NUMERICAL SIMULATION OF THE FLOW IN THE DRAFT TUBE OF THE KAPLAN TURBINE

Nicoleta Octavia TANASE[1], Florentina BUNEA[2], Gabriela Dan CIOCAN[3]

In this paper is presented the numerical simulation of the flow through a known Kaplan turbine geometry (Turbine-99) using the OpenFOAM program. Flow in a draft tube turbine is characterized as a complex turbulent flow with coexistence of different flow phenomena: separation, unsteadiness, swirling flow.

The numerical simulation of the flow in the draft tube still remains a challenge. A classical test case was chosen: Turbine-99 workshops test case to make a simulation of the draft tube flow. The OpenFOAM software was used to perform this simulation. The main results are presented in this paper.

Keywords: draft tube, numerical simulation, OpenFOAM.

1. Introduction

The draft tube of a hydraulic turbine is the component where the flow exiting the runner is decelerated, by converting the excess of kinetic energy into the static pressure. The power output of a low head hydraulic turbine is especially affected by the performance of its draft tube. There is a potential of improving the pressure recovery in the draft tube by modifying and optimizing its geometry.

Turbine-99 draft tube test case is an application challenge of *The Turbomachinery Internal Flow* section of QNET-CFD Thematik Network. Three Turbine-99 workshops had already been organized in 1999, 2001 and 2005, in which the geometry and experimental data were provided as boundary conditions to the participants. A model of Kaplan turbine was mounted in VUAB's turbine rig at the Älvkarleby laboratory, Sweden. Turbine-99 draft tube is an elbow sharp-heel draft tube developed for use in Kaplan turbines. Draft tube geometry, given by workshop organizers, is fixed as well as defined cross sections [1]. OpenFOAM-1.5-dev with the library OpenFOAM Turbo was used to simulate the fluid flow in this Kaplan draft tube.

[1] Teaching Assistant, PhD Student, Dept. of Hydraulics, Hydraulic Machinery and Environmental Engineering, University POLITEHNICA of Bucharest, Romania
[2] PhD Eng., National Institute for R&D in Electrical Engineering ICPE-CA Bucharest, Romania
[3] Associate Prof., Université Laval, Laboratoire de Machines Hydrauliques, Pavillon Adrien-Pouliot, Canada.

Using the given geometry and boundary conditions, in this paper has been developed a flow simulation for this known and documented test case.

2. Simulation and numerical method

OpenFOAM (Field Operation and Manipulation) [2], a numerical simulation software for fluid mechanics, is designed to solve complex physics problems. OpenFOAM is an open - source simulation package which is freely available under the GNU General Public License (GPL). It consists of a vast C++ library, many different applications and additional tools. Although most of the existing applications are flow solvers, OpenFOAM can be used in many different areas, as varied as fluid and solid dynamics, electromagnetics or pricing of financial options.

OpenFOAM implements several modeling paradigms (Finite Volume, Finite Element, Lagrangian Particle Tracking, Finite Area) in library form using object - oriented design, handles complex geometries through polyhedral mesh support, automatic mesh motion and topological changes.

For visualization the simulations of OpenFOAM we used ParaView. ParaView is an application designed for data parallelism on shared – memory or distributed - memory multi computers and clusters. It can also be run as a single computer application.

For the present computations the simpleFoam solvers was used as a base. SimpleFoam is a steady – state solver for incompressible, turbulent flow of non – Newtonian fluids. It is a finite volume solver using the SIMPLE algorithm for pressure – velocity coupling [3].

3. Geometrical model and computational grid

The main geometric dimensions of the draft tube are shown in Fig. 1.

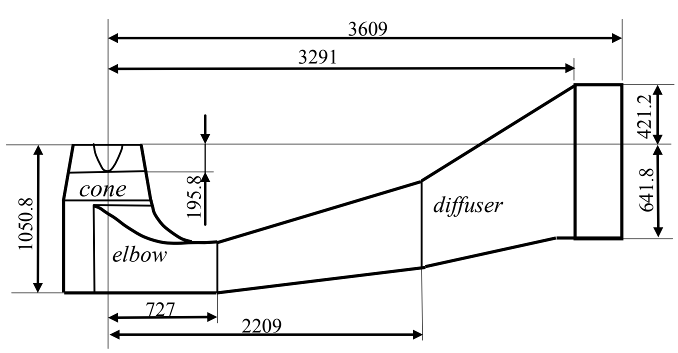


Fig. 1 Geometrics parameters of draft tube

Coordinate system: The x-axis is pointing in the downstream direction towards the outlet of the draft tube. The z-axis is pointing upwards the runner and the y-axis is pointing to the right when watching the draft tube from the outlet (Fig. 2).

