

INFLUENCE OF TAIL WATER LEVEL ON THE HYDRAULIC PERFORMANCE OF ELBOW DRAFT TUBE

VPLIV SPODNJE VODE NA HIDRAVLIČNE KARAKTERISTIKE SESALNE CEVI

Matej Fike[✉], Aleš Hribernik

Keywords: CFD, Elbow draft tube, Numerical analysis, Ansys CFX

Abstract

Numerical simulations of the flow in the elbow draft tube of a Kaplan turbine were carried out in order to elucidate the effects of tail water level on draft tube performance and flow stability. The influence of the tail water level is explored considering the correct boundary condition at the inlet of the draft tube. It is found that the increasing tail water level affects the inlet boundary condition of the draft tube, the flow rate through the turbine, and draft tube. Thus, head recovery, efficiency and the possibility of cavitation phenomena decreases. However, the head loss increases, and the flow rate through the turbine and draft tube decreases.

Povzetek

Izvedene so bile numerične simulacije toka v kolenasti sesalni cevi kaplanove turbine z namenom preučitve vpliva spodnjega nivoja gladine vode na brezdimenzionalne karakteristike in na stabilnost toka v sesalni cevi. Vpliv nivoja spodnje vode je raziskan z upoštevanjem pravih robnih pogojev na vstopu. Ugotovljeno je bilo, da naraščanje spodnjega nivoja vode vpliva na pretok skozi turbino in posledično na vstopni robni pogoj sesalne cevi. Posledično se izkoristek in verjetnost nastanka kavitacije zmanjša. Na drugi strani se hidravlične izgube povečajo, pretok skozi turbino in sesalno cev pa se zmanjša.

[✉] Corresponding author: Matej Fike, Faculty of Energy Technology, Hočevarjev trg 1, SI-8270 Krško, Tel.: +386 7 6202 228, E-mail address: matej.fike@um.si

1 INTRODUCTION

Hydropower plays an important role in the energy sector and helps to meet increasing demand for energy. The hydraulic turbines convert hydro energy into electric energy and, therefore, the increase in efficiency of the turbine is an important task. Each component of turbine has some specific function, and hence the design of every component of the turbine is equally important. However, out of all components, draft tube converts kinetic energy coming out of runner into useful pressure energy that would otherwise be wasted. Therefore, the design of the draft tube is important and contributes to improving the efficiency and overall performance of the turbine.

The main aim of this research was to examine flow conditions within the draft tube of a hydropower plant at different tail water levels using 3D CFD simulation to inspect for possible irregularities within the flow.

Numerical flow simulation has been carried out for steady viscous turbulent flow in a Kaplan turbine assembly consisting of a semi-spiral casing with stay and guide vanes, a 4-blade runner and an elbow draft tube using Ansys CFX commercial CFD code. The $k-\varepsilon$ turbulence model has been used. The head loss, head recovery, and efficiency characteristics of the draft tube have been predicted at low, normal and high-water conditions using simulation results (at three different tail water levels). In addition, velocity streamlines and contours of velocities and pressure at selected planes within the draft tube have been obtained.

2 NUMERICAL MODEL

Commercial CFD software Ansys CFX 16.2 was used for this study. Simulations of two-phase flows can be achieved by assuming the vapor/liquid mixture as a multi-phase single fluid, with variable densities. No slip exists between the two phases in this mixture model. We performed analyses with the $k-\varepsilon$ turbulence model with scalable wall functions.

2.1 Geometry

Three-dimensional geometry of the draft tube was modelled in SolidWorks. The numerical model consists of two main parts: draft tube and draft tube prolongation. The inlet cross-section is a circle with a diameter of 1.6 m. The outlet cross-section is a rectangle 5 m in width and 1 m in height. The length of the draft tube is 8.2 m. Draft tube prolongation (Figure 1) with the free surface between the air and water stream was added to properly simulate the outflow conditions and to avoid any influence of the outflow boundary conditions on the draft tube flow.

