Industrial Engineering Journal
ISSN: 0970-2555
Volume : 53, Issue 4, No. 6, April : 2024

ANALYSIS OF THE KAPLAN TURBINE DRAFT TUBE EFFECT

Mr. Govind Kumar Thakur, 20ME84CL, Student, B. Tech. Mechanical Engineering (Evening), School Of Mechanical Engineering, Lingayas Vidyapeeth, Faridabad, Haryana.
Mr. Krishna Dev Kumar, 20ME75CL, Student, B. Tech. Mechanical Engineering (Evening), School Of Mechanical Engineering, Lingayas Vidyapeeth, Faridabad, Haryana.

Abstract
The aim of this paper is to present information about possible problems and errors which can appear during numerical analyses of low head Kaplan turbines with a view to the runner - draft tube interaction. The setting of numerical model, grid size, used boundary conditions are the interface definition between runner and draft tube are discussed. There are available data from physical model tests which gives a great opportunity to compare CFD and experiment results and on the basis of this comparison to determine the approach to the CFD flow modeling. The main purpose for the Kaplan turbine model measurement was to gather the information about real flow field. The model tests were carried out in new hydraulic laboratory of CKD Blansko Engineering.
The model tests were focused on the detailed velocity measurements downstream of the runner by differential pressure probe and on the velocity measurement downstream of the draft tube elbow by Particle Image Velocimetry method (PIV). The data from CFD simulation were compared to the velocity measurement results. In the paper also the design of the original draft tube modification due to flow improvement is discussed in the case of the Kaplan turbine uprating project. The results of the draft tube modification were confirmed by model tests in the hydraulic laboratory as well.

Keywords:
draft tube interaction, Particle Image Velocimetry method (PIV), uprating, flow modeling, pressure probe, Modelling, CFD.

1. Introduction

Draft tube is very important component of Kaplan turbines due to high amount of kinetic energy leaving the runner and entering the draft tube. As was presented by many papers [Lit], there is still problem to simulate reasonably the real flow in the draft tube with elbow. Many papers are focused on the numerical simulation of turbulent flow at individual draft tube separately [Lit].
The results of such simulation are very sensitive to the inlet boundary condition, which is represented by profile of velocity components and by turbulence parameters. Although flow in the draft tube is almost always unsteady, there is still tendency to substitute expansive unsteady flow simulation by steady state one.
The reason is requirement of fast response especially in case of designing the new runner for existing draft tube. Such need is very important in the process of upgrading projects of existing hydropower plants.
The previous experience shows the coupled runner-draft tube flow simulation as proper technique for fast and reasonable results. However, the verification of the CFD computational methodology is necessary. The acceptable verification can be done based on the detailed real flow measurement on the Kaplan turbine model in the hydraulic laboratory only.

2. Measurements on the model turbine

The Kaplan turbine model consists of a semi spiral case, four-blade runner type and the draft tube type with elbow. The four blade Kaplan runner has diameter 350 mm The model was operated at net head 5 m. The inlet energy was measured at the spiral case inlet while outlet energy was measured at the draft tube outlet. The model was adapted for the detailed flow field measurement at two different locations. In order to evaluate the flow quality in the individual components like the runner and the draft tube. The first section was located downstream of the runner and second one downstream of the draft tube elbow.

2.1 Measurement of the velocities downstream of the runner

The five holes differential pressure probe made by United Sensors Corporation type DA-187-24-H-22-D was primary calibrated in the special hydraulic circuit. The calibration was carried out for pitch and yaw angle as well and therefore the probe was during measurement only shifted to the radial position and should not be rotated in order to balance pressures. The resulted calibration diagram was digitalized and directly simply used for velocity components evaluation based on the measured pressure difference. The probe was located in the draft tube cone below the runner blades. The axis of the probe was located perpendicular to the axis of the runner (Fig.1). As the result of a measurement the axial, radial and tangential velocity components as well as the static pressure were evaluated. The values were time averaged and are shown as a function of the radius.

