Modeling Flow Phenomena in Pump Station with Overflow

Milan SEDLÁŘ¹,* , Jaromír PŘÍHODA² and Michal VYROUBAL¹

¹Centre of Hydraulic Research, Kosmonautů 313, 78349 Lutín, Czech Republic
²Institute of Thermomechanics, Czech Academy of Sciences, Dolejškova1402/5, 18200 Prague, Czech Republic

*Corresponding author

Keywords: CFD, Supercritical flow, Free-water level, Pump station, Overflow, DDES.

Abstract. This paper deals with the numerical analysis of flow in a complete hydraulic system representing the pump station including the influence of the free water level. Particular attention is paid to the discharge object with overflow, which is characterized by the critical flow across the crown of the overflow wall. All basic calculations were accomplished by means of the commercial software ANSYS CFX and ANSYS Fluent with the SST turbulence model. As the flow in the discharge object represents the most interesting part of the results, additional numerical simulations of flow just inside this object were carried out with the DDES scale resolving simulations. Basic calculations inside the complete pump station passage gave surprisingly steady water level in both the intake and discharge objects. They also provided a good picture of the distribution of water velocity close to the water level and corresponding height of water level above the overflow wall crown. DDES simulations indicated a periodically formed vortex on a water level behind the overflow wall crown, with a (not very distinct) Strouhal number about 0.3.

Introduction

This paper deals with the numerical analysis of flow in a complete hydraulic system representing the pump station including the influence of the free water level. Such a system is typically represented by one or several pumps, connecting piping, and the intake and discharge objects. There are two important reasons to simulate the flow in a complete system. First, the pressure losses generated between the inflow and outflow sections should be evaluated all at once, not component by component, as the flow interactions play an important role here, especially at off-design conditions. Second, the dynamics of flow phenomena in the piping and water tanks is highly influenced by the flow in the hydraulic machine itself and vice versa.

There are many references which concern the flow with free water level in suction objects of pumping stations. They concentrate mainly on vortex structures and cavitation phenomena in the pump intake [1-4]. But really few sources can be found concerning the flow in discharge objects with syphons or overflows [5,6]. This study concentrates on the pump station with the overflow based discharge object. Overflows are commonly used construction tools enabling to control easily flow and water surface level and to guarantee that back-water could not be transmitted upstream. There are many empirical formulas for 1-dimensional overflow modeling, but these are valid only for simple reference geometries. The basic hydrodynamic phenomenon there is the critical flow across the crown of the overflow wall. While subcritical flows are widely modeled using CFD and well validated by experiments, simulations of supercritical and especially near-critical flows are more complicated and less validated. In [7-9], simulations and their validation of subcritical, supercritical as well as the near-critical flows over an inclined backward-facing step in an open channel can be found. The range of the Froude numbers (1) is between 0.42 and 2.14. These simulations are the background of the numerical techniques used in this study, though the range of Reynolds numbers (2) from 44100 to 200100 is different from those in this study. Here, the Froude and Reynolds numbers are defined as follows.
Fr = \frac{U_m}{(gh)^{1/2}}, \quad (1)

Re = \frac{U_m h}{\nu}, \quad (2)

where \( U_m \) is the mean bulk velocity, \( h \) is the height of the water level, \( \nu \) is the kinematic viscosity and \( g \) is the gravity constant. In these referenced simulations, very good results were obtained using ANSYS CFX commercial code with the SST turbulence model and especially EARSM model in the form corresponding to works of Wallin and Johansson [10].

Case Description

The calculations were made for a virtual drainage pumping station with a set of five axial-flow pumps, each of them installed in a basin between the concrete pillars. The width of the basin is 4.5 m.
(Figure 1). The low water level is 3.15 m from the basin floor. Each pump (with impeller diameter of 1 m) is equipped with a draft tube and its nominal flow rate is $Q = 6 \text{ m}^3/\text{s}$. The angle between the axis of the pump and the ground floor is 45°. Each discharge piping runs through the welded diffuser to the tank with the overflow walls. The overflow walls between the concrete pillars are V shaped to get sufficiently long wall crown. Quite complicated shape of the ground floor can be found in the discharge tank (Figure 2) aimed to optimize the flow over the walls and to minimize hydraulic losses due to large vortices. The maximum height of the overflow wall is 3.35 m. The overall length of this wall is about 18 m and the thickness 0.3 m.

As it is supposed for simplicity that all the pumps are running in parallel regime, it is possible to include only one complete pump passage into the computational domain and to use symmetry boundary conditions on the interfaces of the passages.