For the numerical analysis the geometry is discretized which leads to a computational grid consisting of 1 044 146 grid points and 1 022 854 hexahedral elements, see figure 2.

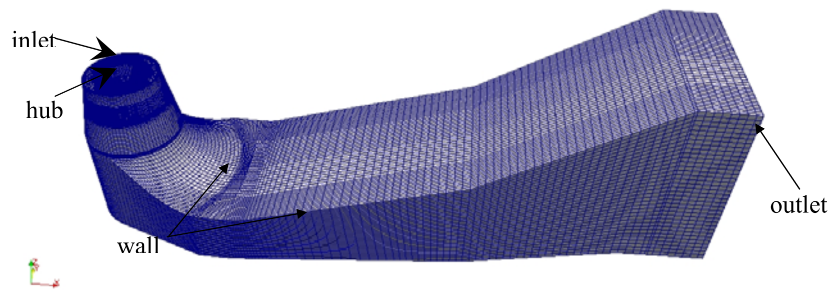


Fig. 2 The draft tube computational domain and grid

4. Boundary conditions

Following Turbine-99 Workshop III, [4], [5], data are supplied for the operational mode T conducted at 60 % load of the Kaplan turbine, which is close to the best efficiency for the system and at the test head ($H = 4.5$ m). The mode is on-cam, i.e. the top-point (T) on the propeller curve (single runner blade angle curve). The settings are:

- runner speed: $N = 595$ rpm (rotation per minute),
- flow rate: $Q = 0.522$ m^3/s ,
- unit runner speed: $DN/\sqrt{H} = 140$, where D is the runner diameter in meter,
- unit flow: $Q/D^2\sqrt{H} = 1.00$,
- water temperature: $t = 15$ °C.

The inlet boundary conditions were obtained from a linear interpolation of the measurements along a diameter at inlet section. They are represented with velocity profiles obtained by extensive LDV measurements for *u, v, w* and *k*. At the outlet boundaries a constant pressure of $p = 0$ Pa is used [4].

Inlet boundary conditions for turbulence models based $k-\varepsilon$ turbulence quantities were calculated from measured Reynolds stresses components [5].

$$k = \frac{1}{2}\left(u_x'^2 + v_y'^2 + w_z'^2\right), \tag{1}$$

$$\varepsilon = \frac{C_{\mu}^{\frac{3}{4}} \cdot k^{\frac{3}{2}}}{l_{\varepsilon}}; \quad l_{\varepsilon} = 0.1\left(R_{wall} - R_{cone}\right). \tag{2}$$

In the Fig. 3 are shown the resulting boundary conditions profiles applied at the inlet section.

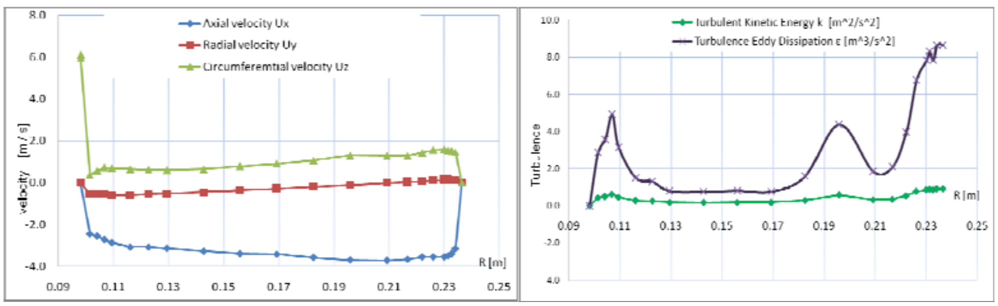


Fig. 3 Inlet boundary conditions

5. Convergence

To check the local convergence, not only the global convergence was followed, but were chosen seven points in different sections inside de draft tube and was represented the variation of parameters depending on the number of iterations. The variation charts are shown in Fig. 4. The residuals changes rapidly in the initial phase of the computation. However, later they become constant.