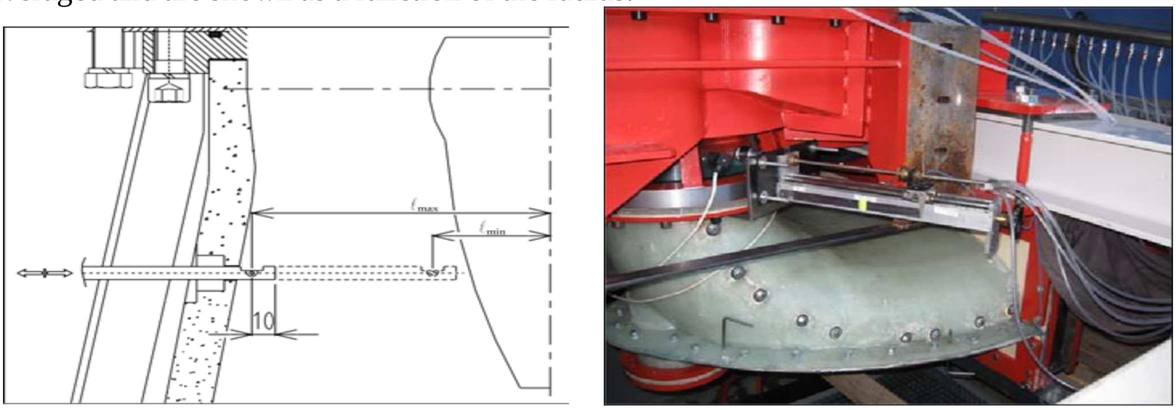


Fig. 1 Differential pressure probe location and measurement setup

2.2 Measurement of the velocities downstream of the draft tube elbow

Particle Imaging Velocimetry (PIV) is a kind of fluid visualization method allowing the evaluation of the instantaneous velocity vectors in the measurement plane. In this case it is used to obtain time averaging velocity distribution and related properties in fluids. At the connection between laminate draft tube elbow and steel draft tube liner the special transparent component was mounted.
The measurement in this section has been done at 8 light planes which are parallel to the flow direction. The pulse laser was mounted below the draft tube elbow on the ground while the high speed camera was located at the draft tube side (Fig.2). On the instantaneous snapshot of the planar velocity vectors at plane 2 the vortex can be seen as an consequence of the flow character at the vertical pier (Fig. 3). The main result of this measurement was evaluated by averaging of the instantaneous velocities (Fig. 4) and by matching all planes together. Finally the normal velocities were evaluated and used for CFD verification.

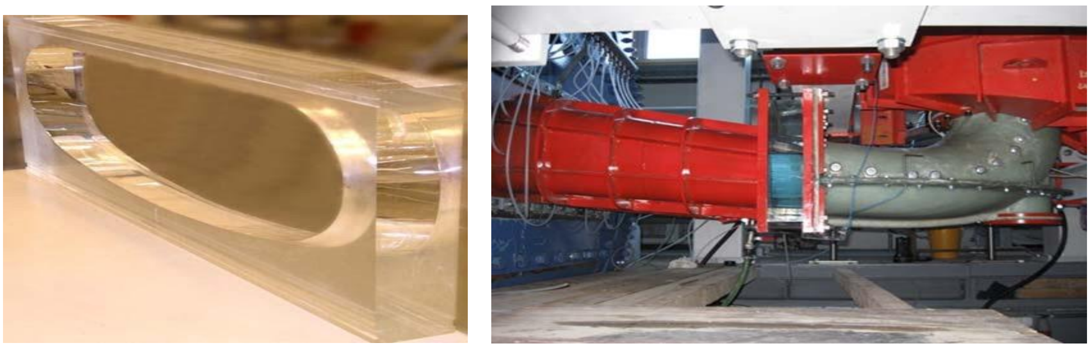
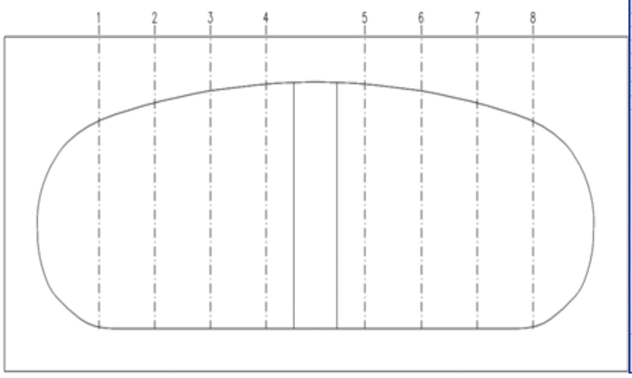


