IAHR
24th Symposium on Hydraulic Machinery and Systems
OCTOBER 27-31, FOZ DO IGUASSU

THE FLOW IN THE U9 KAPLAN TURBINE - PRELIMINARY AND PLANNED SIMULATIONS USING CFX AND OPENFOAM

AUTHORS

Olivier Petit(1), Håkan Nilsson(1), Thi Vu(2), Ovidiu Manole(2), Svante Leonsson(3)

[1] Chalmers University of Technology, Sweden
[2] Andritz VA Tech Hydro Ltd., Canada
[3] Andritz Hydro Inepar Sweden AB
Corresponding author: Olivier Petit, olivierp@chalmers.se

ABSTRACT

The present work compares the CFX and OpenFOAM CFD codes with respect to the prediction of the flow in the U9 Kaplan turbine spiral casing, distributor and draft tube. The simulations use similar settings and the same computational grids – unstructured wall-function grids with 10.3M cells in the spiral casing and distributor, and 1.04M cells in the draft tube. The results show that the two codes give similar results in the spiral casing and distributor, and almost identical results in the draft tube. Previous studies [1] have shown the same behaviour in the Turbine-99 draft tube, for a block-structured wall-function grid. There are however no previous studies where the flow in a spiral casing and distributor have been studied and compared using the same settings and computational grid in CFX and OpenFOAM.

The next phase of the project consists of comparisons with the results from an on-going experimental investigation.

KEY WORDS: CFD, Water Turbine, Draft Tube, Spiral Casing, Distributor, CFX, OpenFOAM

INTRODUCTION

The U9 Kaplan turbine is a unique combination of model and full scale research units. The full scale turbine is located in Porjus, in the most northern parts of Sweden, and the model scale turbine is located at Vattenfall Research and Development in Älvkarleby, in the center of Sweden. The present paper describes the initial work done, and the planned activities, in a newly started PhD and collaboration project regarding simulations of the flow in the U9 Kaplan turbine. CFX simulations are being done at Andritz VA Tech Hydro Ltd. (Canada), and OpenFOAM simulations are being done at Chalmers University of Technology, Sweden. With similar settings for the simulations and using the same computational grid it is possible to compare different aspects of the simulations, such as accuracy, efficiency and general features of the two codes. Detailed measurements of the flow in both model and full scale will be made in other PhD projects, and those measurements will be used to validate the computational results.

The present paper shows preliminary computational results from CFX and OpenFOAM, of the flow in the U9 spiral casing, distributor, and draft tube. Andritz Hydro Inepar Sweden AB has designed and provided the geometrical information. Andritz VA Tech Hydro Ltd. (Canada) has provided the CAD models, the grids, and the CFX computations. The OpenFOAM computations have been done at Chalmers. The preliminary results show that CFX and OpenFOAM give similar results, but with some significant differences. Further work will be done in order to find out the reason for the differences, and to propose how to minimize the differences in order to get the most accurate results.

In this initial phase of the project the focus has been on the flow in the spiral casing, distributor and draft tube. Soon also the runner, and the coupling between the different parts of the system will be studied. One of the main aims of the project is to study the influence of different choices of boundary conditions on the computed flow. As an example, in both the model and full scale there is a 90 degree bend on the inlet pipe at approximately 5 diameters before the inlet of the spiral casing. The influence of this bend on the flow throughout the turbine will be studied. Internal boundary conditions, such as rotor-stator interfaces, and GGI interfaces will also be studied.

METHODOLOGY

Both the CFX and OpenFOAM CFD codes use similar approaches to solve for the velocity, pressure, and modeled turbulence fields. The basis is the Finite Volume Method, where the computational domain is divided into a large number of polyhedral control volumes (cells) in which the equations are discretized and solved iteratively. In the present work the computational grids consist of a mix of tetrahedral, pyramid and prism cells. The prism cells are used in the boundary layers in order to increase the accuracy of the results in those regions where there are large gradients. The effect of turbulence is modeled using the standard k-ε turbulence model. All the computations in the present work are steady. Boundary conditions for all the variables must be set at all the outer boundaries, and discretization schemes must be specified in order to complete the set-up of the problem. Those settings are described below for each of the two cases. The name of the OpenFOAM solver is simpleFoam, and the version of OpenFOAM is 1.4.1. The version of CFX is 10.0 SP1.

