A swirl generator case study for OpenFOAM

2010 IOP Conf. Ser.: Earth Environ. Sci. 12 012056

(http://iopscience.iop.org/1755-1315/12/1/012056)

IOP Conf. Series: Earth and Environmental Science **12** (2010) 012056 doi:10.1088/1755-1315/12/1/012056

A swirl generator case study for OpenFOAM

O Petit[1], A I Bosioc[2], H Nilsson[1], S Muntean[3] and R F Susan-Resiga[2]

[1]Division of Fluid Mechanics, Chalmers University of Technology Hörsalsvägen 7A, SE-41296 Göteborg, Sweden
[2]Department of Hydraulic Machinery, "Politehnica" University of Timisoara Bv. Mihai Viteazu 1, Timisoara 300222, Romania
[3]Centre of Advanced Research in Engineering Sciences, Romanian Academy – Timişoara Branch Bv. Mihai Viteazu 24, RO-300223, Timişoara, Romania

E-mail: olivierp@chalmers.se

Abstract. This work presents numerical results, using OpenFOAM, of the flow in the swirl flow generator test rig developed at Politehnica University of Timisoara, Romania. The work shows results computed by solving the unsteady Reynolds Averaged Navier Stokes equations. The unsteady method couples the rotating and stationary parts using a sliding grid interface based on a GGI formulation. Turbulence is modeled using the standard k-ε model, and block structured wall function ICEM-Hexa meshes are used. The numerical results are validated against experimental LDV results, and against designed velocity profiles. The investigation shows that OpenFOAM gives results that are comparable to the experimental and designed profiles. This case study was presented at the 5th OpenFOAM Workshop, held in Gothenburg, Sweden, as a tutorial on how to treat turbomachinery applications in OpenFOAM.

1. Introduction

Nowadays, due to the variable demand of the energy market and new intermittent energy sources, a new parameter is often important for water power: flexibility. Water turbines now operate over an extended range of regimes that can be quite far from the best efficiency point. The runner is designed so that the swirl generated by the guide vanes is more or less neutralized at the best efficiency point. However, at part load operation (away from the efficiency point), a strong swirling flow exits the runner. This is called a vortex rope. This phenomenon leads to large periodic pressure fluctuations that increase the risk of fatigue.

The importance of predicting such phenomena has lead to many studies. One of those is the Flow Investigation in Draft Tubes (FLINDT) research project [1]. The main objective of the FLINDT project was to investigate such phenomena and to provide an extensive database for a range of different operating points. Such experimental projects are usually complex and measurements are performed on reduced scale models. The team at the Politehnica University of Timisoara (UPT), National Centre for Engineering Systems with Complex Fluids (NCESCF) has developed such a simplified swirl generator to further study the precessing vortex rope [2]. The swirl generator was designed to give a swirl profile similar to that in the FLINDT project. The test rig was developed and manufactured in order to provide a good visualization of the phenomenon, as well as to investigate the velocity field of the swirling flow [3, 4]. The stay vanes and runner blades were designed using the inverse design methodology [15] in order to create a precessing vortex rope [5]. One of the purposes of this test rig is to investigate the viability of reducing pressure fluctuation of a precessing vortex rope by axial jet control in the discharge cone. This novel technique was introduced by Susan-Resiga et al. [6] to control the draft tube instability at partial discharge. Measurements on the test rig showed that a 10% jet discharge gives a maximum pressure recovery and creates no pressure fluctuations [7]. This conclusion was asserted by 3D unsteady numerical investigation using Fluent of the swirling flow using jet control [8]. It is nonetheless not acceptable to bypass the runner with such a large fraction of the turbine discharge. However, Susan-Resiga et al. [9] have also developed a flow feedback approach for the jet, that supplies the jet without any additional losses in the turbine.

The simple geometry of the test rig, as well as the quality of the measurements done by Bosioc et al. [10] makes this a very good case study for turbomachinery applications in OpenFOAM. OpenFOAM is an object oriented OpenSource library written in C++. With regards to basic features, such as turbulence models and discretization schemes, OpenFOAM is a competitive and high quality tool that is constantly evolving. Preliminary simulations were realized on the conical diffuser of the test rig [11], which showed that OpenFOAM gives as accurate results as commercial software.

The community driven OpenFOAM Turbomachinery Working Group [12] develops and validates OpenFOAM for turbomachinery applications. The swirling flow generator was chosen as a case study for the 5th OpenFOAM workshop held in Gothenburg, Sweden, and comparison between numerical results and measurements were presented. The goal of the Turbomachinery Working Group is to release this case study so that anyone who would like to learn OpenFOAM, or become more familiar with turbomachinery features in OpenFOAM, can learn how to set up, compute and analyse such problems.

