ISROMAC 2017
International Symposium on Transport Phenomena and Dynamics of Rotating Machinery
Maui, Hawaii
December 16-21, 2017

[Extended Abstract]

Investigations of different blade angle distributions for axial and mixed flow inducer operating with high rotational speed

Björn Gwiasda, Institute of Fluid Mechanics and Fluid Machinery, Department of Mechanical and Process Engineering, Technical University of Kaiserslautern, Kaiserslautern, Germany

Matthias Mohr, Institute of Fluid Mechanics and Fluid Machinery, Department of Mechanical and Process Engineering, Technical University of Kaiserslautern, Kaiserslautern, Germany

Martin Böhle, Institute of Fluid Mechanics and Fluid Machinery, Department of Mechanical and Process Engineering, Technical University of Kaiserslautern, Kaiserslautern, Germany

Introduction

Inducers are often necessary to avoid cavitation at the inlet of a pump impeller. Especially for pumps not acting under common conditions, like pumps for rocket engines. These kind of pumps have to provide a high mass flow and high pressure rise with a minimum need of weight and space. For this task high rotational speed is required. Even though inducers are employed in many application there is still no well-established and reliable design practice available in the open literature for inducers. Furthermore the influence of the geometrical design parameters on the performance can often only be seen in a late phase of the design procedure with the help of 3D-CFD tools. These results must be validated with experiments. In case of examining cavitation behavior, 3D-CFD simulation is very time-consuming and fraught with uncertainty, so that experiments are mandatory. Although experimental data for a lot of different inducers has been published in the past, usually only few details about the geometrical design parameters are included in these reports. This makes judging of the influence and the interaction of important design parameters like hub to tip ratio and blade angle difficult.

1. Methods

Especially for suction performance tests the investigation at the actual design speed is of huge interest because scaling of cavitation effects is still an unresolved problem. Because of this circumstance a new test facility was developed at the Institute of Fluid Mechanics and Fluid Machinery at the Technical University Kaiserslautern (SAM). This test bench allows to perform experiments with a rotational speed of up to 40000 rpm and high pressure ranges with water as working fluid. Suction performance can be investigated by adjusting the pressure at the inlet of the inducer. For this purpose the test rig is also equipped with a degassing and dissolved oxygen monitoring system. An optical access allows visual analysis of different cavitation phases while measure the characteristic and NPSH curves. With these experiments the influence of different design parameters on performance and cavitation behavior can be examined. The CFD simulations can be validated and further design modifications can be made faster with reliable simulation procedures. Also the results act as a recommendation for further design procedures, especially for inducer operating under sophisticated conditions. In a first step axial

and mixed flow inducer with an identical inlet blade design a constant blade angle distribution are examined experimentally. Simulations show that with this constant distribution along the contour the non-cavitating performance of the mixed flow inducer drops remarkably. Therefore the inducer with the mixed flow hub shape is modified keeping the inlet design untouched but varying the blade angle distribution towards the trailing edge. Two variants with different exit angles are manufactured and tested. For the variant with best results regarding performance and cavitation behavior the influence of tip clearance will also be examined. All experiments are performed for different inlet pressures and different rotating speeds of up to 24000 rpm. In this paper a detailed overview of the new test facility for tests with high rotational speed is given. The influence of chosen design parameters on performance and cavitation behavior is shown. The experimental results are compared with 3D-CFD simulations so the accuracy of the simulation can be assessed. For all characteristic and NPSH curves the flow conditions are visualized to estimate the influence of cavitation.