Description of the U9 spiral casing and distributor case

Figure 1 shows a view of the computational domain of the U9 spiral casing and distributor. The geometry has been scaled to correspond to a runner throat diameter of 1m in this study. There are 10.3M control volumes in the unstructured grid. The final near-wall grid resolution corresponds to an average $y^+ = 35$, and standard wall-functions are used at the walls.

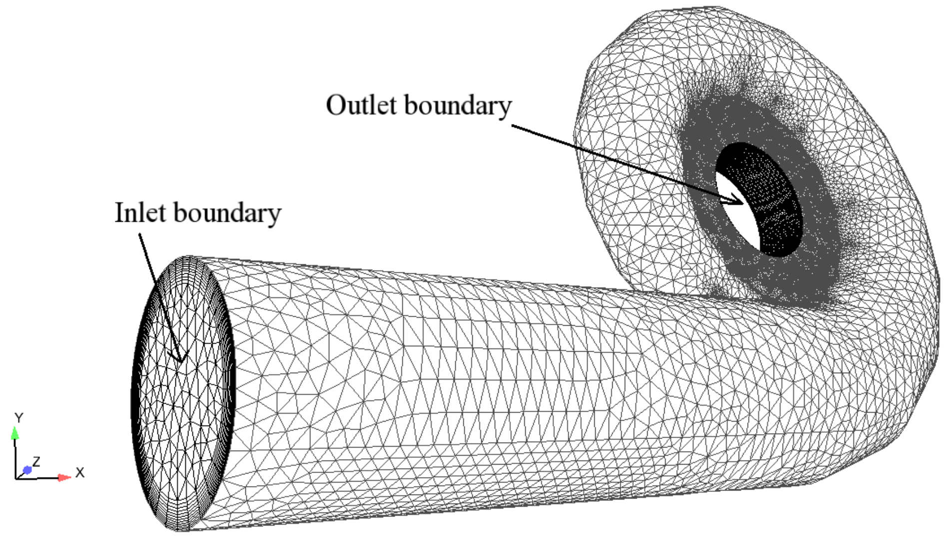

Figure 1: The spiral casing and distributor computational domain and unstructured grid

Constant inlet values are specified for the velocity and turbulence quantities. The turbulence intensity, I, is 2%, and the turbulence length scale is given by the viscosity ratio $\nu_t/\nu=50$. The values are given by $\mathbf{U}=[1.88727, 0, 0]$m/s, $k=1.5(U_x{*}I)^2 m^2/s^2$, and $\varepsilon=C_\mu\, k^2/((\nu_t/\nu)\ \nu) m^2/s^3$. The Neumann boundary condition is used for the velocity and turbulence quantities at the outlet. For the static pressure, the Neumann boundary condition is used at all boundaries, but the level of the static pressure is corrected by setting the average outlet static pressure to zero.

The convection terms in the Navier-Stokes equations are discretized using the GammaV discretization scheme [2] in OpenFOAM, while the "High Resolution" discretization scheme has been used in CFX.

Description of the U9 draft tube case

Figure 2 shows the computational domain of the U9 draft tube. The geometry has been scaled to correspond to a runner throat diameter of 1m in this study. There are 1.04M control volumes in the unstructured grid. The final near-wall grid resolution corresponds to an average $y^+ = 231$, and standard wall-functions are used at the walls.