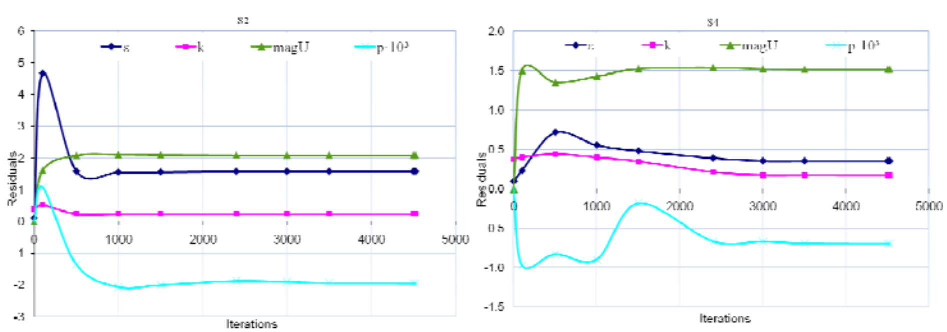


Fig. 4 Evolution the parameters in the computing points

6. Computational results

The Turbine 99 program was provided with LDV measurements in many sections - see Fig. 5.

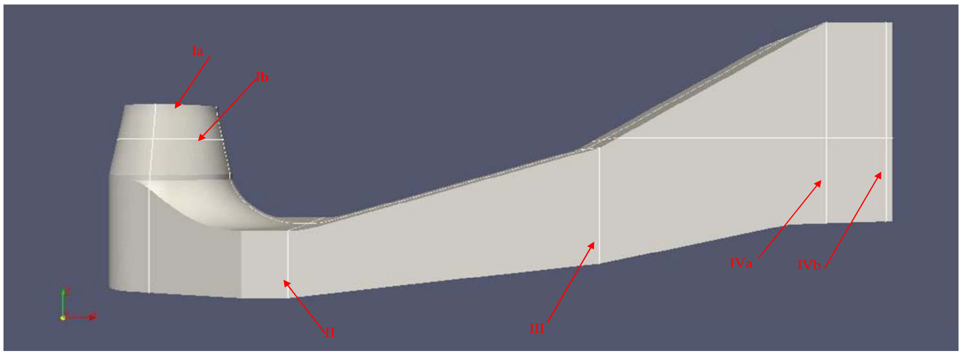
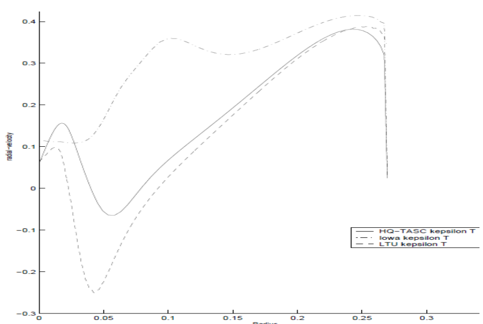


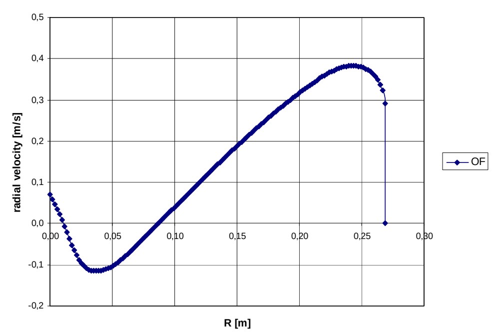
Fig. 5 Available reference sections

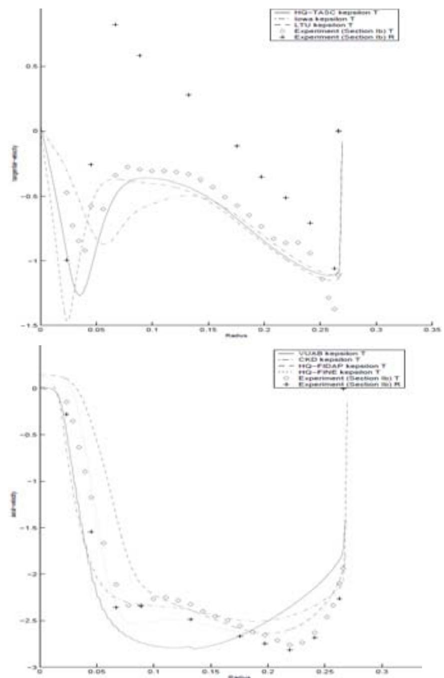
The section Ib and IVa was chosen as representative to validate the present calculation.