2. Experimental setup

A cross-section of the test rig is presented in Fig. 1. The original test case was presented by Bosioc et al. [10] and was developed at Politehnica University of Timisoara. The swirling flow apparatus consists of four leaned strouts, 13 guide vanes, a free runner with 10 blades and a convergent divergent draft tube [3-5]. The guide vanes create a tangential velocity component, while keeping practically a constant pressure . The purpose of the free runner is to re-distribute the total pressure by inducing an excess in the axial velocity near the shroud and a corresponding deficit near the hub, like Francis turbine operation at 70% partial discharge. The runner blades act like a turbine near the hub, and a pump near the shroud. Thus the runner spins freely, without any total torque.

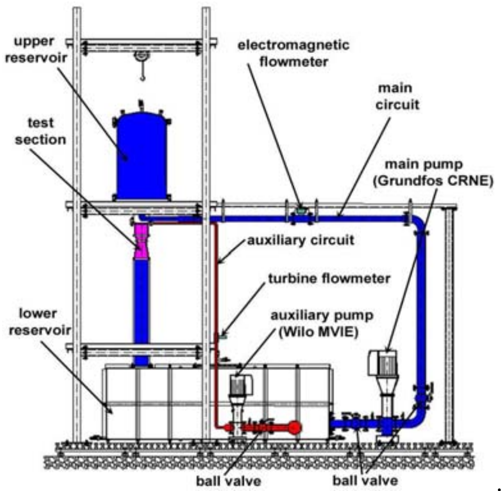


Fig. 1: Closed loop test rig for experimental investigations of swirling flow.

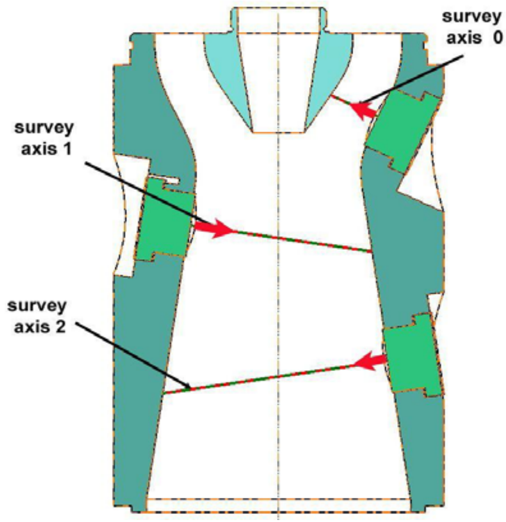


Fig. 2: Survey axes for LDV measurements

The test rig consists in a main circuit, used to supply the swirling flow section, and an auxiliary circuit that is used when tests are performed using axial jet control (see Fig. 1). The main centrifugal pump that provides the flow has a variable speed delivering 0-35 l/s. The swirling flow test section is made of plexiglass, so that visual observation is possible and to facilitate optical measurements. The measurements were realized at Politehnica University of Timisoara and were first presented by Bosioc et al. (10). The experimental data was measured with help of two-component Laser-Doppler Velocimetry (LDV). 10 μm aluminum particles were used to reflect the laser beam. The velocity measurements were realized in three different optical windows, the first located in the convergent part of the simplified draft tube, and the other two located in the axi-symmetric diffuser (see Fig. 2). The reference for the survey axis is set at the wall, as shown by the arrows in Fig. 2. On survey axis 0, 31 points were measured, while survey axis 1 contains 113 points, and survey axis 2 contains 141 points. The flow rate was kept at 80% of the maximum power of the pump, that is 30 l/s. The rotational speed of the free runner was 870 rpm. In order to get a time-averaged velocity profile, each point was sampled for a period of 25 seconds (5000 samples). Dimensionless form was used in the analysis of the velocity profiles, normalizing the abscissa by R_{Throat}=0.05m, and the velocity profiles by $v_{throat}=Q/\pi R^2_{throat}$, Q=30 l/s.

For clarity, in the following discussion, the first survey axis is called **W0,** the second survey axis is called **W1** and the third one **W2**.

