Comparison of numerical and experimental results of the flow in the U9 Kaplan turbine model

2010 IOP Conf. Ser.: Earth Environ. Sci. 12 012024

(http://iopscience.iop.org/1755-1315/12/1/012024)

Comparison of numerical and experimental results of the flow in the U9 Kaplan turbine model

O Petit[1], B Mulu[2], H Nilsson[1] and M Cervantes[2]

[1]Division of Fluid Mechanics, Chalmers University of Technology, Hörsalsvägen 7A, SE-41296 Göteborg, Sweden
[2]Division of Fluid Mechanics, Luleå University of Technology, SE-971 87 Luleå, Sweden

E-mail: olivierp@chalmers.se

Abstract. The present work compares simulations made using the OpenFOAM CFD code with experimental measurements of the flow in the U9 Kaplan turbine model. Comparisons of the velocity profiles in the spiral casing and in the draft tube are presented. The U9 Kaplan turbine prototype located in Porjus and its model, located in Älvkarleby, Sweden, have curved inlet pipes that lead the flow to the spiral casing. Nowadays, this curved pipe and its effect on the flow in the turbine is not taken into account when numerical simulations are performed at design stage. To study the impact of the inlet pipe curvature on the flow in the turbine, and to get a better overview of the flow of the whole system, measurements were made on the 1:3.1 model of the U9 turbine. Previously published measurements were taken at the inlet of the spiral casing and just before the guide vanes, using the laser Doppler anemometry (LDA) technique. In the draft tube, a number of velocity profiles were measured using the LDA techniques. The present work extends the experimental investigation with a horizontal section at the inlet of the draft tube. The experimental results are used to specify the inlet boundary condition for the numerical simulations in the draft tube, and to validate the computational results in both the spiral casing and the draft tube. The numerical simulations were realized using the standard k-ε model and a block-structured hexahedral wall function mesh.

1. Introduction

Harvesting energy from the flow of water has been done for centuries, and it presents many advantages that makes it widely attractive. Hydro power generates nowadays more than 17 % of the total electricity needs in the world. One of the advantages of a hydro power plant compared to other types of plants is its longevity. Many of the existing dams have been operative for the last century, and it seems that they will continue to do so for many more decades. The power plant is of course closely monitored and its health is regularly checked, but if well-built, a dam can last for a long time. However, due to diverse issues, such as cavitation or vibration, parts of the turbine-generator units need to be replaced every two decades or three. During this elapsed time, technology evolves, and so does the turbine design. It is the case for the Porjus hydraulic power plant located in the north of Sweden. First activated in 1915, one of its units called U9 was replaced during the last decade. The main focus of the new U9 Kaplan turbine is to allow research on a real scale turbine, rather than production [1]. Few detailed measurements are made in water turbines. During design phase, overall quantities such as efficiency are usually measured, rather than detailed pressure and velocity profiles. The U9 prototype facilitates such detailed investigations. It is somewhat smaller than the previous unit, and a curved pipe was inserted into the old penstock when assembling the new prototype.

The use of computational fluid dynamics (CFD) in the design and refurbishment process is becoming increasingly popular due to its flexibility, its detailed flow description and cost-effectiveness compared to the model testing usually used in the development of turbines. However, to get a good description of the flow in such large systems, the appropriate simulation parameters need to be set. Hence many models in CFD are calibrated and validated through simpler geometries and flow patterns. Analytical results, as well as experimental databases are often used to compare and validate simulations. The aim of the U9 project is to create a database of pressure and velocity patterns that can be use to validate CFD simulations, and to investigate the impact of the curved pipe on the flow in the U9 turbine [10]. The measurements are performed on a 1:3.1 scale Kaplan turbine model. The database is constituted of three different operative conditions: peak efficiency and two off-design points.

Accurate CFD results provide a good understanding of the flow in the turbine, and due to technological improvements, increase of RAM memory and CPU's speed for example, the whole water turbine can now be computed. However, studying such a system requires to do simulations on many computers, in parallel. This can become costly when commercial softwares are used. The alternative that was chosen in this work is OpenFOAM, which is an OpenSource CFD tool written in C++. Previous comparisons [2], [3], and [4] show that it is a competitive and high quality tool that gives similar results as commercial softwares.

This work presents comparisons between experimental and numerical results, in the U9 spiral casing and draft tube. In this work, the comparisons are limited to one working point, the best efficiency point (BEP), and limited to the velocity profiles.