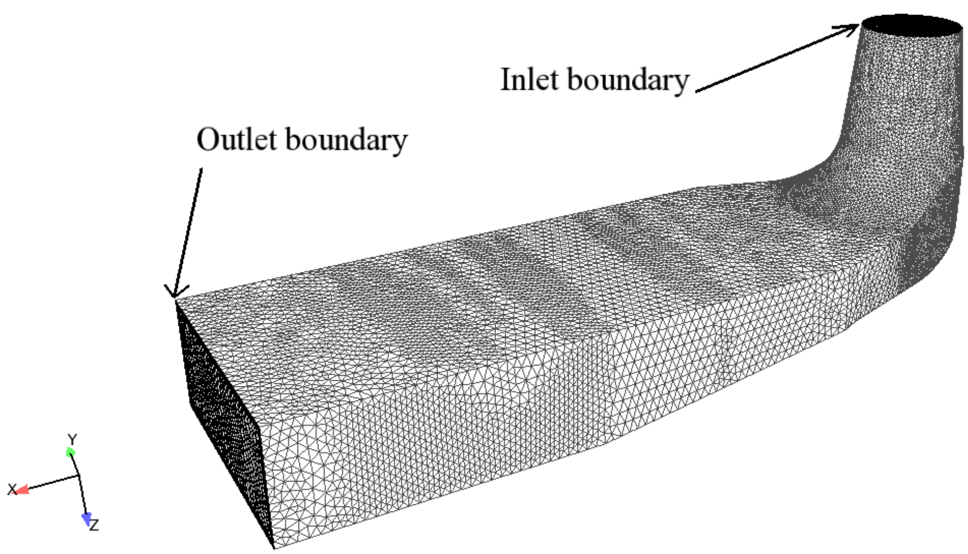

Figure 2: The U9 draft tube computational domain and unstructured grid

At the inlet a solid-body swirl is applied since there are yet neither experimental nor computational results available at that cross-section. The turbulence intensity, I, is 5%, and the turbulence length scale is given by the viscosity ratio $\nu_t/\nu=100$. Neumann boundary conditions are used at the outlet. For the static pressure, the Neumann boundary condition is used at all boundaries, but the static pressure level is corrected by setting the average outlet static pressure to zero.

The convection terms in the Navier-Stokes equations are discretized using the GammaV discretization scheme [2] in OpenFOAM, while the "High Resolution" discretization scheme has been used in CFX.

RESULTS

The computational results are first compared and discussed for the spiral casing and distributor, and then for the draft tube.

The spiral casing and distributor

Here the results from the CFX and OpenFOAM simulations of the flow in the U9 spiral casing and distributor are compared. The integrated losses are compared between the three cross-sections shown in Figure 3. The velocity and modeled turbulence are compared along the inlet and outlet lines, which are also shown in Figure 3. In the evaluation of the losses the *Spiral Casing* is defined between the cross-sections referring to the *inlet of the spiral casing*, and the *inlet of the distributor*. The *Distributor* is defined between the cross-sections referring to the *inlet of the distributor*, and the *outlet of the distributor*.

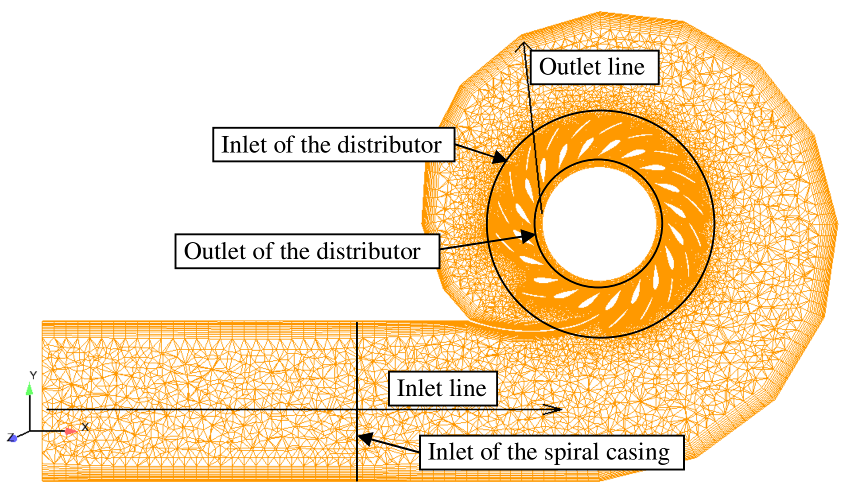

Figure 3: Locations of the three cross-sections, and the inlet and outlet comparison lines