3. Computational domains and OpenFOAM numerical set-up

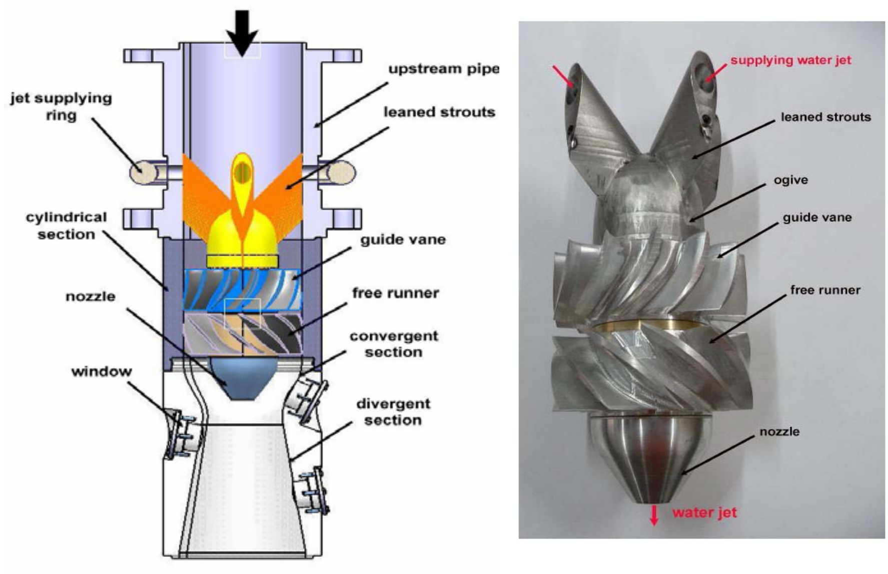


Fig. 3: Meridional cross-section of the swirling flow apparatus.

The computational domain consists of the whole test rig shown in Fig. 3. The mesh was generated in ICEM-Hexa, and consists of four different parts (see Fig. 4): the leaned strouts, the guide vanes, the free runner and the draft tube. The four different parts are coupled in OpenFOAM using the General Grid Interface (GGI), developed by Beaudoin and Jasak [13]. The mesh is fully hexahedral, and consists of 2.8 million cells. The incompressible unsteady Reynolds-Averaged Navier-Stokes equations are solved, using the finite volume method and the standard k-ε turbulence model. At the walls, the log-law treatment is applied, and the average y+ values range between 50-200. The boundary condition at the inlet is a plug-flow with the nominal discharge 30 l/s. The turbulence kinetic energy is set to 0.1, and the dissipation to 90, so that the turbulence intensity is of the order of 10% and the viscosity ratio $\nu_T/\nu=10$. The velocity and turbulence equations use the homogeneous Neumann boundary condition at the outlet. The pressure equation uses a homogeneous Neumann boundary condition at all boundaries, and at the outlet, the mean pressure is set to zero. The convection terms are discretized using a first-order upwind scheme, and the time terms are discretized using a second-order backward scheme. The time step is 2.7 10^{-4}, yielding a maximum Courant number of 5, and a mean Courant number of 0.2.

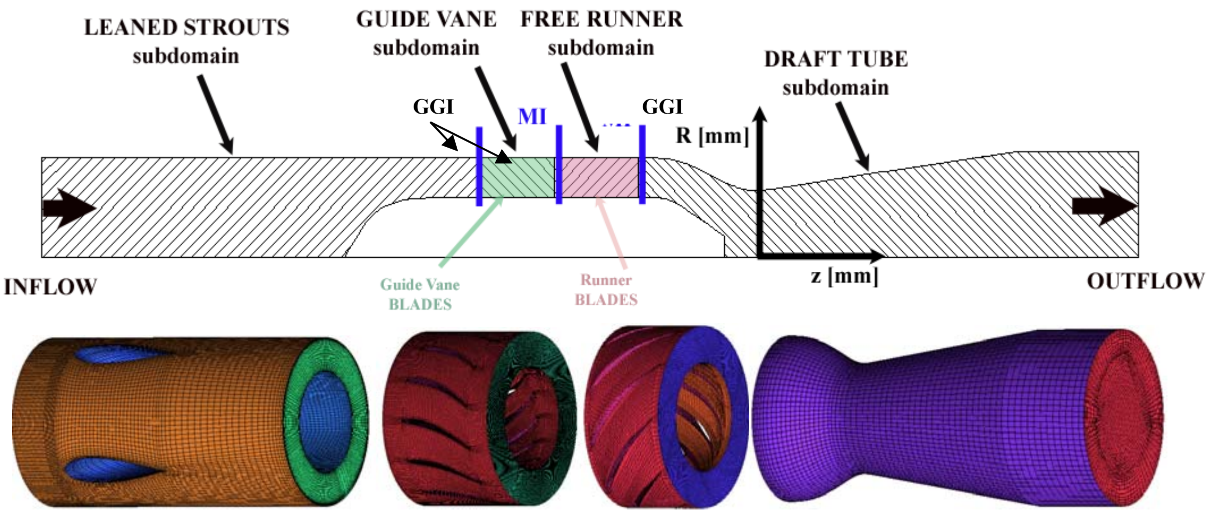


Fig. 4 . Half of meridional cross-section of the swirling flow apparatus (above) and associated computational domains and grids (below).

