



Technical Paper

Cleanability and careful conveying

Session No. 9-1

Validation of Velocity Distributions in Hygienic Designed Multistage Centrifugal Pumps

Author:

Dipl.-Ing. Heiko Kipp
Research assistant, Institute for Fluid Machinery and Fluid Mechanics
Department of Mechanical and Process Engineering
Technical University of Kaiserslautern
67663 Kaiserslautern, Germany

Co-Author 1:

Prof. Dr.-Ing. D.-H. Hellmann
Formerly Head of Institute for Fluid Machinery and Fluid Mechanics
Department of Mechanical and Process Engineering, TU Kaiserslautern
67663 Kaiserslautern, Germany

Co-Author 2:

Prof. Dr.-Ing. E. Schlücker
Scientific counsellor of the Berdelle-Hilge Foundation
Head of Institute for Process Technology and Machinery
Faculty of Engineering Science, University of Erlangen-Nuremberg
91058 Erlangen, Germany



Summary

To estimate the potential of a reliable cleanability caused by fluid mechanical effects in closed processing equipments the applicability of Computational Fluid Dynamics (CFD) is examined and evaluated on a multistage centrifugal pump. The aim of these investigations is the use of CFD as a virtual tool to verify the hygienic design according to the relevant CIP (Clean In Place) cleanability tests of the European Hygienic Engineering and Design Group (EHEDG). Thereby, the CFD model of the investigated three-stage centrifugal pump includes all wet parts in detail. Three pump stages are examined in order to test the virtual cleanability of this type of pump. For the CFD simulations the SST- $k-\omega$ -model with the standard wall functions are used. Due to unsteady flow interactions the simulation for velocity validations has to be transient. Therefore the impeller-stator interaction is modeled by the use of the sliding mesh method. To validate the velocity distributions inside the pump, the CFD and the experimental measurements are arranged with air driven pump tests. The presented work is dealing with the validation of the velocity distributions. Aim of this work is the estimation of possible numerical velocity deviations caused by the used turbulence model with regard to further numerical wall shear stress proposals. The operating point for experiment and simulation is the EHEDG-relevant off-design condition of the pump. Flow velocities near to the outlet of the impeller are measured with hot-wire anemometry in each stage. A comparison of the steady-state numerical and experimental characteristic curves shows the transferability of the results from water to air-driven measurements and simulations. Furthermore the velocity profiles predicted by CFD are in good agreement with the real pump flow behavior.