2. Test rig and experimental setup

The measurements were performed at Vattenfall Research and Developmen test facility (VRD) [11]. Detailed velocity measurements were performed and reported by Mulu and Cervantes [5, 6], and pressure measurements by Jonsson and Cervantes [7]. The purpose of those measurements was to investigate the effect of the curved inlet pipe on the flow, and to generate a database for future numerical simulations, at the spiral casing inlet and in the spiral casing before the wicket gates.

2.1. Measurements in the U9 spiral casing

The test rig used to do the measurements is located at the Vattenfall Research and Development (VRD) model test facility in Älvkarleby, Sweden. It is a closed loop system where the absolute pressure in the downstream tank can be regulated to control the presence of cavitations. It is a 1:3.1 scale model of the U9 prototype. The best efficiency point corresponds to a guide vane angle of 26 degrees, and a discharge of 0.71 m^3/s. An overview of the test rig is given in Fig.1 where the curved inlet pipe can be observed.

The inlet of the U9 model spiral casing is a circular pipe with an inner radius of 316 mm. A plexiglas pipe 290 mm long was mounted between the inlet of the spiral casing, see Fig. 1. To avoid refraction caused by the surface curvature of the inlet pipe, a square optical box, filled with index matching liquid was placed around the pipe [5]. LDA measurements were performed from both sides of the pipe, along five horizontal and one vertical axis, as shown in Fig. 1.

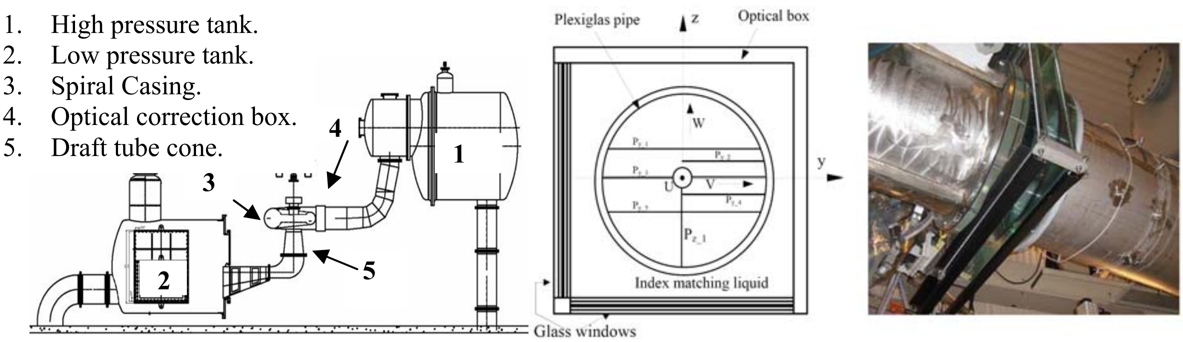


Fig. 1: Test rig of the U9 Kaplan turbine model (left), inlet section of the spiral casing and locations of the measured velocity profiles in the circular section (centre, right).

In the spiral casing, two windows were installed at the bottom, at the angular positions -56.25 degrees, (SI) and -236.25 degrees (SII) to perform LDA measurements, as shown in Fig. 2. The windows were placed at the centre of the casing. The LDA measurements were easier to perform than at the inlet, as the windows are planar, and thus do not cause any refraction problem, and do not need any correction.

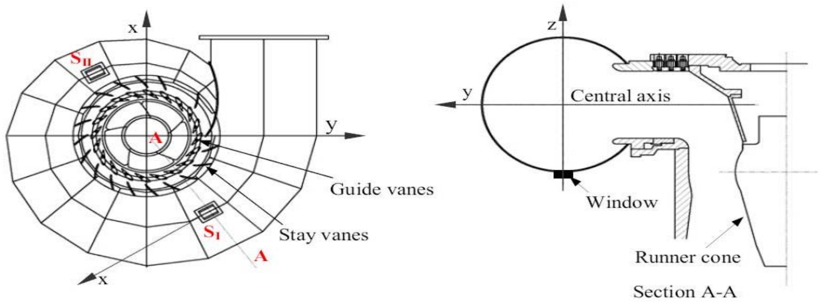


Fig. 2: Location of the windows in the spiral casing (left) and of the measurement axis z (right).