Two different computational techniques were used to predict the flow in the swirling flow test rig:

- The first method is a steady-state method, solving the steady Reynolds Averaged Navier Stokes equations. Because of the rotor-stator interaction in this case study, a frozen rotor solver is used. It is a steady state formulation where the rotor and stator are fixed with respect to each other, and different reference frames are used in the rotating and stationary parts. Though this method does not predict the flow features behind the runner accurately, it is a fast preliminary method and the general behavior of the flow is predicted.
- The sliding grid approach is a transient method where the runner mesh actually rotates with respect to the stator mesh. The URANS equations are solved and most of the unsteady flow features are predicted. However, due to the extra dimension added to the resolution (time), the simulation is time and computer resources consuming. The interaction between the rotor and stator is realized with the help of a sliding General Grid Interface [13].

However, it was shown in previous studies [14] that the prediction of the flow features by the steady-state method is not accurate enough. So the initial condition of the unsteady simulation is generated by the steady simulation, but the present work is focusing only on the unsteady results, and a comparison of those results with experimental and designed velocity profiles [5] is shown.

4. Comparison of numerical results against experimental and design data

4.1 Designed and computed velocity profiles for swirl generator

The comparison between the designed velocity profiles [5] and OpenFOAM is realized at four Section 1 (S1) and Section 2 (S2) shown in Fig. 5. The dimensionless velocity profiles are plotted against the radius of the different sections, divided by R_{throat}. Section 1 is downstream the guide vanes, and section 2 is downstream the free runner. The results computed by OpenFOAM, are time averaged. The results are shown in Fig. 6 and 7. The two velocity profiles at section 1 and 2 were designed for the swirling flow downstream in the draft tube. At section 1, the swirl created by the guide-vanes should have a free-vortex configuration, with quasi-constant axial velocity. The numerical results obtained with OpenFOAM are in good agreement with the designed profile, see Fig. 6. At section 2, although the axial velocity follows the intended profile rather well, the tangential velocity can not reach the intended value near the shroud, as shown in Fig. 7. This is probably due to the error in the rotational velocity of the runner. The error in prediction for the tangential velocity can be observed at each comparisons section located below the runner. If the velocity of the runner is corrected to the one that gives a free torque, the prediction of the tangential velocity should be more accurate.

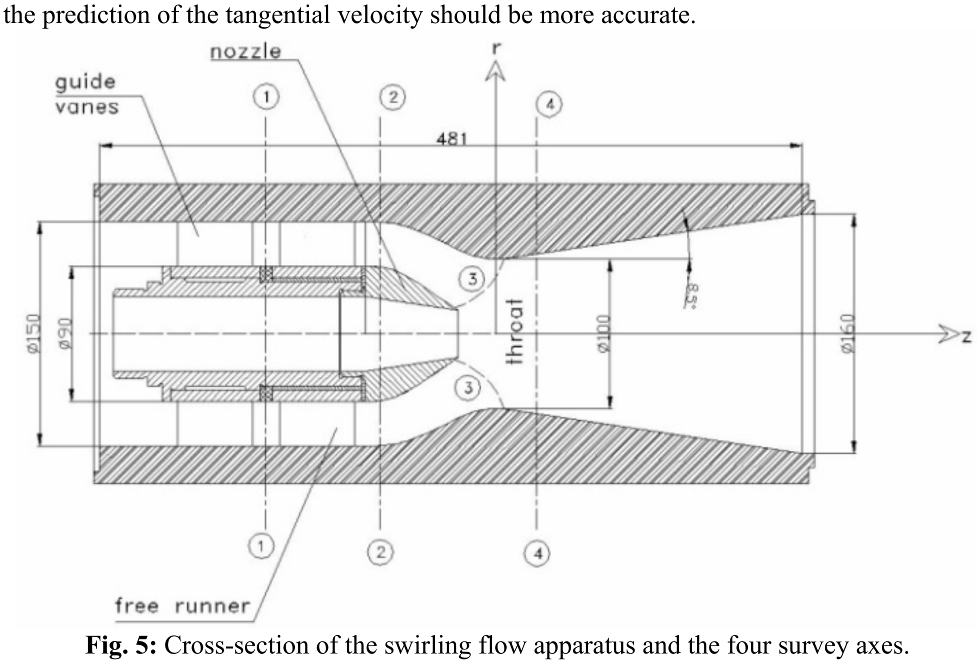


Fig. 5: Cross-section of the swirling flow apparatus and the four survey axes.