Fig. 2 The plexi-glass segment and model assembly at the laboratory



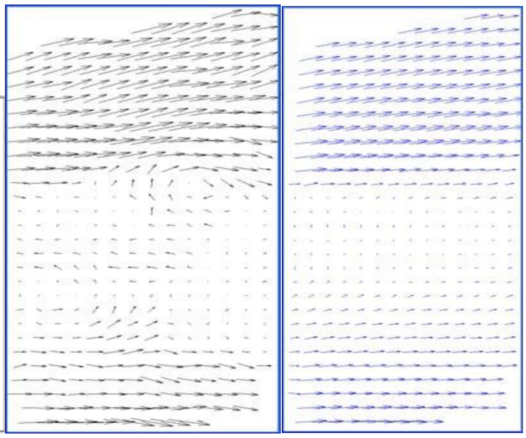


Fig. 3 Location of the planes at the section plane 2

Fig. 4 Instantaneous and averaged vectors in

3. Verification of the CFD computational methodology

The accuracy of the CFD modelling is still depending on many factors concerning in the computational methodology. This is the complex of solver settings, modelling properties, boundary conditions and simplifications of the computational domain. It is not possible to determine the computational topology generally for all CFD tasks and therefore it is necessary to verify the methodology individually. The verification has been carried out on the computational model of the Kaplan turbine runner coupled with the draft tube having both horizontal and vertical pier.

The verification process started with high degree of model simplification. The reason was to shorten time for the calculation as much as possible. Only one segment of the runner was modelled and the gap between blade and discharge ring as well as the gap between blade and the hub was neglected. At the inlet of the domain the constant radial and tangential velocity components were defined. The Ansys CFX commercial software with SST turbulence model was used for the analysis. Flow is modelled as steady state and the circumferential averaging interface is placed between stationary and rotating parts (stage simulation). The runner segment contains 250.000 cells and the draft tube contains about 600.000 cells.

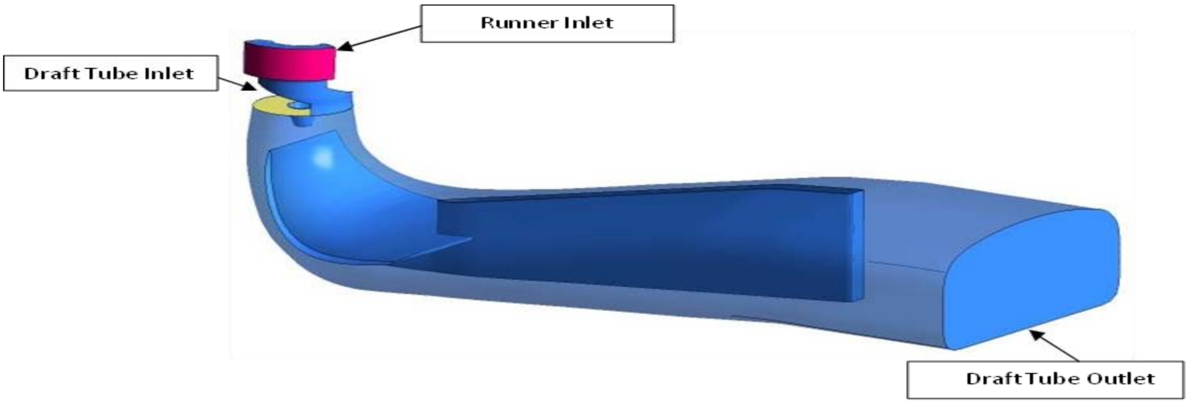


Fig. 5 Computational domain of the coupled runner with draft tube

In the first phase the verification was focused on the velocities downstream of the runner. Measured and computed velocity components are compared in Fig. 6. The differences were analyzed at axial and tangential velocity components as well. After this establishment the frozen rotor simulation was analyzed but the simulation results did not changed too much. Also improving the grid quality with increased number of cells does not help. The next step was to increase the quality of the results by putting away used simplification. The full runner was modelled and unsteady simulation used, however the expansive computation did not bring any noticeable improvements.