2.2. Measurements in the U9 draft tube

The draft tube velocity measurements were performed in the cone. To create a detailed database, four windows were mounted, with angular positions (AP) of 0, 90, 180 and 270 degrees, as shown in Fig. 3. Two types of measurements were performed. To create a inlet boundary condition for the numerical simulations, the axial and tangential velocity profiles were measured along section P at window a (0°), and d (270°). The velocity profiles are orthogonal to the draft tube cone axis, see Fig. 3. An average of both profiles was used as inlet boundary conditions for the draft tube simulations. To provide with a detailed database and validate the simulations, velocity measurements were also performed at sections I, II and III for the four different angular positions. For these measurements, the axis of the probe is orthogonal to the window and has an angle of 96.1° with the draft tube cone axis. Therefore, the tangential velocity is effectively measured while part of the axial and radial velocity is obtained for the other component. However, the snal cone angle (α=6.1°) combined with a probably negligible radial velocity (U_{radial}) allow to assume the measured velocity ($U_{measured}$) as axial velocity (U_{axial}). The error is estimated to be below 1%.

$$U_{measured} = U_{axial}\cos(\alpha) + U_{radial}\sin(\alpha) \approx U_{axial}$$

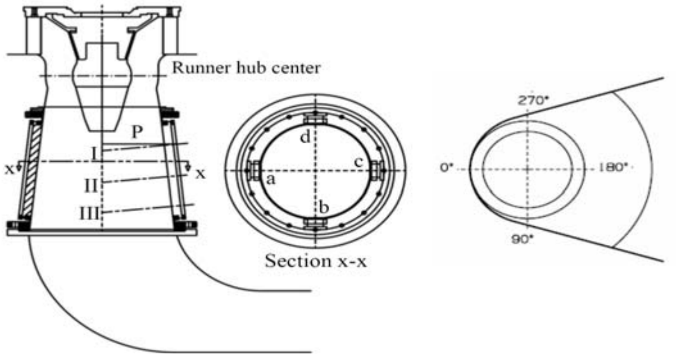


Fig. 3: Window locations at the draft tube cone: a is at 0°, b at 90 °, c at 180°, and d at 270°. P represents the velocity section measured for the inlet boundary condition for numerical simulations, and sections I, II and III for the validation.

2.3. Operating conditions

The measurements have been carried out at three different loads, at best operating point of the turbine and at two off-design operating points (to the left and right of the propeller curve). The operational net head H=7.5 m, the runner blade angle β=0.8°, and a runner speed N=696.3 rpm were used throughout the entire period of measurements. The working guide vane angle and the volume flow rate of the three working conditions are summarized in Table. 1. The results presented in this paper are at the best efficiency point.

Operating point	Left	BEP	Right
Guide vane angle (α) in degree	20	26	32
Volume flow rate Q (m^3/s)	0.62	0.71	0.76

Table. 1: Operational condition parameters.

3. Computational domain and OpenFOAM setup

3.1. Spiral casing computational domain and setup

The spiral casing computational domain starts with the high-pressure tank and includes the curved inlet pipe, the spiral casing as well as the wicket gate, see Fig. 4. The domain was realized in ICEM Hexa, and is divided in four different parts: part of the inlet tank, the inlet curved pipe, the spiral casing, and the wicket gate. Only the inlet tank is included in the computational domain, as a honeycomb is installed at the inlet of this tank, so that the velocity at the inlet is considered constant. Those four different parts are coupled in OpenFOAM using the General Grid Interface (GGI) developed by Beaudoin and Jasak [8]. The mesh is fully hexahedral, and consists of 5 million cells. The steady-state incompressible Reynolds-Averaged Navier-Stokes equations are solved, using the finite volume method and the standard k-ε model closure. At the walls the log-law treatment is applied and the average y+ values range between 50-100. The boundary condition at the inlet is a plug flow with the nominal discharge 0.71 m^3/s. The turbulent kinetic energy is calculated so that the turbulent intensity is 10%, and the turbulence dissipation is chosen so that $\nu_T/\nu=10$. At the outlet, a mean pressure of 0 is set. A second-order upwind scheme is set for the convection, while the first-order upwind scheme is used for the turbulence parameters.

3.2. U9 draft tube computational domain and setup

The draft tube mesh is fully hexahedral, and consists of 1.1 million cells. The draft tube computational domain is shown in Fig. 5. The same numerical set-up as in the spiral casing is used. The inlet boundary condition for velocity and turbulence is given by the measurements at section P, see Fig. 3 and Fig. 6.

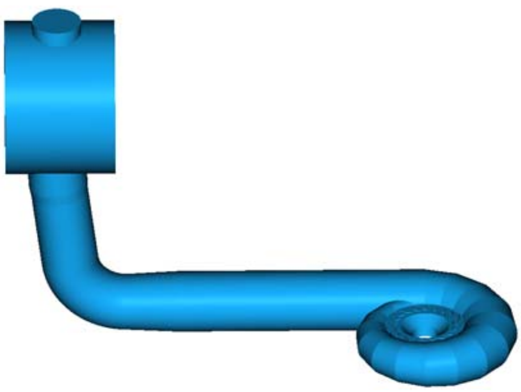


Fig. 4: Computational domain of the U9 spiral casing.