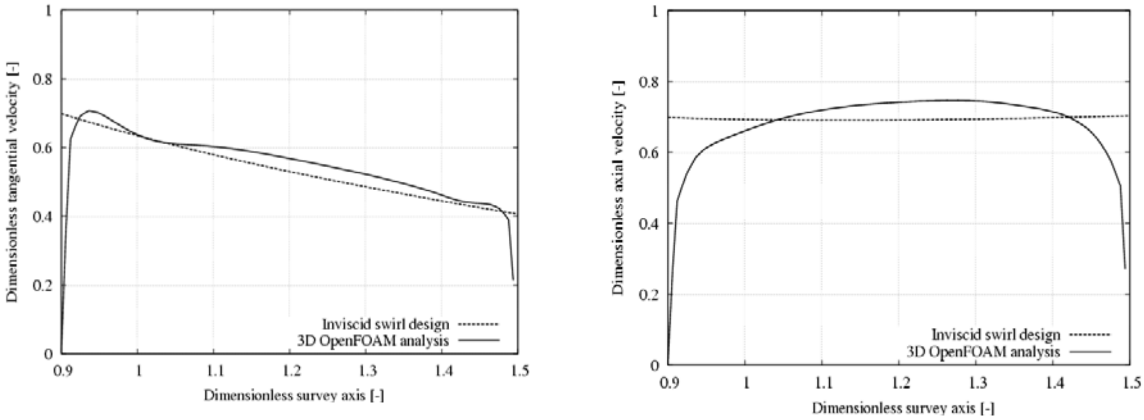


Fig. 6: Tangential and axial velocity profiles at section S1.

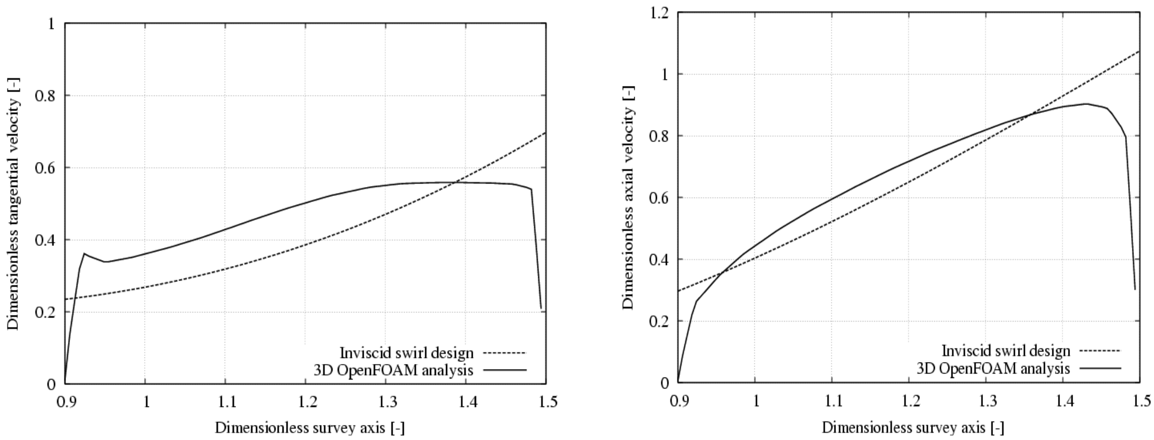
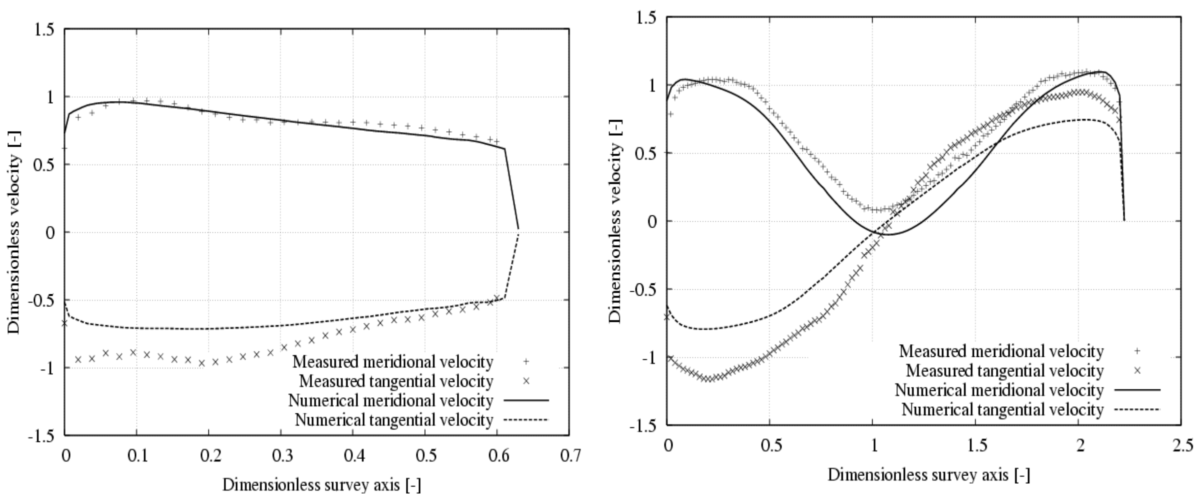