Table 1 shows the integrated losses in the *Spiral Casing* and the *Distributor*. The losses are defined as $Loss = \frac{\Delta P_{tot}}{\frac{1}{2} * \rho * v_{Throat}^2} * \frac{1}{Q_{11}^2} * 100$, where ΔP_{tot} is the change in mass-flow averaged total pressure between the two cross-sections. $Q_{11} = \frac{Q * \sqrt{2 * g}}{D_{Throat} * v_{Throat}} = \frac{v_{Throat} * A_{Throat}}{D_{Throat} * v_{Throat}} * \sqrt{2 * g} = \pi * \sqrt{\frac{g}{2^3}}$ is the reduced discharge, where subscript *Throat* refers to the runner cross-section. The major difference between the results of the two codes with respect to the loss is in the *Spiral Casing*, although that is more or less flow in a curved channel. The properties of the simulations that are mostly affecting the predicted losses in this region are the near-wall modeling, the turbulence modeling, and the numerical diffusion due to the choice of convection discretization scheme. The reason for the difference will be examined thoroughly in future work, but a small preliminary discussion will also be made when comparing the static pressure results along the outlet line.

	Loss from CFX (%gH)	**Loss from OpenFOAM (%gH)**
Spiral Casing	0.10146	0.27687
Distributor	0.31322	0.390207
Total	0.414679	0.667086

Table 1: Losses computed in CFX and OpenFOAM

Figure 4 shows comparisons of the distributions of velocity magnitude and static pressure along the inlet line and outlet line shown in Figure 3. The results from the two codes are very similar, although there are some significant differences. The most obvious difference is the static pressure level in the spiral casing, along the inlet line, and at the outer part of the outlet line. The difference in static pressure is rather constant in those regions. However, at the cross-section at the inlet of the distributor (Abscissa ~0.62) the two codes give identical results both for the static pressure and the velocity magnitude, and thus also for the total pressure. This suggests that the main contribution to the difference in the loss from the two codes originates in the region where the flow enters the distributor. That region is part of the evaluation of the loss in the *Spiral Casing*, which is then blamed for the difference although the difference seems to be much more localized to the inlet of the distributor. This will be investigated thoroughly in the near future.

Figure 5 shows comparisons of the distributions of turbulent kinetic energy and turbulence dissipation along the inlet line and outlet line shown in Figure 3. The differences in the turbulence properties are significant, but not unexpected. The turbulence properties are very sensitive, and the small differences we see in the velocity distribution give slightly different flow direction and velocity gradients, which may give large differences in the modeling of the turbulence. It can only be concluded that the results are identical along the inlet line, and show qualitative similarity at the outlet line with respect to the turbulence quantities.

