Numerical investigation of the swirling flow and the vortex control in a straight diffuser

Dipl.-Ing. Ivana Buntic-Ogor, buntic@ihs.uni-stuttgart.de **Dipl.-Ing. Timo Krappel**, timo.krappel@ihs.uni-stuttgart.de **Dipl.-Ing. Oliver Kirschner**, oliver.kirschner@ihs.uni-stuttgart.de **Dr. -Ing. Albert Ruprecht**, albert.ruprecht@ihs.uni-stuttgart.de	University of Stuttgart, Institute of Fluid Mechanics and Hydraulic Machinery, D-70550, Stuttgart, Germany

Abstract

This paper presents a numerical investigation of the swirling flow in a straight conical diffuser accompanied with the existence of the unsteady vortex rope, as well as vortex control by means of water injection.

Unsteady vortex rope as a physical phenomenon is nowadays very common in hydropower plants due to increased need to operate at a wide variety of off-design conditions. It especially occurs in those plants running Francis turbines. Rotating vortex rope creates strong pressure fluctuations which can have an excessive impact on the complete plant's hydraulic system. Thus, investigation of this problematic phenomenon also has industrial importance.

For the purpose of the vortex rope study, an experimental test rig representing simplified conical diffuser is constructed and installed in the laboratory of the Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart (Figure 1). The swirl generator with adjustable guide vanes is attached upstream of the diffuser with the aim to produce and investigate different swirl conditions.

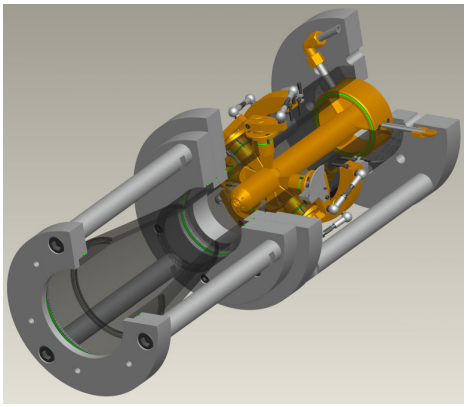

Figure 1: Swirl generator with conical diffuser

Our numerical study is performed on a model of straight diffuser mentioned before. The range of operational points proposed by [1] is used. A diffuser model with and without swirl generator is considered. Furthermore different RANS, VLES and RSM turbulence models are investigated regarding their ability to correctly predict unsteady swirling flow and arising vortex rope (Figure 2). Analysis of the concept to inject water jet axially in the center of the diffuser, as a way to control and prevent rotating vortex, is also carried out.

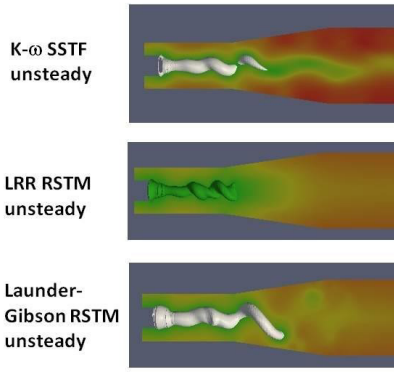

Figure 2: Simulation of vortex rope – applied are different turbulence models

All simulations are performed using OpenFOAM and the results are compared with numerous measurements which have been performed and presented in [2], [3]. By comparing CFD results with experimental data we would like to validate our numerical approach and gain the ability to more accurately elaborate problem of unsteady vortices using numerical computations.

References

[1] J.Grupp, E. Ohlberg and O.Kirschner: *Numerical investigation of vortex control in a straight diffuser*, 4th Romanian-German Workshop on Turbomachinery Hydrodynamics, University of Stuttgart, Stuttgart, Germany, June 11-16, 2008

[2] A.Ruprecht, J.Grupp, A. Al-Salaymeh and O.Kirschner: *Experimental and numerical investigation of vortex control in a simplified straight draft tube model*, IAHR 24th Symposium on Hydraulic Machinery and Systems, Foz do Iguassu, Brazil, October 27-31, 2008

[3] O.Kirschner, J. Grupp and H.Schmidt: *Experimental investigation of vortex control in a simplified straight draft tube model*, 4th Romanian-German Workshop on Turbomachinery Hydrodynamics, University of Stuttgart, Stuttgart, Germany, June 11-16, 2008