Fig. 7: Tangential and axial velocity profiles at S2.

3.2 Experimental and computed velocity profiles in test the section

The comparisons between the velocity profiles numerically predicted with OpenFOAM and experimental data are shown in Fig.8. The computed velocity profiles are time averaged over one vortex rope period.



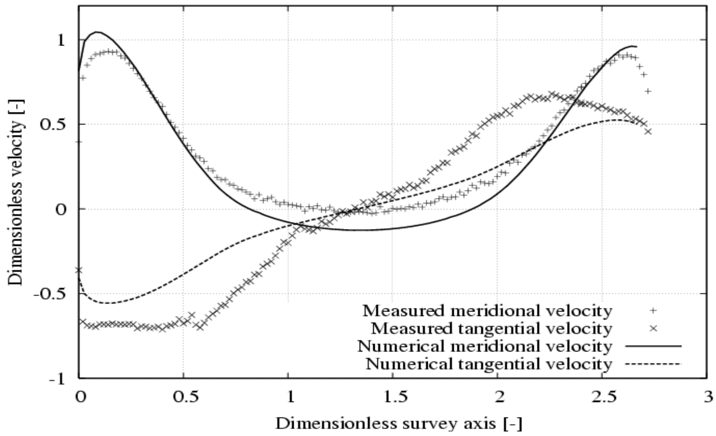


Fig. 8: Velocity profiles at W0 (top left), W1 (top right) and W2 (bottom)

At the first section W0, the numerical meridional velocity computed by OpenFOAM shows very good agreement with that of the experiment**.** However, the computed tangential velocity is a bit under predicted. This could be due to an underestimation of the runner speed. With a higher runner speed, the swirling flow should get stronger. This underestimation of the tangential velocity is found as well in the comparisons at sections W1 and W2, and has an impact on the stagnation region that can be seen in Fig**. 8**, at W1 and W2. Since the tangential velocity is not as large as it should be, more flow is pushed close to the walls, and less in the centre line. It can be seen that the stagnation region predicted by the measurements is becoming a recirculation region for the computed flow, with negative meridional velocity. This can probably be avoided by increasing the runner speed.

The slight difference between the numerical results and the measured data can also be justified by the fact that for this simulation, a first-order scheme was set for the convection schemes. A first-order scheme smears out gradients in the flow. A second-order scheme should predict those gradients much better. The unsteady simulation predicts the periodic fluctuations in the draft tube rather well, and a visualization of the computed vortex rope is shown in Fig. 9. Due to the time resolution of the sliding grid model, this visualization of the vortex rope oscillation can be done. Fig. 9 also shows the same vortex rope visualized in the experiments.