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 53, Issue 4, No. 6, April : 2024

Next the simulation of flow in the spiral case concerning stay and guide vanes has been carried out in order to find real runner inlet conditions. The evaluated velocity components downstream of the guide vanes show effect of the unsymmetrical semi spiral case. The higher velocities are located at lower discharge ring. The analyzed velocity profile was used for inlet boundary conditions at runner inlet.
The CFX stage computation with SST turbulence modelling and application of real inlet profile was carried out., the accuracy of velocity distribution downstream of the runner improved significantly. The tangential velocity components are almost identical to the measured values. The smaller differences are still in the axial velocities especially near discharge ring, where the velocities are underpredicted.

Fig. 6 Difference between measured and calculated velocity profiles (before verification)

Nevertheless it can be concluded that consideration of constant velocity distribution at the runner inlet can lead to the inaccurate velocity distribution downstream the runner. This conclusion is individual and do not play such importance in case of steel lined spiral case or symmetrical concrete semi-spiral case.
Next changes were done in the domain topology. In all computations above were carried out with simplified domain without any modelling of the real tip clearance between blade and other parts. In the next modelling the tip clearance between blade and discharge ring was modelled in order to check the effect of the clearance to the velocities downstream of the runner. The importance was paid to modelling of tip clearance. There is a necessity to keep number of cells in a tip clearance above minimum value, otherwise the solver is not be able to model the flow across tip clearance well. After application of above mentioned conditions, the results match to the model test results better especially at the proximity of discharge ring as was expected The real inlet conditions was used at this computation as well.
After successful verification of the velocity profile downstream of the runner the research continues with verification of the flow modeling at the draft tube elbow. The results from PIV measurement were used and the normal velocities at the section downstream of the draft tube elbow were compared.. In case of real flow (PIV results) there is a significant wake downstream of the vertical pier , low velocities at the upper part of the section but not any back flow there (Fig.7) . Again, the results obtained before and after verification were compared to the measured values. The velocity field before verification shows higher non uniformity then measured (Fig.8). The results after verification are much closer to the velocity
In fig 9