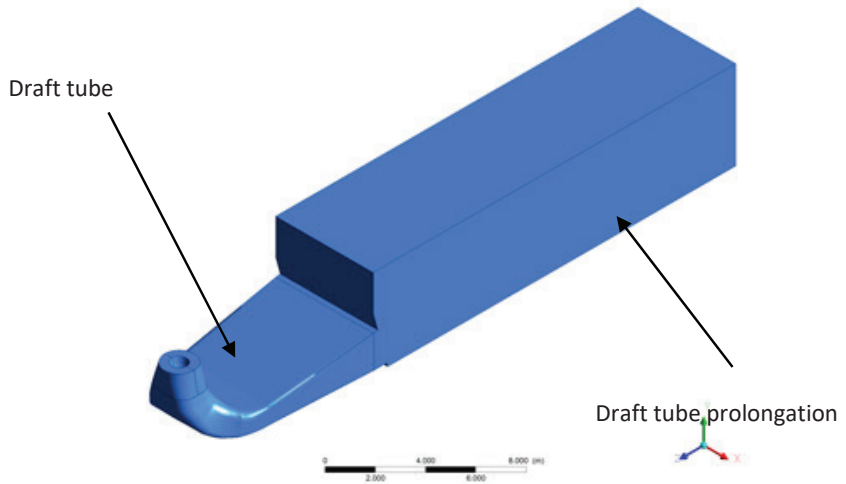


Figure 1: Draft tube model with draft tube prolongation

2.2 Mesh creation

Structured computational meshes were used in all parts and were designed with ICEM CFD 16.0 software. Numbers of elements used are specified in Table 1.

Table 1: Mesh data

Domain	Nodes	Elements
Draft tube	382,328	387,971
Draft tube prolongation	630,350	605,395
All Domains	1,012,678	993,366

Figure 2 shows the structural mesh of a draft tube and a draft tube with prolongation. The resolution of the mesh is greater in regions where greater computational accuracy is needed, such as close to the inlet and close to the walls.

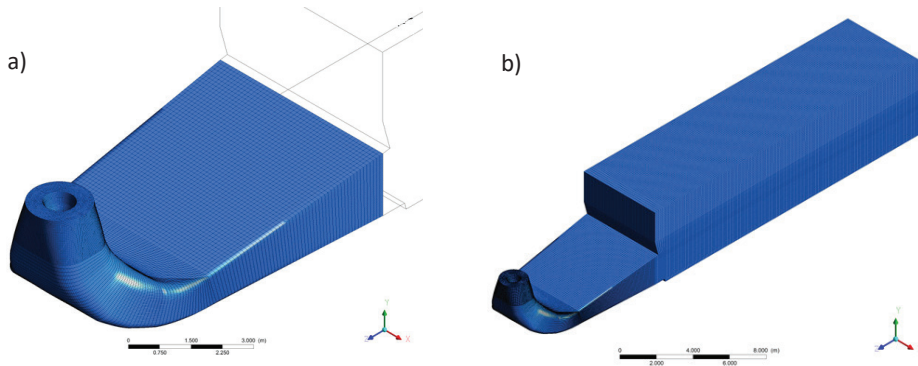


Figure 2: Numerical mesh a) draft tube and b) draft tube with prolongation

2.3 Boundary conditions and convergence criteria

Correct inflow boundary conditions are necessary in order to correctly simulate the flow conditions within the draft tube. These may be prescribed by experimental data or by the application of a turbine model for their prediction. The turbine model was used in our case. A simplified turbine model was used for the optimal set of guide vane and rotor blade angles. Only the guide vane passage and rotor blade passage flows were simulated.

The guide vane passage flow was simulated within the static frame while rotor passage flow was within the rotational frame of reference. The interface boundary condition was applied to connect both frames. Using the periodic boundary condition on both sides of the simulated flow passages, it was possible to simulate only one pair of passages and not the whole turbine.

At the inlet and at the exit of the domain, static pressure was prescribed. This enabled predicting the flow rate and velocity components at the outlet of the rotor for different available heads.

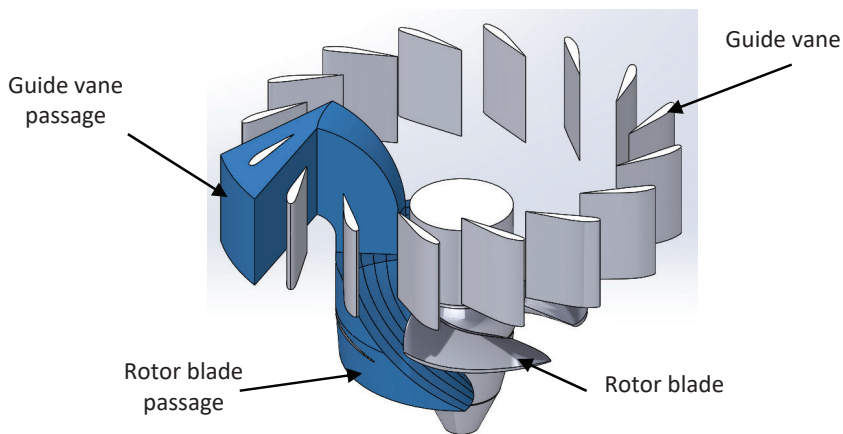


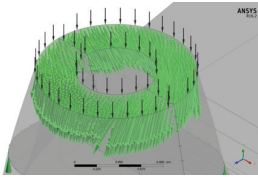
Figure 3: Inlet boundary conditions

Three different inlet boundary conditions were prescribed in order to perform simulations of the flow within the draft tube for low, normal and high-water conditions.