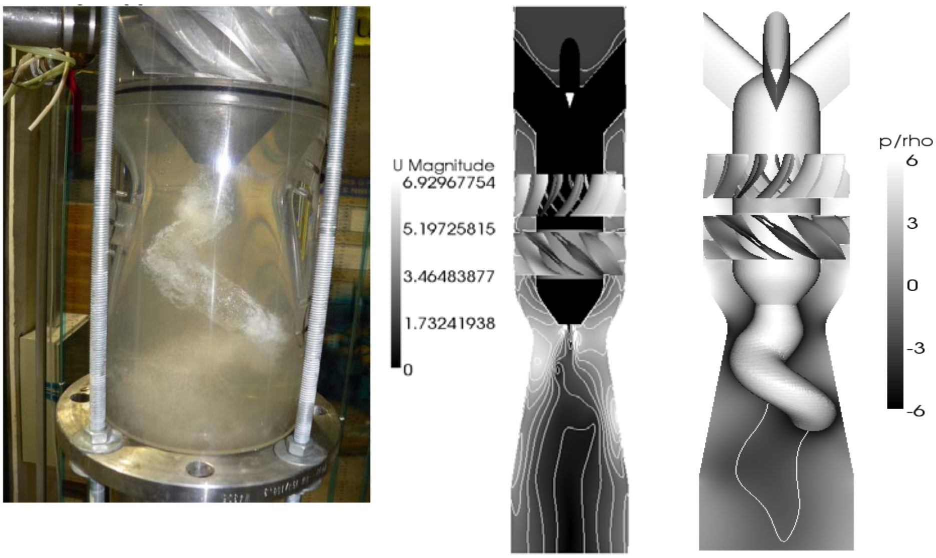


Fig. 9: The precessing vortex rope visualized in the experimental test section (left), and snap-shots of the velocity (centre) and pressure field (right) predicted by the numerical simulations in OpenFOAM. For the pressure snap-shot, the iso-surface visualizes a surface of constant pressure, while the iso-line shows where the axial velocity is zero.

3.3 Free runner speed

In the swirl flow test rig, the runner spins freely, without generating any torque. The measured rotational velocity of the runner while achieving a zero torque is Ω=870 rpm. For such angular velocity, the OpenFOAM simulation gave a negative moment of -0.7 Nm in the axial direction. By linear extrapolation, using a preliminary simulation at Ω=920 rpm, the angular velocity that makes the runner spins freely in OpenFOAM was estimated to be Ω=890 rpm, which was used in the present simulation. On the other hand, the moment reported by the simulation might correspond to the friction in the bearings in the physical model. A study of the impact of the runner speed will be made in the future.

The impact of the vortex rope on the runner moment is shown in Fig. 10, where it is possible to see two main frequencies. The low frequency is due to the rotating vortex rope, creating an oscillation in the runner torque, while the high frequency represents the rotor-stator interaction. This will be analyzed thoroughly in future studies. From Fig. 10, the average value of the moment can be evaluated to about -0.079 Nm. Though it is reasonably close to zero, a more accurate runner velocity can surely be found, and should give slightly higher swirling flow, and thus a better prediction of the tangential velocity and stagnation region.

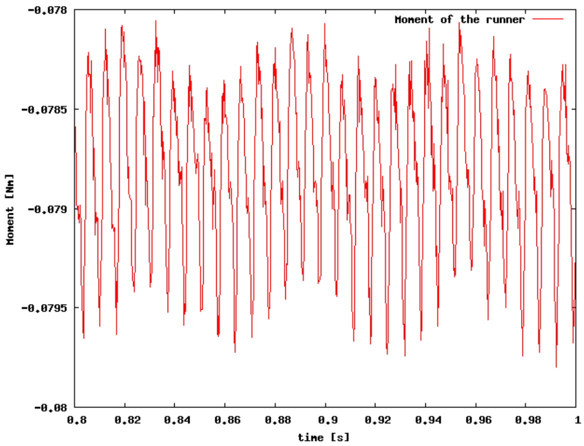


Fig. 10: Torque of the runner as a function of time.

5. Conclusion

Unsteady three-dimensional numerical investigations of the swirling flow with a precessing vortex rope in the swirling flow test rig have been performed. The flow is computed using the standard k-ε model in OpenFOAM. The velocity profiles predicted by the CFD code are accurate, and in good agreement with experimental LDV measurements and designed velocity profiles. The tangential velocity is under estimated, which is probably related to the inaccuracy that comes with the first-order scheme for the convection, and to the value of the runner moment. A more accurate turbulent model than k-ε (such as LES, or DES) should improve the predicted turbulent flow features in the middle of the cone. The same can be said by using a second-order scheme instead of a first-order for the convection terms.

Future work is to find the accurate runner speed for which in OpenFOAM the runner spins freely. An investigation of different turbulence models should as well lead to a better prediction of the flow, and the computed periodic draft tube pressure should be validated with experimental results.

Acknowledgements

Olivier Petit and Prof. Håkan Nilsson are financed by the Swedish Hydropower Centre (SVC). SVC has been established by the Swedish Energy Agency, Elforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University, www.svc.nu. They would like to would like to thank the Swedish National Infrastructure for Computing (SNIC) and Chalmers Centre for Computational Science and Engineering (C^3SE) for providing computational resources.

A. Bosioc, Dr. Sebastian Muntean and Prof. Romeo Susan-Resiga would like to thank the support of Romanian National Authority for Scientific Research through the CNCSIS PCE 799 project.