**Computational Setup and Methods**

All basic calculations in the complete pump station passage were accomplished by means of the commercial software ANSYS CFX and ANSYS Fluent release 18 [11], solving Reynolds-averaged Navier-Stokes equations including the gravity effect. The numerical solution of free-surface flow was carried out by means of the Volume-of-Fluid (VOF) method based on the monitoring of the volume fraction $\alpha_i$ of both fluids. Metal surfaces were treated as the smooth non-slip walls, concrete surfaces were represented as the walls with the estimated roughness. All calculations were fully unsteady with the time step 0.001666 s corresponding to the rotational speed and number of blades of the pump. For the CFX software, the high resolution scheme (which is of the second order) as proposed by Barth and Jespersen [12] was used. Time derivative was treated with the second-order backward Euler scheme. For the Fluent software, the PISO pressure-velocity coupling with the PRESTO pressure interpolating scheme were used. Time derivative was treated with the bounded second-order implicit scheme. The computational grid represented about 15 mil. nodes. All these calculations in the complete pump station passage were done with the SST turbulence model, as it is most suitable for the rotating machinery calculations.

As the flow in the discharge object represents the most interesting part of the results, additional numerical simulations of flow just inside this object (starting from the interface between piping and the welded diffusor) were carried out with the DDES (Delayed Detached Eddy Simulations) scale resolving simulation available inside the Fluent software. For these calculations, the bounded central differencing scheme was used. The time step was the same as for the basic calculations, which enabled to use previous data as the inlet boundary conditions. Related to the mean velocity and water height over the overflow wall crown, time step was about $\Delta t = 0.0125 \text{ h}/U_m$. The computational grid of this single discharge object represented about 17 mil. nodes.

**Simulation Results**

Basic calculations inside the complete pump station passage gave surprisingly steady water level in both the intake and discharge objects. Though the flow inside the pump is naturally unsteady due to the interaction of impeller blades with the stator parts, the shaft rotational frequency as well as the blade passing frequencies do not influence the free water level significantly. There is a stable air pocket on the upper wall at the inlet to the draft tube (Figure 3) which does not propagate towards the pump. In the discharge object, we can find a vortex with “white water” between the ending edges of the overflow walls. The 3D view of the water level in both the intake and discharge objects can be seen in Figure 4.
Figure 3. Water volume fraction in symmetry plane of stage passage.

Figure 4. 3D view of water level (in cyan) in intake and discharge objects.

Figure 5 shows the distribution of water velocity 0.05 m below the water level and corresponding height of water level over overflow wall crown. For better resolution the color scale in the bottom picture (water height) is reduced to only positive values. It means that the negative heights are represented with same color (dark blue) as the zero one. It can be seen that the distribution of velocity and height is not uniform along the overflow wall crown. Moreover, this figure cannot capture the existing velocity profiles in front of the overflow wall and between the crown and the water level. It means that for this complicated geometry, it is not possible to apply simple 1-dimensional models of critical flow, which are applicable for straight broad-crested overflows [6].

More information on flow phenomena in the discharge object come from the additional numerical simulations with the DDES model. Figures 6–8 show water volume fraction, normalized spanwise vorticity and normalized helicity [11] in the symmetry plane of discharge object. The dark red curve in Fig. 7 - 8 represents the approximate position of water level; it is based on the water volume fraction $\alpha_w = 0.5$. A vortex on a water level behind the overflow wall crown is formed periodically, with a (not very distinct) Strouhal number (non-dimensional frequency) about 0.3. Here the Strouhal number is defined as follows

\[
St = \frac{f L}{u_c},
\]

where $f$ is the frequency of the vortex shedding, $L$ is the characteristic length (typically the width of the discharge object), and $u_c$ is the centerline velocity.
St = fh/Uₘ, \quad (3)

where \( f \) is the phenomenon frequency. Also, between the ending edges of the overflow walls we can find a large vertical structure, forming the “white water” area which changes its size and shape. The dominant frequency corresponding to this structure unfortunately was not able to detect. It is a question if this missing dominant frequency is due to insufficient amount of data or just due to the random character of this phenomenon.

Figure 5. Distribution of water velocity just 0.05 m below water level (top) and height of water level over overflow wall crown (bottom). Zero and negative heights are of the same (dark blue) color.

It is very difficult to find references which could be used to verify non-dimensional frequency obtained with DDES modeling in this study. Some comparison could be done with frequencies related to the step-mode and shedding-mode instabilities behind the backward-facing step [13]. But it must be considered, that the physical mechanisms are not the same, because in our case the dominant vortices are formed in the air due to its interaction with accelerating free water level above and behind the overflow wall. Still, the calculated non-dimensional frequency about \( St = 0.3 \) is close to the step-mode instability frequency [13], which is about \( St = 0.2 \).

