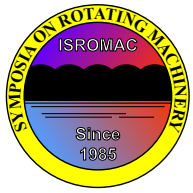
Study of the Losses in Fluid Machinery with the Help of Entropy

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

Annika Fleder, Institute of Fluid Mechanics and Fluid Machinery, Department of Mechanical Engineering, Technical University of Kaiserslautern, Germany

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



Long Abstract

Abstract

In the current design process of turbomachinery the use of CFD methods is state of the art. With CFD calculations we are able to predict the performance characteristics of the machines. In most of the cases the machines are rated based on their performance alone. A detailed evaluation of the losses involved often is not conducted, mainly because there is no established method to analyze the location and mechanisms of loss production.
This leads to the fact that the loss mechanisms for some turbo machines are not known in detail, although they are very important to completely understand the operation principle of the machines and have a huge influence on their performance. If the location and quantity of losses were known it would be possible to perform a selective optimization of the machine. The question is how to calculate and localize the losses and evaluate them with CFD.
In his thesis Kock introduces a method, to identify losses in CFD. His method is based on the entropy generation rates in turbulent flow fields. In this paper the method is applied on hydraulic fluid machinery and two different machine types are discussed in detail. Those are a side channel pump and an inducer. The locations of the losses are detected and the size of losses will be evaluated by applying the method of Kock.

Introduction

In all fluid machinery there exist different kind of losses. We have to differentiate between inner losses and outer losses. The outer losses are, for example, losses in bearings, gears and shaft seals. Those losses are not included in the CFD calculation whereas the inner losses are considered by CFD. Among the latter are shock losses and friction losses. The different types of inner losses can be summarized in dissipation.
The numerical simulation offers the opportunity to calculate the flow in the machines spatially and temporally dependent. Thus the flow in the machines can be observed three dimensionally. The calculated quantities are the fields of pressure, temperature and velocity. Derived quantities like head coefficient and efficiency can be calculated directly during post processing. To identify also the different inner losses Herwig and Kock [1], [2], [3] proposed a direct method based on the evaluation of entropy production in turbulent flow fields. The advantage of the method is that it can be applied to a CFD simulation in a pure post processing step. The content of the present paper is the application of this method to two examples of hydraulic turbomachinery, a side channel pump and an inducer. For those two types of machines an analysis of losses is performed and the conservation of powers is demonstrated.

References

[1] F. Kock and H. Herwig. Local entropy production in turbulent shear flows: a high-Reynolds number model with wall functions. *International Journal of Heat and Mass Transfer*, 47:2205–2215, 2004.

[2] F. Kock and H. Herwig. Entropy production calculation for turbulent shear flows and their implementation in cfd codes. *International Journal of Heat and Fluid Flow*, 26:672–680, 2005.

[3] H. Herwig and F. Kock. Direct and indirect methods of calculating entropy generation rates in turbulent convective heat transfer problems. *International Journal of Heat and Mass Transfer*, 43:207–125, 2007.