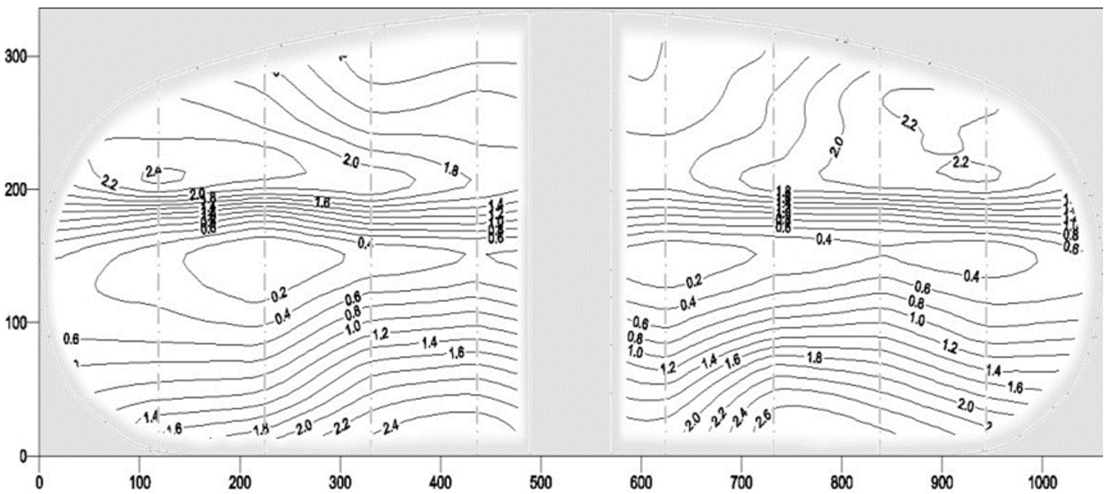


Fig. 7 Normal velocity contours obtained by PIV velocity measurement

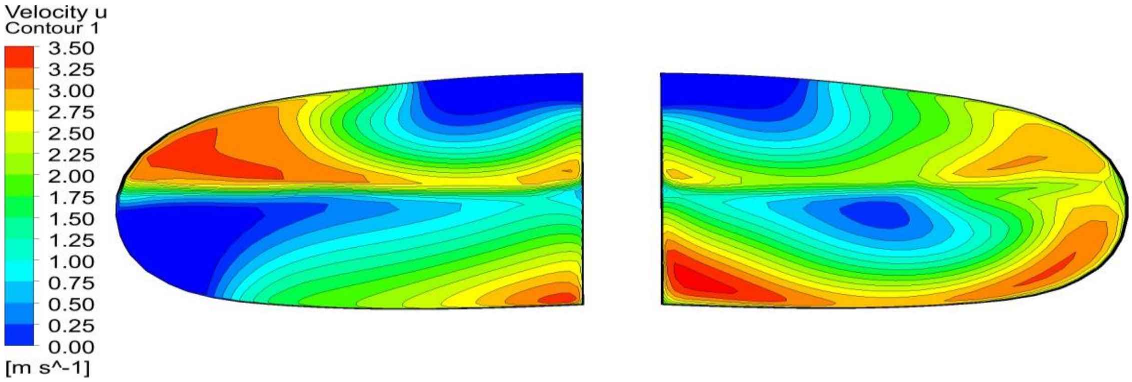


Fig. 8 Normal velocity contours calculated by CFD before verification

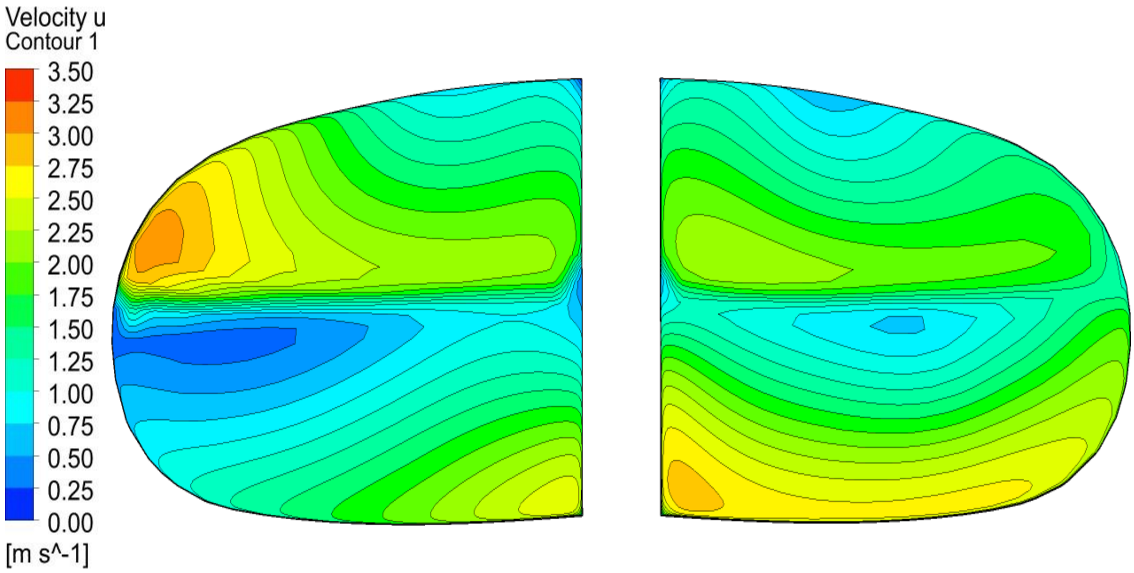


Fig. 9 Normal velocity contours calculated by CFD after verification

4. Modification of the draft tube geometry

The Kaplan turbines uprating is a common task of nowadays. In such cases the uprated turbine with the new runner in combination with the original draft tube should operate at higher output. In some cases the shape of the original draft draft tube is unsuitable and it is necessary to change draft tube hydraulic profile. Thereinafter the possibility of increasing of turbine efficiency by draft tube shape modification is presented.

