Technical Paper

CFD Technology

Session 6-2

CFD Analysis of Sump Flow and its Impact on the Hydraulic Forces Acting on the Impeller of a Vertical Pump

Author:

Dr. Philippe Dupont
Manager Hydraulic Development
Sulzer Pumps Ltd
CH-8401 Winterthur, Switzerland

Co-Author 1:

Dr. Youcef Aït Bouziad
Development engineer
Sulzer Innotec
CH-8404 Winterthur, Switzerland

Co-Author 2:

Mike Cugal
Hydraulic engineer
Sulzer Pumps Ltd
Brookshire, USA

Summary

The use of commercial Reynolds-Average Navier-Stokes (RANS) codes has become a standard in modern hydraulic design. This is not only true for the hydraulic design of modern pumps but has recently been more and more frequently used for the analysis of sump flow and its impact on the behaviour of the pump.

The aim of this article is to show how CFD (Computational Fluid Dynamics) can be used in the design of modern vertical pumps, for the analysis of sump designs and, in particular, for understanding the possible impact of sump design on the behaviour of these pumps.

The first part of the article presents the numerical prediction of the hydraulic performance, in the range between minimum and maximum flow, of a semi-axial turbine pump for ideal suction conditions and compares them with experimental test results obtained in a closed loop. It shows the ability of modern RANS CFD codes to accurately predict pump performance if the pump inlet flow conditions are sufficiently well known.

The second part of the article presents the numerical method used for the calculation of the flow in sumps and compares the numerical results with model test results for an original and an improved sump design. This comparison shows the CFD approach to be able to reproduce very well the flow characteristics observed, in particular the formation of vortices within the sump, and to be a very useful tool for analyzing and eventually improving a given sump design.
In the third and final part of the article, the two presented numerical methods are combined, on the one hand, in order to analyze the flow for a second sump design having quite extreme characteristics, and on the other hand, in order to assess the possible impact of the suction conditions induced by the sump on the behaviour of the pump. Indeed, recurrent bearing failures were observed for pumps used in the analyzed sump. For this analysis, the flow at the inlet of the bellmouth of the vertical pump resulting from the sump CFD calculation is imposed as a boundary condition for the calculation of the flow within the pump. This approach allows the analysis of the impact of sump design on the performance of the pump and, in this particular case, on the hydraulic forces acting on the impeller of the pump. These calculations show that the bearing failures are due to the flow induced by sump design.