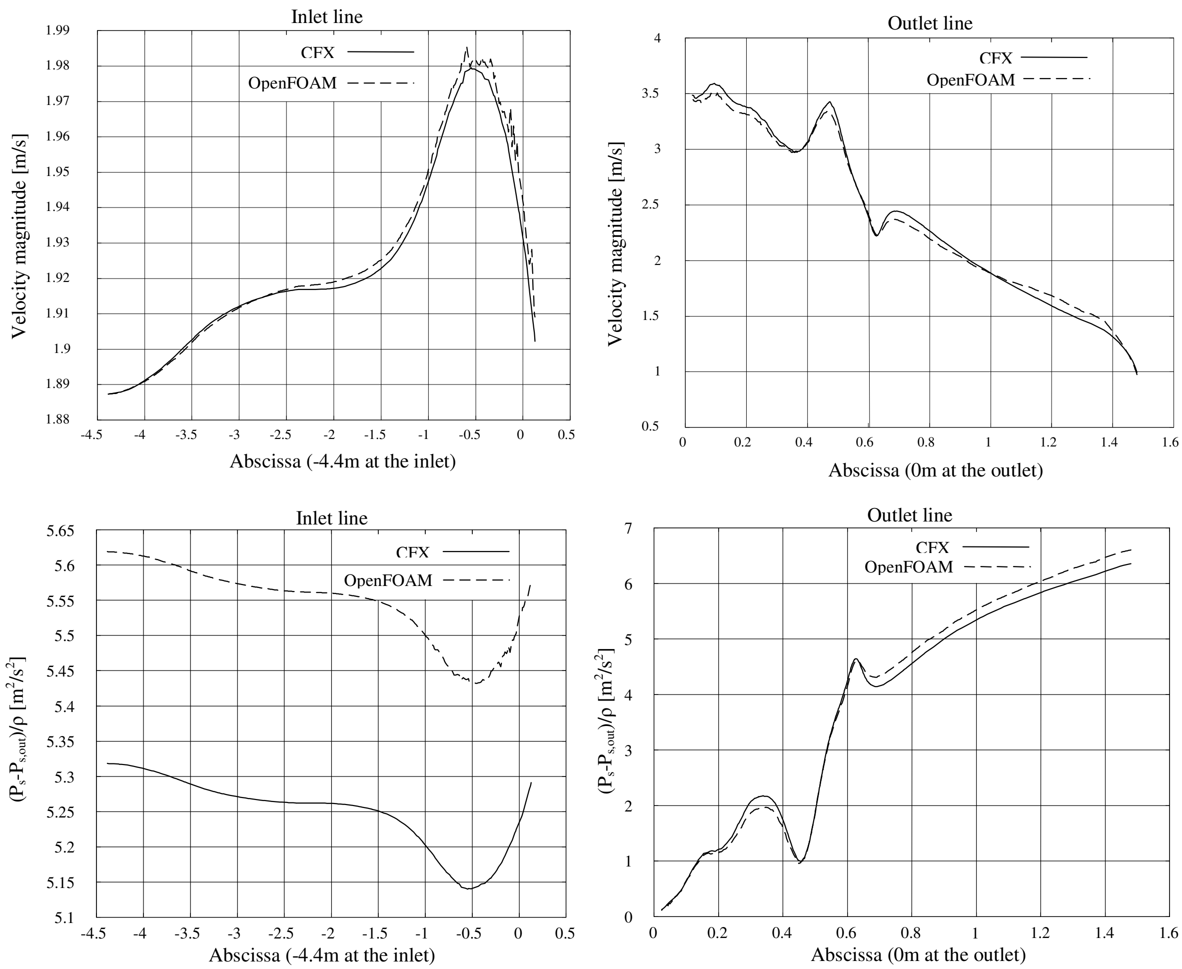

Figure 4 Static pressure, represented by $(P_s\text{-}P_{s,out})/\rho$, and velocity magnitude in the spiral casing and distributor, computed by CFX and OpenFOAM. Note that the difference in the static pressure at the inlet line is amplified by the choice of y-axis scaling. The difference at the inlet is similar as that at Abscissa ~1m along the outlet line.