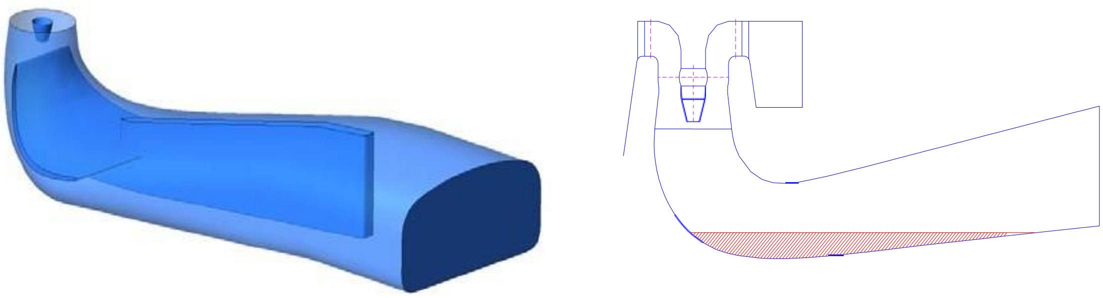
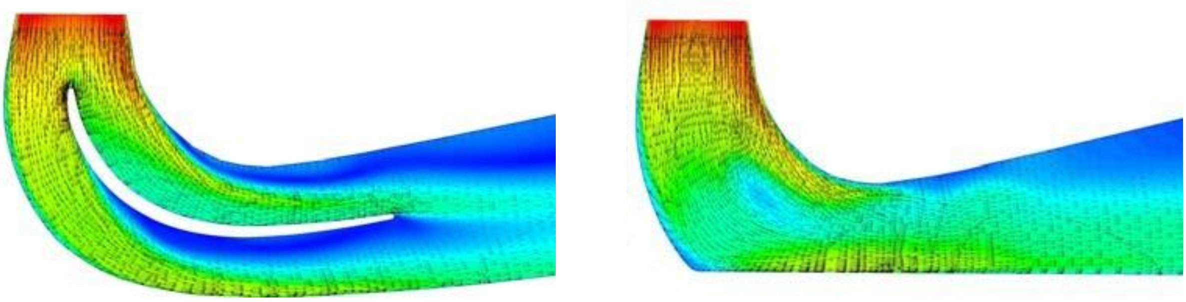


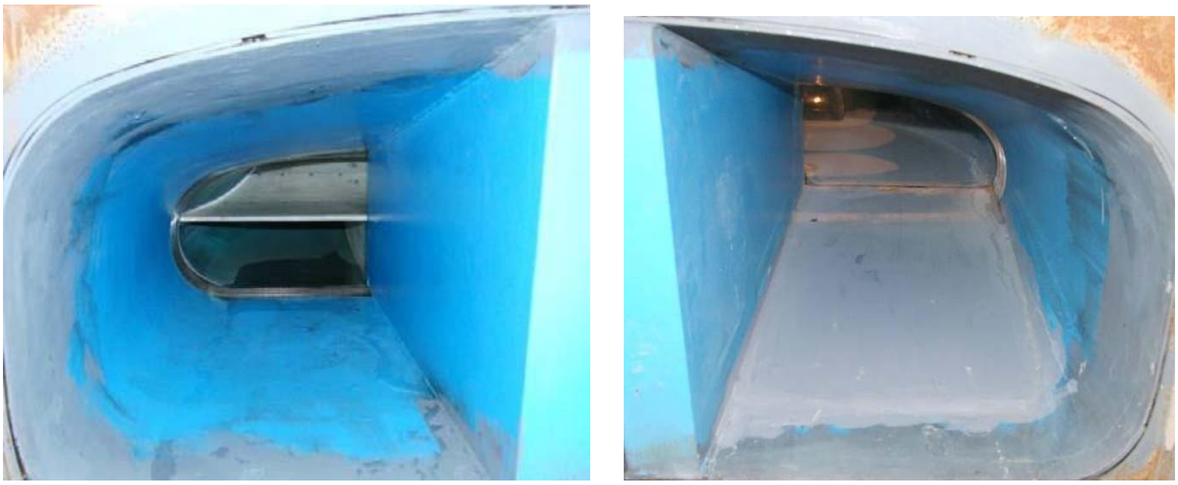
Fig. 10 Geometrical model of original draft tube and sketch of the modification

