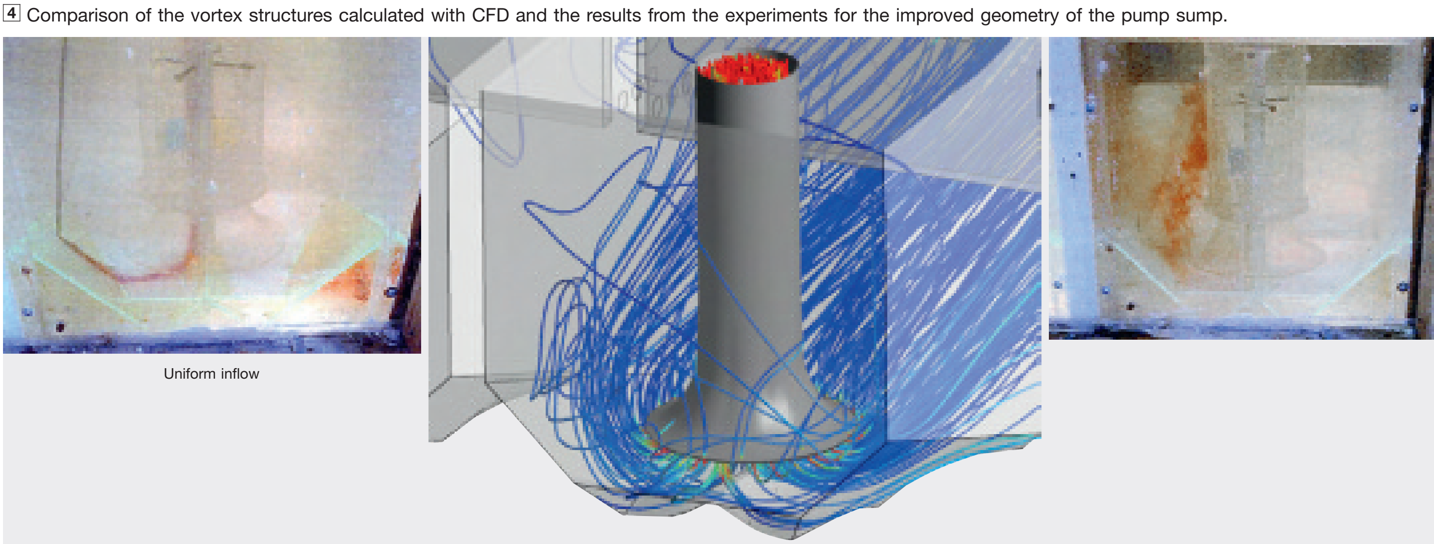
The presentation of the streamlines for the case with the modified geometry 4 indicates a much more uniform flow, which is also confirmed by the pictures from the experiments. As already mentioned, no significant vortex structures arise with this configuration. This behavior can be predicted with both types of CFD modeling—with and without free surface of the water.

The results of these simulations show that the numerical models are able to recognize the different types of vortices



3 Comparison of the vortex structures calculated with CFD and results from the experiments for the basic geometry of the pump sump.

4 Comparison of the vortex structures calculated with CFD and the results from the experiments for the improved geometry of the pump sump.



as defined by the Hydraulic Institute[1]. The impact of the modification of the geometry of the sump on the vortex formation is also reflected.

The validation shows that, in the case of the basic geometry where impermissible vortices structures occur, the CFD modeling with free surface predicts these better than the simpler method with fixed water surface. However, these slightly more accurate results require a significantly larger computational effort. For this reason, it would be sufficient in most cases in industrial practice to carry out the simulation with the more efficient method with fixed surface.

Determination of the dynamic forces acting on the shaft

The impact of pump inflow on radial forces acting on shaft and bearings is also of interest in pump design. In order to investigate this, engineers must carry out a transient simulation of the vertical pump with the impeller.

The computational domain only includes the pump without the sump and begins at the pump inlet. In order to obtain the correct inflow conditions into the impeller in the simulation, engineers use the inlet velocity profile from a sump simulation. This information is obtained as described above. For comparison, a uniform inlet velocity profile is also used.

