Investigation of the effects of runner gap width on the flow field in the draft tube

Bernd Junginger, Stefan Riedelbauch, Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Germany

Long Abstract

Introduction

Draft tubes play a substantial role for the performance of low head turbines. In the turbine design process the geometry of axial machines is often simplified. This means that gaps are neglected for the design process. Due to these simplifications the draft tube flow can be falsified, in particular in off-design operation points. For the investigated axial propeller turbine a numerical comparison of several different gap sizes and turbulence models is carried out. For the investigated operating point a full load vortex in the draft tube develops.

For the numerical study a 4-blade propeller turbine is investigated. The gap size between runner and shroud is varied to analyze the effects on the overall flow field and especially on the draft tube flow in order to get a better understanding of the developing flow phenomena. Therefore the different numerical approaches, meaning different turbulence models are evaluated against experimental measurements. In this paper the k-ω-SST model and the SAS-SST turbulence model are applied.
An evaluation of the integral quantities, torque, head and discharge compares numerical results with experimental data. The experimental measurements follow the standard of the IEC 60193 and were executed in the laboratory of the Institute of Fluid Mechanics and Hydraulic Machinery in Stuttgart [1].

An evaluation of the turbulent structures for the investigated turbulence models is performed, showing that a higher class turbulence model, like the SAS-SST model are capable to resolve the turbulent structures in more detail [2]. The vortical structures in the draft tube are illustrated for a mesh with about 30 million elements, see Figure 1. In this grid about 14 million elements are placed in the draft tube. The effects of the tip clearance and vortex in the draft tube are moreover evaluated by velocity profiles in cone and diffusor of the draft tube. Therefore the meridional, circumferential and radial velocity components are time-averaged and compared for the different numerical setups. The shape of the vortex, which develops in the draft tube, is furthermore compared with the results of the different numerical approaches. Finally the pressure signal at selected positions in the draft tube cone is investigated to identify dominant frequencies of the vortex structures in the draft tube.

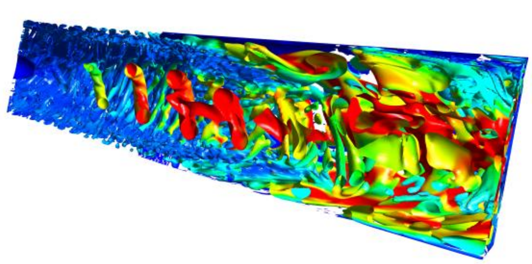

Figure 1: Isosurface of the velocity invariant Q=1 colored by viscosity ratio 0-1000 for a mesh with about 30 million elements

1. Numerical setup and turbulence modelling

The flow simulation of the turbine is carried out using the commercial CFD code Ansys CFX version 16.0. The vortical structure in the draft tube can be symmetric or asymmetric [3]. Thus, a full transient simulation for all machine components is performed. The numerical model for the CFD simulation is built up consisting of following parts: inflow, guide vanes, runner and draft tube with expansion tank. The modelling of the draft tube with expansion tank is similar to the test rig set up. The numerical model is in the size of the test rig, installed in the laboratory of the Institute. For all transient simulation the geometry is not simplified. This means, all existing gaps of the experiment are represented in the geometry of the simulation.

Two different turbulence models are compared. On one hand the k-ω-SST turbulence model is used which represents the standard in turbo machinery. On the other hand the SAS-SST turbulence model is used for comparison purposes. The k-ω-SST model is a classical RANS model whereas the SAS-SST model is a hybrid RANS-LES turbulence model. The k-ω-SST model by Menter which was applied for selected gap sizes for the transient analyses. In contrary to the k-ω-SST model, the SAS-SST model provides an approach to operate in Scale Resolution Simulation mode (SRS) [4]. Therefore, an additional source term is introduced in the transport equation for the turbulent eddy frequency of the k-ω-SST model [5, 6, 7]. The SAS-SST model needs a sufficient fine mesh resolution and time step. Otherwise the operation of the turbulence model is switched to RANS mode, which is an advantage over other hybrid RANS-LES models like turbulence models of the DES-type [8].

References

[1] International Electrical Commission (editor): International standard IEC 60193 Second Edition 1999-11. *Hydraulic turbines, storage pumps and pump-turbines – Model Acceptance Tests*, Geneva, Switzerland, 1999

[2] D. Jost and A. Skerlavaj. Efficiency prediction for low head bulb turbines with SAS SST and zonal LES turbulence models. *27th IAHR Symposium on Hydraulic Machinery and Systems*, Montreal, Canada, 2014.

[3] O. Kirschner. Experimentelle Untersuchung des Wirbelzopfes im geraden Saugrohr einer Modellpumpturbine, Dissertation, *IHS-Mitteilung 32*, University of Stuttgart, Germany, 2011.

[4] F. R. Menter. Improved two-equation k-ω Turbulence Models for Aerodynamic Flows. *NASA Technical Memorandum*,103975, California, USA, 1992.

[5] Y. Egorov and F. R. Menter. Development and Application of SST-SAS Turbulence Model in the DESIDER Project. *Advances in Hybrid RANS-LES Modelling, Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, 97:261–270, 2008.

[6] Y. Egorov, F. R. Menter and D. Cokljat. The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 2: Applications to Aerodynamic Flows. *Journal Flow Turbulence and Combustion*, 85(1):139-165, 2010.

[7] F. R. Menter and Y. Egorov. The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 1: Theory and Model Description. *Journal Flow Turbulence and Combustion*, 85(1):113-138, 2010.

[8] M. L. Shur, P. R. Spalart, M. K. Strelets and A. K. Travin. A hybrid RANS-LES approach with delayed-DES and wall modelled LES capabilities. *International Journal of Heat and Fluid Flow*, 29:1638-1649, 2008