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 53, Issue 4, No. 6, April : 2024



Original geometry Modified geometry

Fig. 11 Draft tube flow CFD results - contours of velocity magnitude at longitudinal section



Original geometry Modified geometry

Fig. 12 Look into the draft tube physical model

By means of CFD analyses the modification by vertical pier removing and draft tube bottom was designed and flow in the draft tube was improved (Fig. 11). According to the calculations the modification increases the efficiency of turbine approximately by one percent. This assumption was confirmed by model tests when the efficiency characteristics for the original geometry were measured, after that the draft tube were modified and the efficiency characteristics were measured again.

5. Conclusion

During the second phase of the verification of the CFD simulation on the Kaplan turbine model the new investigations have been adopted. The main topic was to find the real flow pattern downstream of the runner and downstream of the draft tube elbow in order to compare it with the CFD results carried out with standard computational methodology approach. The comparison was done for set of operational points, however most important was to compare the best efficiency point and high flow rate point. The discrepancies were found in axial as well as tangential velocity components downstream of the runner. Consequently, the computational methodology has been successfully corrected and the new CFD results fit well the measured velocity components.

The main changes have been done in the inlet boundary condition for the runner inlet and in the modelling of the real gap between runner blade and the discharge ring. Next comparison has been done for velocity pattern downstream of the draft tube elbow. After improvement of the computational methodology the differences of the velocities downstream of the draft tube elbow are reasonable. The minima and maxima of the normal velocities are at very similar locations and consequently the flow in the draft was modeled properly.

The approach for the draft tube improvement has been verified on the older type of the draft tube with vertical and horizontal pier as well. The idea was to improve the draft tube performance by simple removing of the vertical pier and by modification of the draft tube bottom. The CFD design has been proved by measurement on the laboratory model, where the turbine efficiency increased for about 1 % near optimal operation point. There is still need to increase efficiency at the maximum turbine discharge.

The next step has been done in the tip vortex cavitation problem. By using the anti-cavitation lips the effect of the tip vortex cavitation damage on the blade surface was suppressed.

The all conclusions from the verification should be applied on the Kaplan turbine runner design procedure in order to improve the hydraulic parameters of the low head Kaplan turbines. The optimization of the runner geometry by CFD modelling should be focused on the efficiency increase and on the improvement of the cavitation features.

References

[1] Skoták A, Lhotáková L and Mikulášek J 2002 Effect Of The Inflow Conditions On The Unsteady Flow in the Draft Tube XXI IAHR Symp. (Lausanne, Switzerland)
[2] Skoták A and Kopecký J 2004 Considering the tip clearance effect at Kaplan turbine runner blade (in Czech) HydroTurbo conf. (Brno, Czech Republic)
[3] Skoták A and Obrovský J 2006 Shape Optimization of a Kaplan Turbine Blade 23rd IAHR Symp. on Hydr. Mach. and Syst. (Yokohama, Japan) p 233
[4] Skotak A and Obrovsky J 2007 Low swirl flow separation in a Kaplan turbine draft tube 2nd IAHR Int. Meeting of the WorkGroup on Cavitation and Dynamic Problems (Timisora, Romania)
[5] Skotak A and Obrovsky J 2007 Analysis of the flow in the water turbine draft tube in Fluent and CFX 25th CADFEM Users Meeting (Dresden, Germany)
[6] Gagnon J M 2008 Experimental investigation of runner outlet flow in axial turbine with LDV and stereoscopic PIV IAHR 24th Symp. on Hyd. Mach. and Syst. (Foz do Iguazzu, Brazil)
[7] Loiseau F 2008 Importance of draft tube in rehabilitation projects IAHR 24th Symp. on Hydraulic Mach. and Syst. (Foz do Iguazzu, Brazil)
[8] Skoták A, Motyčák J, Obrovský J, Štegner P and Pola J 2010 Sophisticated approach to the Kaplan turbines upgrading HydroVision Russia (Moscow, Russia)