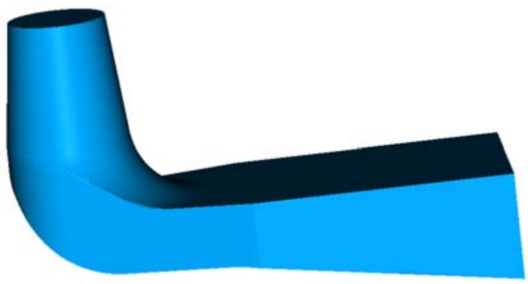
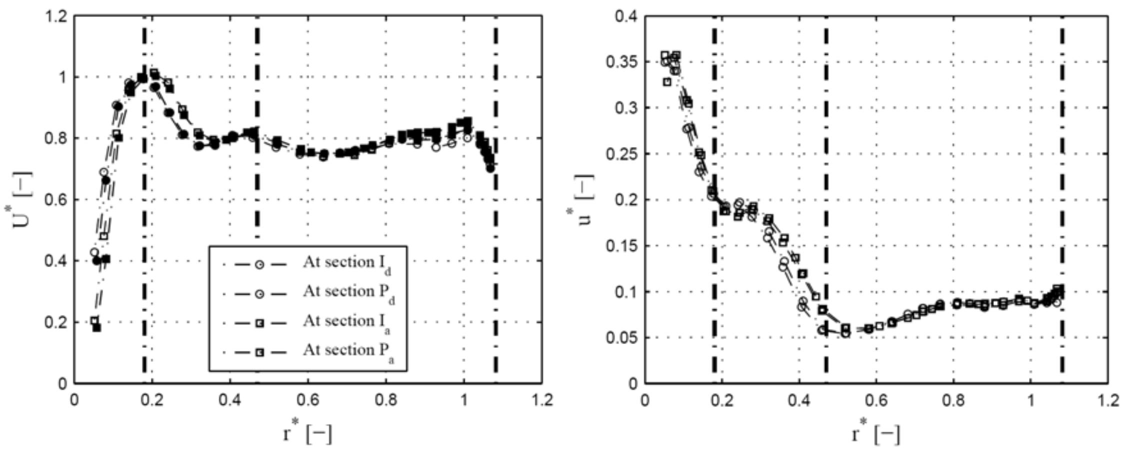


Fig. 5: Computational domain of the U9 draft tube.



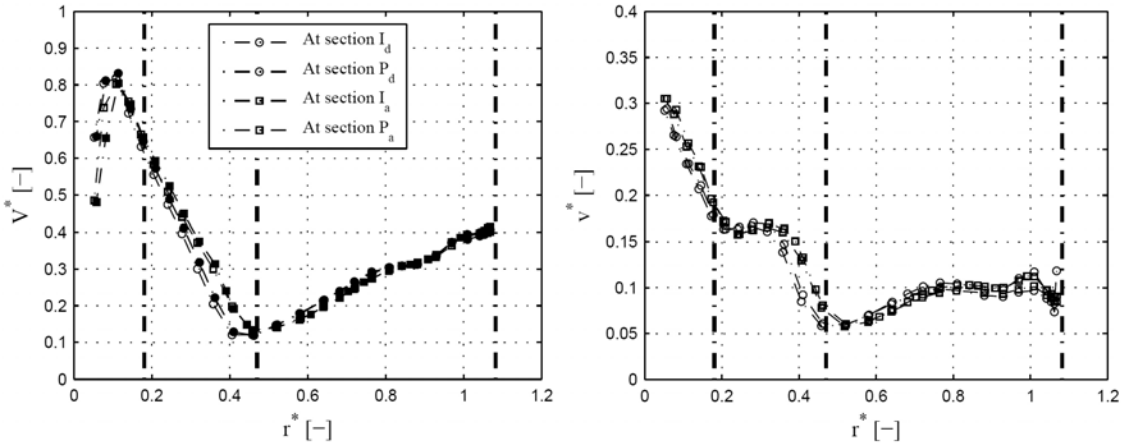


Fig. 6: Measured axial (U*) and tangential (V*) velocity profiles at sections P and I. The profiles on the right hand side represent the corresponding measured RMS for the axial (u*) and tangential (v*) velocities.