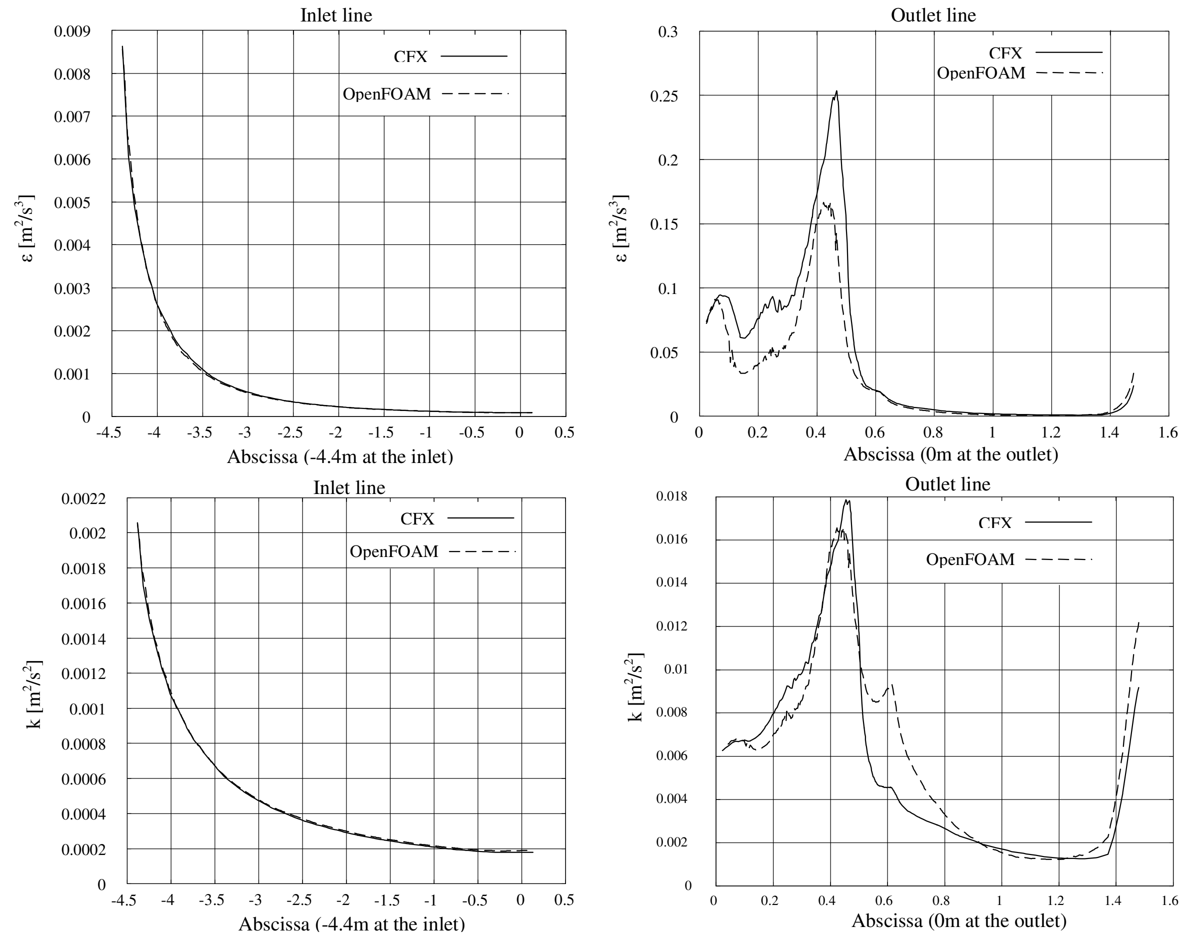

Figure 5 Turbulence properties in the spiral casing and distributor computed by CFX and OpenFOAM

The Draft Tube

Here the results from the CFX and OpenFOAM simulations of the flow in the U9 draft tube are compared. The comparisons are made with respect to the distribution of the static pressure along the upper centerline, shown in Figure 6, and contours of the static pressure and velocity magnitude at the center plane and at a cross-section of the draft tube.

Figure 6 shows the difference in predicted static pressure from CFX and OpenFOAM along the upper center line. The static pressure is set to zero at the outlet. The results from CFX and OpenFOAM are almost identical.

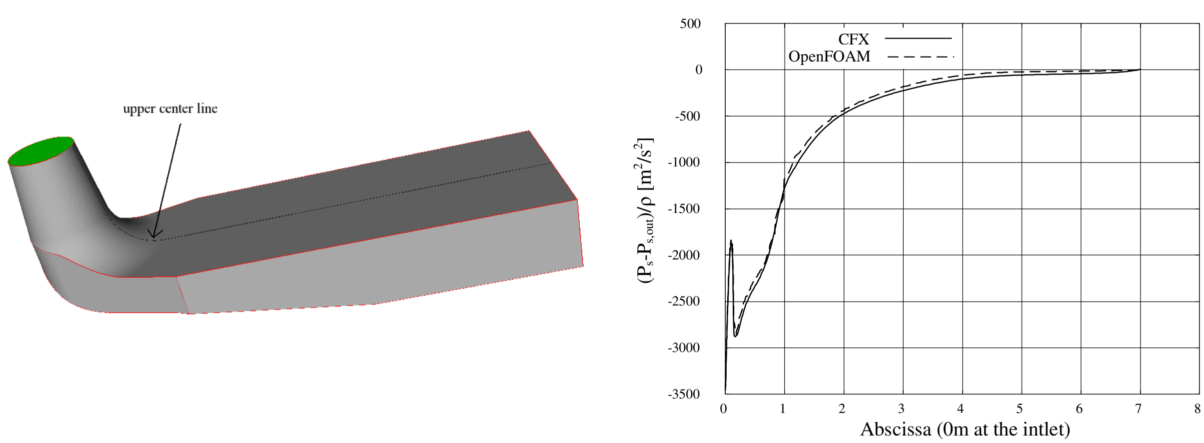

Figure 6: Static pressure computed by CFX and OpenFOAM along the upper center line