In the cross section Ib we obtained a velocity distribution in line with other calculation results presented in literature – Fig. 6. In

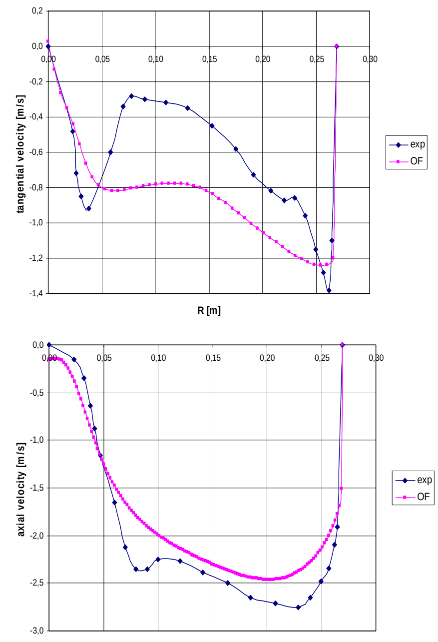
Fig. **7** are presented the main flow and the secondary flow compared with other results in the section Ib. The global flow is well predicted. The difference in the tangential velocity distribution can come from the lack of accuracy of the radial component uses as boundary condition.





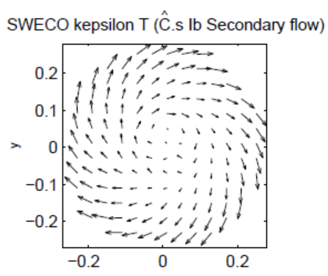


Others calculations [7][8]

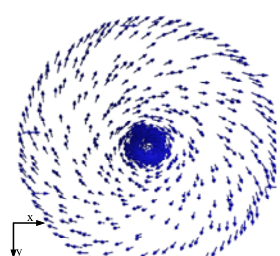


Open Foam (OF) calculation results versus experimental results

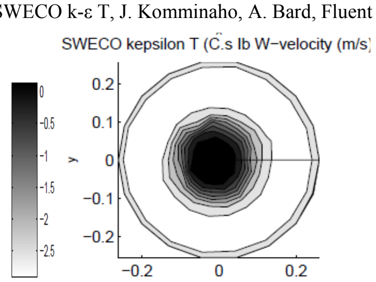
Fig. 6 Comparison of numerical results with experiment and others numerical results of literature



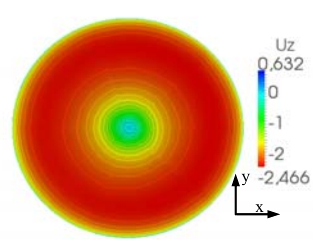
SWECO k-ε T, J. Komminaho, A. Bard, Fluent [8]



OpenFoam



SWECO k-ε T, J. Komminaho, A. Bard, Fluent [8]



OpenFOAM

Fig. 7 Comparison of numerical results with experiment and others numerical results of literature

In the section IVa and IVb we find the classical behavior of the draft tube. The flow privileges the left side and a recirculation is also observed on this side – see Fig. 8.

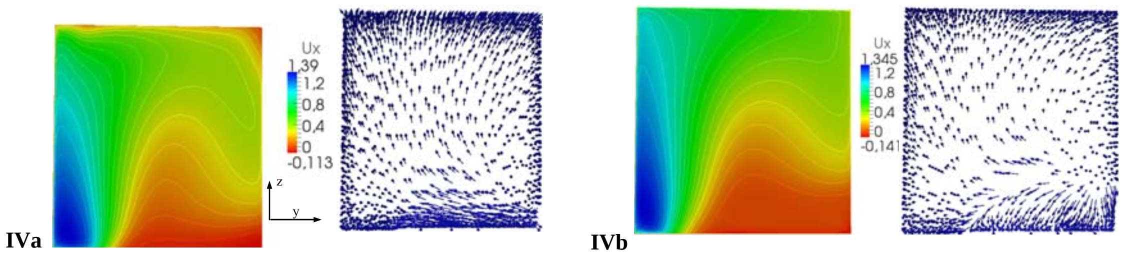


Fig. 8 Velocity field in sections IVa and IVb

In the cross section is observed the classical behavior of the flow in the draft tube – the velocity magnitude is decreasing in the flow direction – see Fig. 9.