4. Results

An overview of the flow in the spiral casing and draft tube is shown in Figs. 7, 8, 9, and 10. The importance of including the curved pipe in the computational domain is shown in Fig.7, and the recirculation is still visible at the inlet of the spiral casing, in Fig. 8. The accuracy of the recirculation prediction in the numerical results is strongly dependent on the choice of the turbulence model that is made, as well as the choice of boundary condition at the wall. Detached-eddy Simulation (DES) should predict the length of the detached boundary layer, and hence the recirculation, much better than the k-ε model. That will be done in future work.

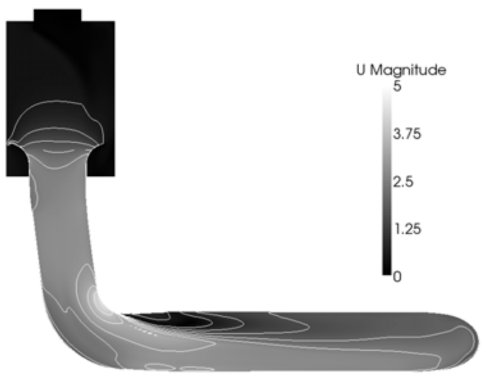


Fig. 7: Velocity magnitude in the inlet pipe.

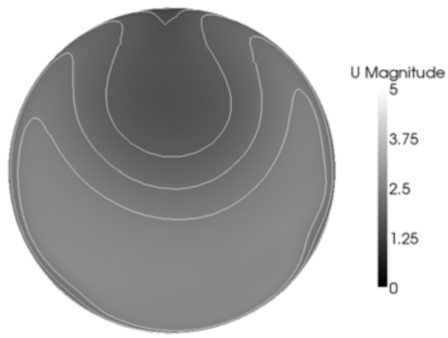


Fig. 8: Velocity magnitude at the spiral casing inlet measurement section.

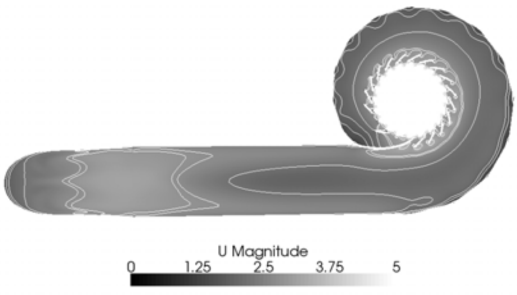


Fig. 9: Velocity magnitude in the spiral casing.

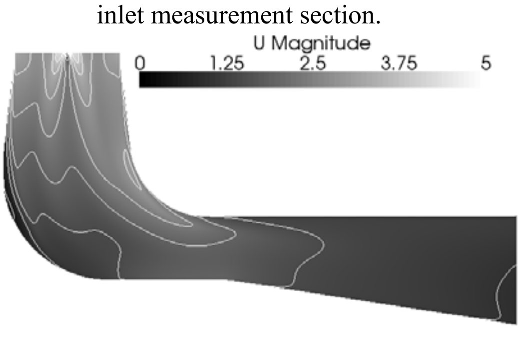


Fig.10: Velocity magnitude in the draft tube.

.

4.1. Results in the spiral casing

At the inlet of the spiral casing, the experimental axial velocity profiles at Py_1, Py_5 and Pz, specified in Fig. 1, are compared with the numerical results in Fig.11. The analysed data are presented in dimensionless form using, R_{inlet}=0.316m, and $v_{inlet}=Q/\pi \cdot R^2_{inlet}$, Q=0.71 $m^3.s^{-1}$. The flow predicted by OpenFOAM shows a similar behaviour as the measured flow. The axial velocity at Py_1 is lower than that at profile Py_5. The M-shape character in the velocity distribution is due to the pair of counterrotating Dean vortices, which is known in a circular bend flow [12]. Since the axial velocity distribution is not uniform in the plane due to lower velocity close to the upper wall, fluid particles with higher velocity are forced to move to the outer side, and those with lower velocity to the centre. This is due to the curvature which causes a positive gradient of the centrifugal force from the centre to the outer wall. This force and the presence of a boundary layer at the wall due to the fluid adhesion to the wall combined are responsible for this kind of flow behaviour [12].

The difference between the prediction of OpenFOAM and the measured flow can be explained by the fact that the simulations use a basic k-ε model that does not fully predict the flow in the recirculation region after the bend, shown in Fig. 7. At profile P_z, which is at the center line of the pipe, the numerical results show good agreement with the experimental values.