Figure 7 compares the CFX and OpenFOAM results with respect to the static pressure distribution at the center plane. The results are almost identical. It can be seen that the static pressure is reduced in the center of the inlet vortex, and that the static pressure gradient is responsible for re-directing the flow in the elbow. It can also be seen that the static pressure is recovered (increased) as the water flows through the draft tube.

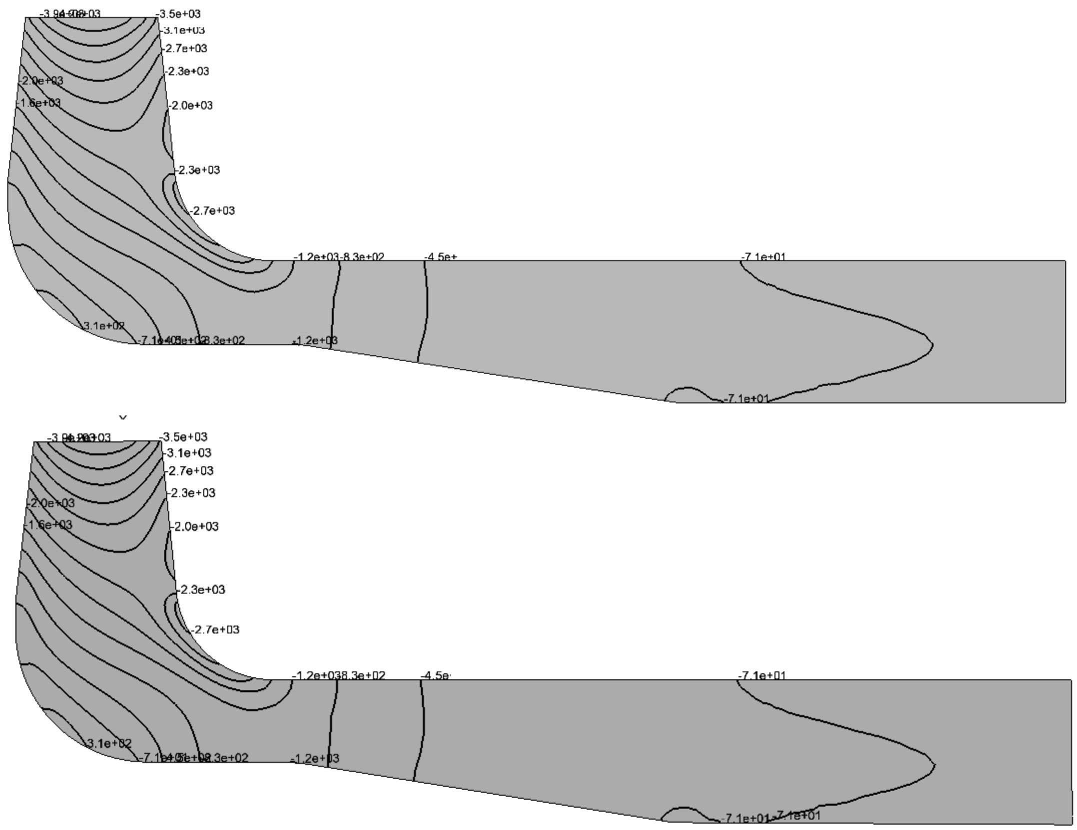

Figure 7: Static pressure contours computed by CFX (top) and by OpenFOAM (bottom)

Figure 8 shows contour lines of the velocity magnitude at the draft tube outlet. The results from the two codes are identical.