The forces can be determined by adding the pressures acting on the rotating parts of the pump. The spatial distribution of the force components F_x and F_y is plotted for one impeller revolution. The resulting forces on the shaft during operation with a uniform velocity profile at the inlet are shown in figure 5.

It can be seen that the forces are largely centered around the origin, the center point of the shaft. The case with an inflow with a non-uniform velocity profile from a sump simulation is shown in 6. The resulting forces are, in this case, no longer located around the center point of the shaft. The forces are increased by around a factor of two against the main flow direction. This means that there is a strong dynamic load on the shaft, which may lead to its failure or to damage at the bearings.

Fewer model tests thanks to state-of-the-art simulation methods

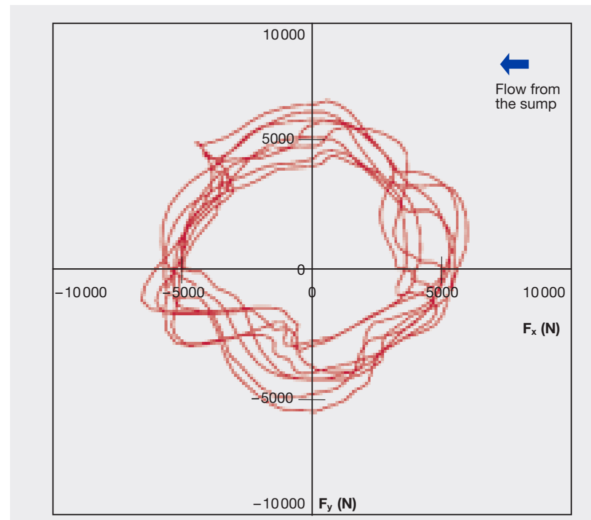
Thanks to state-of-the-art simulation methods, it is now possible to considerably reduce the number of expensive physical model tests. Numerical flow simulation is able to predict the vortex structures and the velocity gradients that are relevant for the assessment of sump pumps according to industrial standards.

If the inflow profile at the pump inlet is transferred into a simulation for the impeller, the radial forces that act on the shaft can be determined. Coupled flow calculations (sump and pump internals) have shown even larger forces. It is thus possible to determine in the design phase whether any impermissible dynamic loads act on the shaft and, thereby, on the bearings.

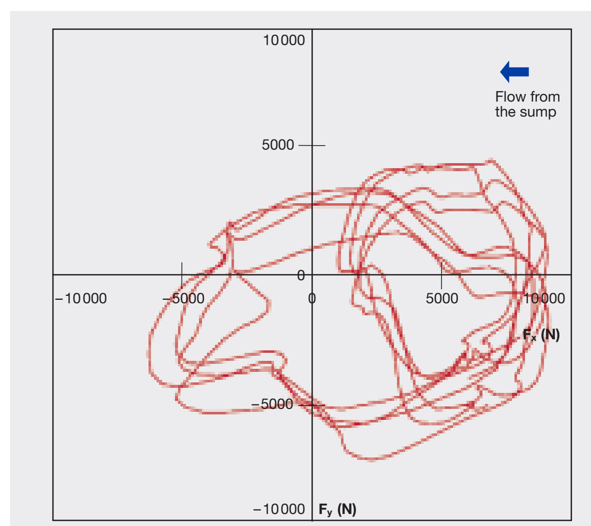
The precondition for reliable predictions from numerical flow simulations is the thorough validation with experimental data, as well as exact knowledge of the limits of these methods.

Felix A. Muggli
Sulzer Markets & Technology Ltd.
Sulzer Innotec
Sulzer Allee 25
8404 Winterthur
Switzerland
Phone +41 52 262 42 52
felix.muggli@sulzer.com

Susanne Krüger
Sulzer Pumps Ltd.
Zürcherstrasse 12
8401 Winterthur
Switzerland
Phone +41 52 262 40 10
susanne.krueger@sulzer.com



5 Spatial distribution of the radial forces acting on the impeller during an impeller revolution as calculated with CFD for a uniform velocity profile at the pump inlet.



6 Spatial distribution of the radial forces acting on the impeller during an impeller revolution as calculated with CFD for a non-uniform inlet velocity profile from a sump simulation.

References

[1] ANSI. *Pump Intake Design Standards* (ANSI/HI 9.8). Parsippany, New Jersey: Hydraulic Institute, 1998.