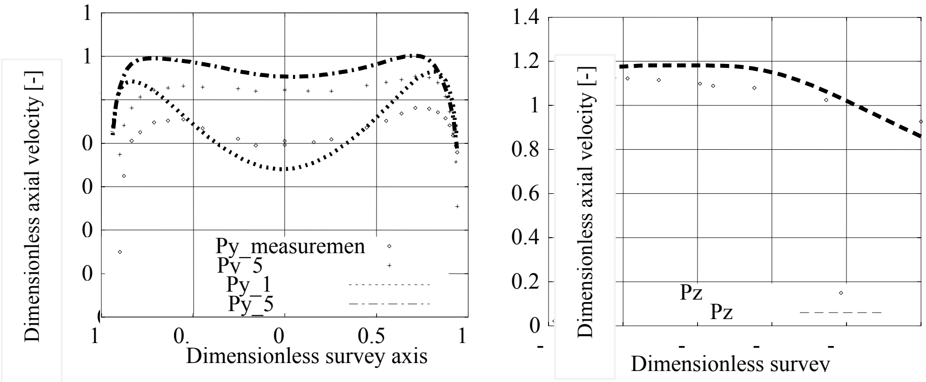


Fig.11: Axial velocity profiles at profile Py_1 and Py_5 (left), and Pz (right).

The comparison inside the spiral casing is done at the two positions (SI and SII) described in Fig. 2. Axial and tangential velocity profiles are shown in
Fig**.** 12. The reference point is located at the lower wall of the spiral casing.

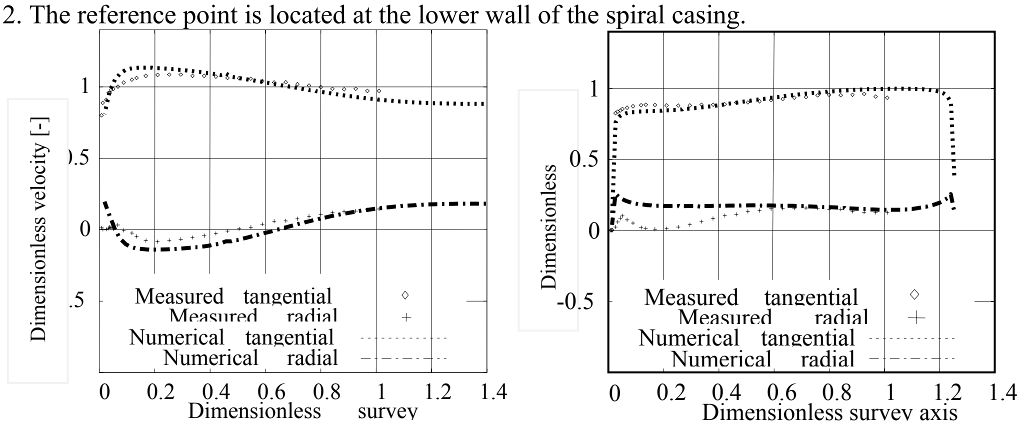
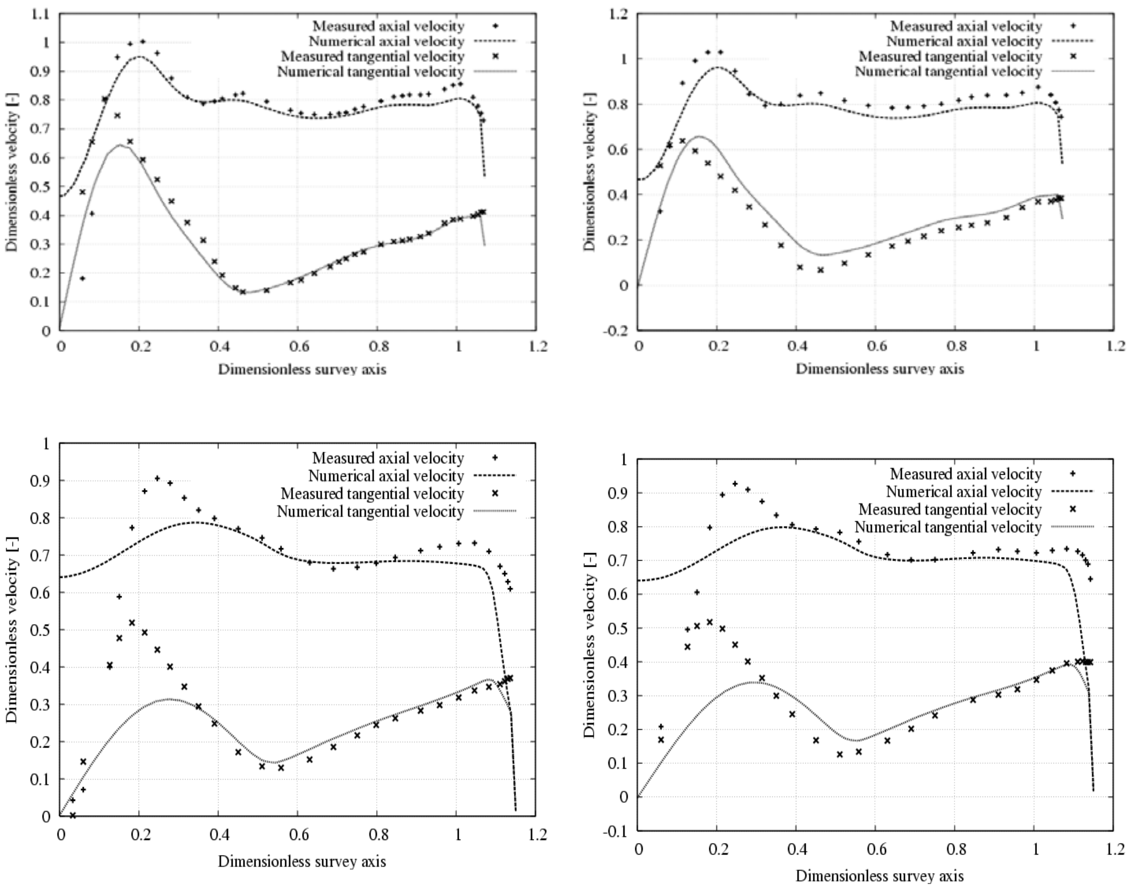