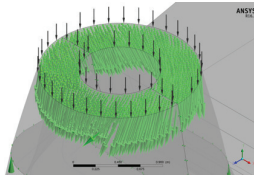
Table 2: Inlet boundary conditions

Water conditions	Low	Normal	High
Tail Water level	H-1m	H	H+1m
Volume Flow Rate	13.2 m ³ /s	13.6 m ³ /s	14.1 m ³ /s
Head	5.7 m	6.7 m	7.7 m

Low water condition



Normal water condition



High water condition

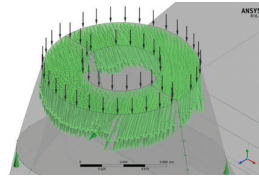


Figure 4: Inlet boundary conditions

Although the flow within the draft tube is almost always unsteady, steady-state simulations were used in order to substitute time-consuming and expensive unsteady simulations. Previous research determined that steady-state simulations assure reliable results.

Draft tube prolongation with the free surface between the air and water stream was added to properly simulate the outflow conditions and to avoid any influence of the outflow boundary conditions on draft tube flow.

The relative hydraulic pressure at the outlet of the draft tube prolongation with the following equation was prescribed (Figure 5).

$$p = g \cdot \rho \cdot VFW \cdot (H - y) \quad (2.1)$$

where:

VFW - volume fraction of water

H - tail water height

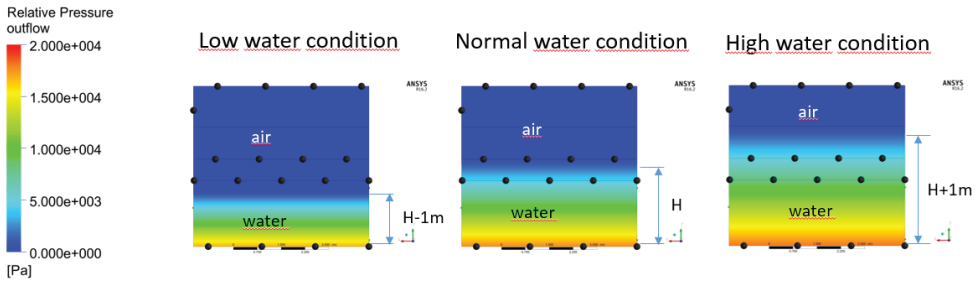


Figure 5: Outlet boundary conditions

To satisfy the convergence criteria, all the RMS (root mean square) residuals from solving equations must be under $1 \cdot 10^{-5}$. We also set the number of maximum iterations to 500 and used automatic timescale control.

3 RESULTS

3.1 Characteristic parameters

The computed results of the flow in the draft tube at different tail water levels were compared with the following non-dimensional hydrodynamic parameters.

Head loss in draft tube

$$H_{LD} = \frac{p_{tin} - p_{tout}}{\rho g} \tag{3.1}$$

Head recovery in draft tube

$$H_{RD} = \frac{v_{in}^2 - v_{out}^2}{2g} - H_{LD} \tag{3.2}$$

Efficiency of draft tube

$$\eta_D = \frac{2gH_{RD}}{v_{in}^2} \times 100 \tag{3.3}$$

where:

- H_{LD} =head loss in draft tube (m)
- p_{tin} =total pressure at draft tube inlet (Pa)
- p_{tout} =total pressure at draft tube outlet (Pa)
- ρ =specific mass of fluid (kg/m^3)
- g =acceleration due to gravity (m/s^2)
- v_{in} =velocity at draft tube inlet (m/s)
- v_{out} =velocity at draft tube outlet (m/s)

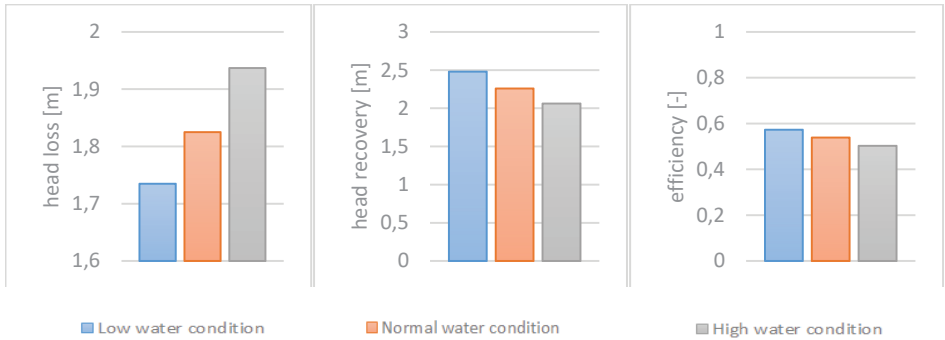


Figure 6: Head loss, head recovery and efficiency for different tail water levels

Diagrams of head loss, head recovery and efficiency are presented in Figure 6. It can be observed that with tail water increases, head loss parameter increases, while head recovery and efficiency decrease.