Summary

During several last years, CFD analysis has become a tool which can prove the functionality and design parameters of hydrodynamic machines. This capability is even more important when guaranteeing a functionality of a new pump or turbine station as any changes and reconstructions of the suction and discharge objects of station are extremely time and money consuming. Moreover, it is practically impossible to provide something like the model tests of a full station in the hydraulic laboratory. That is why an approval of the station parameters by means of CFD is more and more required. From results and validations in [7-9] as well as our recent simulations [14], it can be concluded that in many cases numerical modeling can replace physical model tests. Of course, comparisons of the numerical results with data measured in real pump stations are rare and difficult to obtain. Some measurements of the water velocity and corresponding height of water level over overflow wall crown were done in several existing stations. Nevertheless, due to the unsteady character of flow and available measurement methods the uncertainty of the results is quite high. It can be stated that the differences between calculated and measured data are fully in the scope of the
measurement uncertainty. That is why in the next future laboratory tests with scaled transparent models of suction and discharge objects are planned aimed to compare CFD simulations with LDA and PIV measurements.

Figure 6. Water volume fraction in symmetry plane of discharge object. Two different time steps of DDES simulation ($t_x$, $t_x+0.54$ s).

Figure 7. Normalized spanwise vorticity in symmetry plane of discharge object. Two different time steps of DDES simulation ($t_x$, $t_x+0.54$ s). Approximate position of water level indicated by dark red curve.

Figure 8. Normalized helicity in symmetry plane of discharge object. Two different time steps of DDES simulation ($t_x$, $t_x+0.54$ s). Approximate position of water level indicated by dark red curve.

Acknowledgement

This research was financially supported by the Ministry of Industry and Trade of the Czech Republic under the grant project FV30104 entitled “Suction and Discharge Objects of Pump and Turbine Stations.

References

[1] T. Okamura, K. Kamemoto, J. Matsui, CFD prediction and model experiment on suction vortices in pump sump, Proc. 9th Asian Int. Conf. on Fluid Machinery (2007) 1-10.
[2] S.N. Shukla, J.T. Kshirsagar, Numerical prediction of air entrainment in pump intakes, Proc. 24th Int. Pump Users Symposium (2008) 29-33.

[3] A.C. Bayeul-Lainé, G. Bois, A. Issa, Numerical simulation of flow field in water-pump sump and inlet suction pipe, Proc. 25th IAHR Symp. on Hydraulic Machinery and Systems (2010) 1-8.

[4] C.G. Kim, B.H. Kim, B.H. Bang, Y.H. Lee, Experimental and CFD analysis for prediction of vortex and swirl angle in the pump sump station model, IOP Conf. Series: Materials Science and Engineering 72 (2015) 042044, doi:10.1088/1757-899X/72/4/0420.

[5] A. Ghafoorian, H. Mousavi Jahromi, M. Shafaei Bajestan, Hydraulic of Siphon Spillway by Physical and Computational Fluid Dynamics, World Applied Sciences Journal 14, 8 (2011) 1240-1245.

[6] T. Larsen, L. Nielsen, B. Jensen, E.D. Christensen, Numerical 3-D modelling of Overflows, Proc. 11th Int. Conf. on Urban Drainage (2008) 1-7.

[7] J. Příhoda, P. Zubík, J. Šulc, M. Sedlář, Modelling of supercritical turbulent flow over an inclined backward-facing step in an open channel, Proc. 17th Int. Conf. Engineering Mechanics (2011) 495-498.

[8] J. Příhoda, P. Zubík, J. Šulc, M. Sedlář, Modelling of critical regime in free-surface flow over an inclined backward-facing step, Proc. 18th Int. Conf. Application of Experimental and Numerical Methods in Fluid Mechanics and Energy, (2012) 212-217.

[9] J. Příhoda, P. Zubík, J. Šulc, M. Sedlář, Experimental and numerical modelling of turbulent flow over an inclined backward-facing step in an open channel, Communications 4A (2012) 6-12.

[10] S. Wallin, A. Johansson, A complete explicit algebraic Reynolds stress model for incompressible and compressible flows, Jour. of Fluid Mechanics, 403 (2000) 89-132.

[11] ANSYS, Inc., ANSYS CFD Documentation, Release 18.0 (2017).

[12] T.J. Barth, D.C. Jespersen, The design and application of upwind schemes on unstructured meshes, Proc 27th Aerospace Sciences Meeting (1989), AIAA-89-0366, doi:10.2514/6.1989-366.

[13] R. Hu, L. Wang, S. Fu, Improved delayed detached eddy simulation of flow structures behind a backward-facing step, Proc 54th AIAA Aerospace Sciences Meeting (2016), AIAA 2016-1104, doi:10.2514/6.2016-1104.

[14] M. Sedlář, J. Šoukal, T. Krátký, Numerical analysis of hydraulic systems with free water level, Proc 7th Int. Symp. on Fluid Machinery and Fluids Engineering (2016).