References

[1] Avellan F 2000 Flow Investigation in a Francis Draft Tube: The FLINDT Project *Proc. of the 20th IAHR Symp. on Hydr. Mach. and Syst.* (Charlotte, USA) p 11

[2] Susan-Resiga R, Muntean S, Bosioc A, Stuparu A, Milos T, Baya A, Bernad S and Anton L E 2007 Swirling Flow Apparatus and Test Rig for Flow Control in Hydraulic Turbines Discharge Cone Proceedings 2nd IAHR International Meetings of the Workgroup in Cavitation and Dynamic Problems in Hydraulic Machinery and Systems (*Scientific Bulletin of the Politehnica University of Timisoara, Transcations on Mechanics* **52(66)** Issue 6) 203-16

[3] Susan-Resiga R, Muntean S, Tanasa C and Bosioc A 2008 Hydrodynamic Design and Analysis of a Swirling Flow Generator *Proc. of the 4th German – Romanian Workshop on Turbomachinery Hydrodynamics* (Stuttgart, Germany, June 12-15)

[4] Bosioc A, Susan-Resiga R and Muntean S 2008 Design and Manufacturing of a Convergent-Divergent Test Section for Swirling Flow Apparatus *Proc. of the 4th German – Romanian Workshop on Turbomachinery Hydrodynamics* (Stuttgart, Germany, June 12-15)

[5] Susan-Resiga R, Muntean S and Bosioc A 2008 Blade Design for Swirling Flow Generator *Proc. of the 4th German – Romanian Workshop on Turbomachinery Hydrodynamics* (Stuttgart, Germany, June 12-15)

[6] Susan-Resiga R, Vu T C, Muntean S, Ciocan G D and Nennemann B 2006 Jet Control of the Draft Tube Vortex Rope in Francis Turbines at Partial Discharge *Proc. of the 23rd IAHR Symp. on Hydr. Mach. and Syst.* (Yokohama, Japan) p F192

[7] Muntean S, Susan-Resiga R, Bosioc A, Stuparu A, Baya A and Anton L E 2008 Mitigation of Pressure Fluctuation in a Conical Diffuser with Precessing Vortex Rope Using Axial Jet Control Method *Proc. of the 24th IAHR Symp. on Hydr. Mach. and Syst.* (Foz do Iguassu, Brazil)

[8] Muntean, S, Susan-Resiga R and Bosioc A 2009 Numerical Investigation of the Jet Control Method for Swirling Flow with Precessing Vortex Rope Proc. of the 3rd IAHR Int. Meet. of the Workg. on Cavit. and Dyn. Probl. in Hydr. Machin. and Syst. (Brno, Czech Republic)

[9] Susan-Resiga R and Muntean S 2008 Decelerated Swirling Flow Control in the Discharge Cone of Francise Turbines *Proc. of the 4th Int. Symp. on Fluid Mach. and Fluid Eng.* (Beijing, China) p IL-18

[10] Bosioc A, Susan-Resiga R and Muntean S 2009 2D LDV Measurements od Swirling Slow in a Simplified Draft Tube *Proc. of the CMFF*

[11] Muntean S, Nilsson H and Susan-Resiga R 2009 3D Numerical Analysis of the Unsteady Turbulent Swirling Flow in a Conical Diffuser Using Fluent and OPENFOAM *Proc. of the 3rd IAHR Int. Meet. of the Workg. on Cavit. and Dyn. Probl. in Hydr. Machin. and Syst.* (Brno, Czech Republic)

[12] Nilsson H, Page M, Beaudoin M and Jasak H 2008 The OpenFOAM Turbomachinery Working Group, and Conclusions from the Turbomachinery Session of the Third OpenFOAM Workshop *Proc. of the 24th IAHR Symp. on Hydr. Machin. and Syst.* (Foz do Iguassu, Brazil)

[13] Beaudoin M and Jasak H 2008 Development of a Generalized Grid Interface for Turbomachinery simulations with OpendFOAM *Proc. of the OpenSource CFD Int. Conf.* (Berlin, Germany)

[14] Petit O, Nillson H, Page M and Beaudoin M 2009 The ERCOFTAC Centrifugal Pump OpenFOAM Case-Study *Proc. of the 3rd IAHR Int. Meet. of the Workg. on Cavit. and Dyn. Probl. in Hydr. Machin. and Syst.* (Brno, Czech Republic)

[15] Zangeneh M 1991 A Compressible Three-Dimensional Design Method for Radial and Mixed Flow Turbomachinery Blades *Int. J. for Numerical Methods in Fluids* **13** 599-624