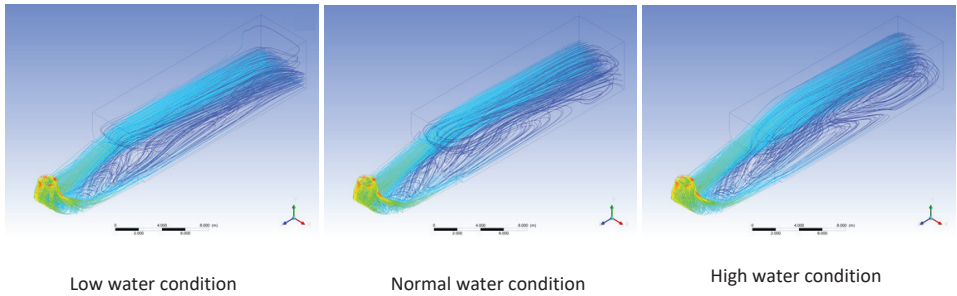


Figure 7: Velocity streamlines

Figure 7 shows velocity streamlines within the draft tube. In the draft tube inflow section, the flow is concentrated near the walls due to small tangential velocity component induced by the runner blade passage. A wake is, therefore, formed in the middle of the stream and it is extended far into the draft tube prolongation. The velocity profile is very asymmetric.

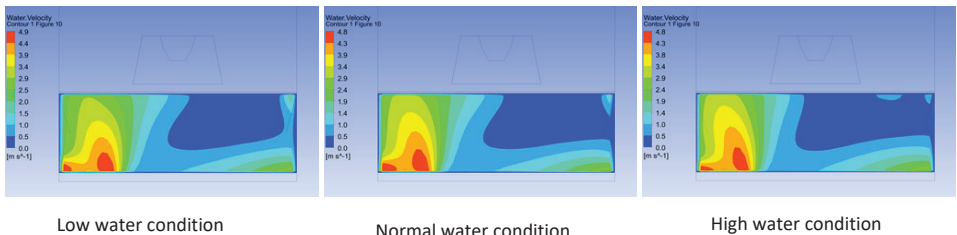


Figure 8: Velocity contours

Velocity contours at the outlet of the draft tube are presented in Figure 8 for three different tail water levels. The high-velocity stream can be seen at the bottom left side of the draft tube with its peak near the bottom in all three cases. All three velocity plots are similar. The difference is in the magnitude of the velocity, which depends on the volume flow rate through the draft tube.

4 CONCLUSIONS

In this paper, numerical simulations of the flow in the draft tube of a Kaplan turbine were carried out to elucidate the effects of tail water level on draft tube performance and flow stability. The tail water level significantly affects the performance of the turbine, when the guide vane and rotor blade angle settings remain unchanged. The influences of the increasing tail water level are as follows:

- the flow rate through turbine and draft tube decreases, and affects the inlet boundary condition of the draft tube
- the head loss increases
- head recovery decreases
- efficiency decreases
- the possibility of cavitation phenomena decreases

References

- [1] **H. Dou, L. Niu, and S. Cao:** *Effects of tangential velocity distribution on flow stability in a draft tube*, Journal of Thermal Science, vol. 23, no. 5, pp. 446–453, 2014
- [2] **H. Ying, C. Heming, W. Quanlong, and Y. Zhikun:** *Computer simulation of turbulent flow through a hydraulic turbine draft tube*, Wuhan University Journal of Natural Sciences, vol. 11, no. 3, pp. 649–652, 2006
- [3] **Z. G. Zuo, S. H. Liu, D. M. Liu, and D. Q. Qin:** *Numerical predictions and stability analysis of cavitating draft tube vortices at high head in a model Francis turbine*, Science China Technological Sciences, vol. 57, no. 11, pp. 2106–2114, 2014
- [4] **L. Motycak, A. Skotak, and J. Obrovsky:** *Analysis of the Kaplan turbine draft tube effect*, Earth and Environmental Science, vol. 12, 2010