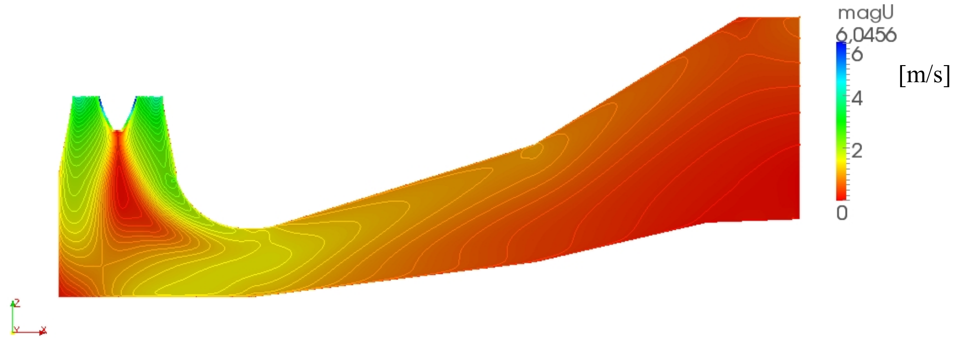


Fig. 9 Velocity contours at draft tube symmetry plane

Pressure distributions

In Fig. 10 are visible the zones of the low pressure at the starting part of draft tube. This low pressure increases the flow rate and consequently the output power of the turbine. At the draft tube elbow is a large increase off static pressure due to the fluid flow deceleration in that region. After that, the draft tube elbow pressure slowly grows to its final value at the outlet section. The major part of the draft tube recovery is observed in the draft tube cone.

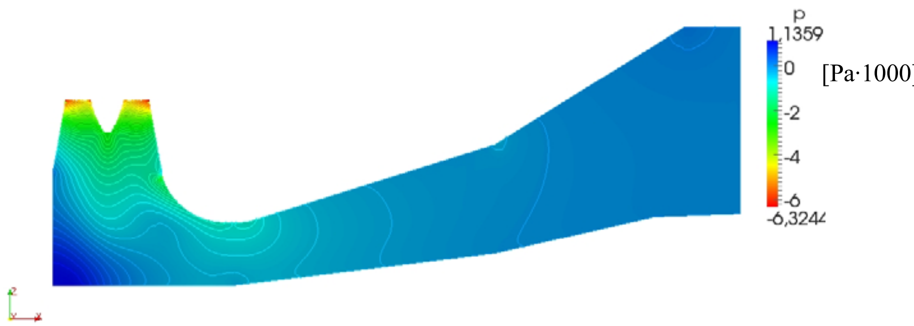


Fig. 10 Pressure variation between section 1 and the end the draft tube

7. Conclusions

In this paper was presented numerical simulation of the flow in the draft tube of the Kaplan turbine, using the OpenFOAM -1.5-dev. The test case of a draft tube of a Kaplan Turbine of the Turbine-99 was simulated.

The results are compatibles with the state of art presented in literature. The general representation of the flow is well catch by the flow simulation. However to use the CFD for quantitative analyses in term of efficiency or local behavior, more accurate inlet condition and turbulence models are needed.

8. Acknowledgment

This paper has been elaborated with the support from Executive Unit for Financing Higher Education, Research, Development and Innovation (UEFISCDI), IDEAS program, contract no. 705/2009, ID_1701.

R E F E R E N C E S

[1]. Turbine-99 Workshop III – Description.
[2]. OpenFOAM, User Guide, June 2011.
[3]. *H. Nilsson*, "Evaluation of OpenFOAM for Turbulent Flow in Water Turbines", in 23rd IAHR Symposium, Yokohama, Oct. 2006.
[4]. *M. Page, M. Beaudoin and A. M. Giroux*, "Steady-State Capabilities for Hydroturbines with OpenFOAM", in 25th IAHR Symposium on Hydraulic Machinery and Systems, IOP Conf., Series: Earth and Environmental Science 12, 2010.
[5]. T.F. Engström, L. H. Gustavsson, R. I. Karlsson, "The second ERCOFTAC Workshop on Draft Tube Flow", Proceedings of Turbine-99 – Workshop 2 on draft tube flow, Älvkarleby, Sweden, 18-20 June, 2001.
[6]. *Z. Čarija, Z. Mrša and L. Dragović*, "Turbulent Flow Simulation in Kaplan Draft Tube", in 5th International Congress of Croatian Society of Mechanics, Trogir / Split, Croatia, Sept. 2006.
[7]. *** Numerical Simulations and Experiments Turbine 99, ERCOFTAC Workshop on Draft Tube Flows, Porjus Hydro Power Center, Sweden, June 20-23, 1999
[8]. "Results from experiments and simulations for the Turbine-99"– Workshop 2.