Fig.12: Dimensionless radial and tangential velocity at SI (left) and SII (right), inside the spiral casing.

In this comparison, the radial velocity is defined as positive toward the centre. The radial velocity measured at location SI shows the presence of a secondary flow yielding negative radial velocities. This secondary flow is not observed at location SII. The numerical prediction of the flow inside the spiral casing is accurate, though it tends to yield a slighlty different radial velocity distribution than the experimental measurements at Section I, which may be a consequence of the flow prediction in the bend.

4.2. Results in the draft tube

In the draft tube, comparisons of the numerically predicted flow and the measurements are made in two different windows, a and c (see Fig. 3). The results are presented in Fig.13. The reference point for this comparison is at the centreline of the draft tube, and the survey axis is normalized by the radius of the runner, R_{Runner}=0.25m. Close to the inlet of the draft tube, the predicted flow is similar to the measured flow. However, the difference at the centre of the draft tube is already visible. The numerical simulation does not predict the flow features very well in the draft tube, and the more downstream it goes, the less accurate the results become. The reason to this unaccuracy is the inlet boundary condition. The velocity profile set at the inlet of the draft tube is an average value of the flow measured at window a and d at section SI. That means that we set a symmetric swirling flow at the inlet. In reality, the swirl flow is asymmetric as wakes are coming down from the runner. Furthermore, close to the centre (0-13 mm), the axial and tangential velocities were approximated since no experimental data were available, see Fig. 6. It is further questionable to specify a steady axi-symmetric inlet boundary condition in a region of high unsteadiness and non-axi-symmetry. Finally, it was shown at the Turbine-99 workshop [9] that ignoring the radial velocity profile at the draft tube inlet when setting the boundary condition leads to a wrong prediction of the flow in the draft tube. In the present work, the inlet boundary condition is much below than for the Turbine-99 test case. The radial velocity measurements were not available, and thus neglected. Close to the wall, however, the flow is rather well predicted, both for the elbow window (window c) and for the window a. This is promissing, and a more precise inlet boundary condition should result in an accurate prediction of the flow. However, for accurate results, the runner must be included in the simulation. That will be done in future work.