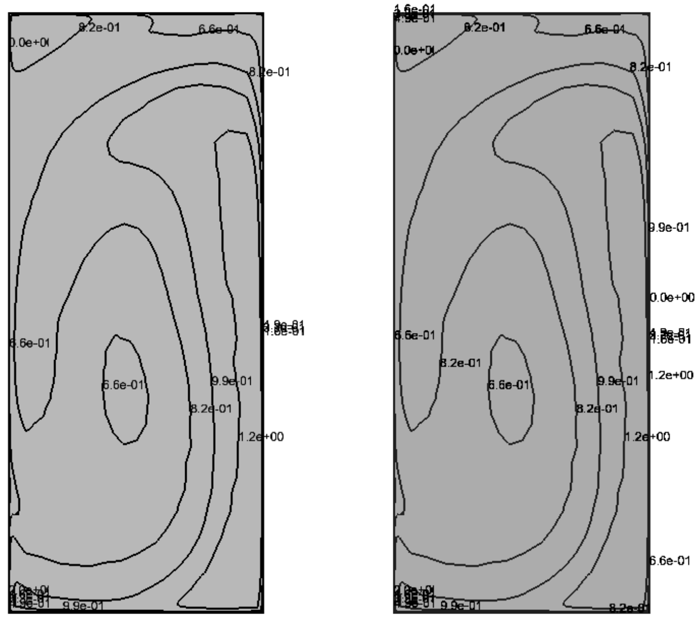

Figure 8: Velocity contours at the draft tube outlet, computed by CFX (left) and by OpenFOAM (right). The view is in the streamwise direction, and the vertical upward direction is to the right.

FUTURE WORK

The next phase of the project consists of comparisons with the results from an on-going experimental investigation of the U9 model. There will be detailed comparisons at the inlet of the spiral casing, in the spiral casing, in the distributor, in the runner, and in the draft tube.

The results presented in this paper are restricted to one unstructured grid for each of the two cases. Block-structured grids are usually used when computing the flow in water turbines in order to increase the accuracy of the results. A fully structured grid for the spiral casing and distributor case has been generated at Chalmers, and the same comparisons will be made using that grid. Furthermore, a grid resolution study should be made.

Also the runner, and the coupling between the rotating and non-rotating parts of the system, will be studied. For this to be possible in OpenFOAM there is a need for some implementations and validations of those implementations.

When it has been verified that both CFD codes give similar results as the experimental results, the main aim is to study the influence of different boundary conditions. Initially the effect of a 90 degree bend before the inlet of the spiral casing will be studied. Such a 90 degree bend is present in the U9 model and prototype, and it is important to investigate how this will influence the details of the flow throughout the turbine.

ACKNOWLEDGEMENTS

We would like to acknowledge the Swedish Water Power Center, SVC, which financed the present work. SVC has been established by the Swedish Energy Agency, ELFORSK and Svenska Kraftnät together with Chalmers University of Technology, Luleå University of Technology, Uppsala University and the Royal Institute of Technology.

We would also like to acknowledge the OpenFOAM community, and in particular Maryse Page and Martin Beaudoin, Hydro Quebéc, in the OpenFOAM Turbomachinery Working Group[3], for fruitful exchange of best-practice guidelines, and for sharing OpenFOAM implementations.

The OpenFOAM simulations in this work has been made on clusters financed by SNIC, the Swedish National Infrastructure for Computing, whom we greatfully acknowledge.

REFERENCES

[1] H.NILSSON AND M.PAGE, December 8-9 2005, *OpenFOAM simulation of the flow in the Hölleforsen draft tube model*, Turbine-99 III, Porjus, Sweden
[2] H.JASAK, H.G.WELLER, AND A.D.GOSMAN, 1999, *High resolution NVD differencing Scheme for Arbritarily Unstructured Meshes*, International Journal for Numerical Methods in Fluids, p.431-449
[3] H.NILSSON, M.PAGE, M.BEAUDOIN, B.GSCHAIDER, H.JASAK, October 27-31 2008*, The OpenFOAM Turbomachinery Working Group, and conclusions from the turbomachinery session of the third OpenFOAM workshop*, 24th IAHR Symposium, Foz do Iguassu, Brazil