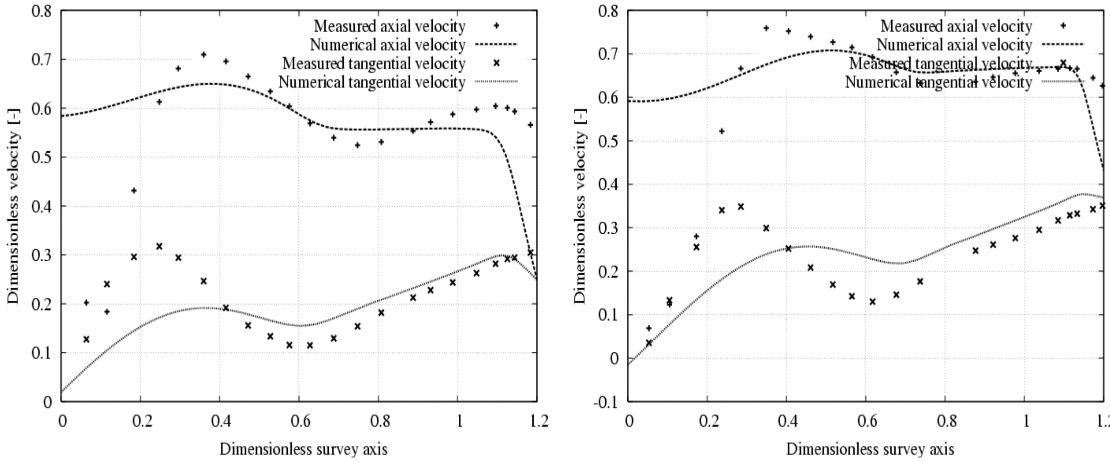


Fig.13: Velocity profiles for sections Ia (top left), Ic (top right), IIa (middle left), IIc (middle right), IIIa (down left), and IIIc (down right).

Conclusion

Comparisons between numerical simulations and experimental results in the U9 spiral casing and draft tube have been presented. The results show the importance of setting appropriate boundary conditions when numerically trying to predict the flow in water turbines.

In the spiral casing, the comparison shows the effect a curved pipe has on the spiral casing inlet flow. In the draft-tube, the importance of defining an appropriate inlet boundary condition is the key to succesfully predict the flow features of the draft tube. However, to predict the flow as accurately as possible, it is important to couple the runner with the draft tube, and to compute unsteady simulation.

The numerical simulations predict the overall flow features in the spiral casing and in the draft tube rather well. The predicted flow is similar to the measured flow, and this work sets the base for future investigations. Future work in the U9 project is the investigation of more accurate turbulence models in the spiral casing, in order to accurately predict the turbulent flow features. In the draft tube, an estimation of the radial velocity should get a better flow prediction. Ultimately, unsteady simulations of the U9 Kaplan turbine coupled with the draft tube using a sliding grid should predict a very accurate flow.

Acknowledgement

The research presented in this work was carried out as a part of 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.

The authors 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. The U9 turbine geometry was shared by Andritz, and their collaboration and help is gratefully ackowledged.

References

[1] Cervantes M J, Jansson I, Jourak A, Glavatskikh G and Aidanpää J O 2008 Porjus U9, A Full-Scale Hydropower Research Facility *Proc. of the 24th IAHR Symp. on Hydraulic Machinery and Systems* (Foz Do Iguassu, Brazil)
[2] Petit O, Nilsson H, Vu T, Manole O and Leonsson S 2008 The Flow in the U9 Kaplan Turbine - Preliminary and Planned Simulations Using CFX and OpenFOAM *Proc. of the 24th IAHR Symp. on Hydraulic Machinery and Systems* (Foz de Iguassu, Brazil)
[3] Nilsson H and Page M 2005 OpenFOAM simulation of the flow in the Hölleforsen draft tube model *Proc. of the 3rd IAHR/ERCOFTAC Workshop on draft tube flows* (Porjus, Sweden)
[4] 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 *3rd IAHR Int. Meetings of the Workshop on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems* (Brno, Czech Republic) p C4
[5] Mulu B 2009 *Experimental and Numerical Investigation of Axial Turbine Models* (Licenciate Thesis Luleå

University of technology, Sweden)

[6] Mulu B and Cervantes M 2009 Experimental Investigation of a Kaplan Model with LDA *33rd IAHR Congress Water Engineering for a Sustainable Environment*

[7] Jonsson P and Cervantes M *Time Resolved Pressure Measurements on a Kaplan Model*

[8] Beaudoin M and Jasak H 2008 Development of a Generalized Grid Interface for Turbomachinery simulations with OpenFOAM *OpenSource CFD International Conference* (Berlin, Germany)

[9] Cervantes M, Gustavsson L, Page M and Engström F 2006 Turbine-99, a summary *23rd IAHR Symp.* (Yokohama, Japan)

[10] Mulu B and Cervantes M 2007 Effects of Inlet Boundary Conditions on Spiral Casing Simulations Proc. of the 2nd IAHR Int. Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, (Timisoara, Romania) (*Scientific Bulletin of the "Politehnica" University of Timisoara Transactions on Mechanics, Tom 52(66), Fascicola 6*)

[11] Marcinkiewics J and Svensson S 1994 Modification of the Spiral Casing Geometry in the Neighbourhood of the Guide Vanes and its influence on the Efficiency of a Kaplan Turbine *Proc. of the 17th IAHR Meetings on hydraulic machinery and cavitation* Vol. 1 pp 429-434

[12] Boiron O, Deplano V and Pelissier R 2007 Experimental and Numerical Studies on the Starting Effect on the Secondary Flow in a Bend *J. Fluid Mech.* **574** 109-209