Modeling and Optimization of Multiphase Flow in Pump Station

M Sedlář1,*, J Machalka2 and M Komárek3

1 Centre of Hydraulic Research, Lutín, Czech Republic
2 Sigmainvest, Lutín, Czech Republic
3 SIGMA Research and Development Institute, Lutín, Czech Republic

*E-mail: m.sedlar@sigma.cz

Abstract. This paper presents the numerical analysis of flow in the pumping station including the influence of free water level in the suction and discharge objects. The suction object is based on a draft tube. Two solutions of the discharge objects (with the overflow walls and with the welded siphon) are compared so as to optimize the efficiency and start-up of the station. All calculations are based on the ANSYS CFX software with the SAS-SST and the DES scale resolving simulations. The numerical solution of the free-surface flow is carried out by means of the Volume-of-Fluid method. Calculations of flow inside one complete pump station passage provide a good picture of the distribution of water velocities and the corresponding heights of water level in front of the suction draft tube, above the overflow wall crown as well as inside and behind the siphon during the start-up. The calculations are verified with some experiments carried out in the hydraulic laboratory of the Centre of Hydraulic Research. Conclusions show the higher overall efficiency of the station with the siphon-based discharge objects but also indicate some special requirements for the start-up of this station.

1. Introduction

This paper deals with the optimal design of a pumping station, based on the numerical analysis of multiphase flow with the free water level. Such a complete hydraulic system is typically represented by one or several pumps, connecting piping, and the intake and discharge objects. Two hydraulic designs of the discharge object are considered. The first one is based on the overflow wall, the second one is based on the siphon. These two hydraulic solutions are very frequently required for safety reasons as they are easy-to-install and easy-to-operate, with minimum mechanical and electronic equipment.

Overflows represent widely used construction tools which without any mechanical components enable to control flow and water level and to prevent long-term backflow in the hydraulic system. Their design is usually based on empirical formulas for 1-dimensional overflow modeling, but these are valid only for simple reference geometries. The most important hydrodynamic phenomenon there is the critical flow across the crown of the overflow wall. Comparing to subcritical flows which are widely modeled using CFD, there is a lack of simulations of supercritical and especially near-critical flows. Some simulations of subcritical, supercritical as well as the near-critical flows including their validation can be found in [1-3], presenting the flow over an inclined backward-facing step in an open channel for the range of the Froude numbers from 0.42 to 2.14, and the range of the Reynolds numbers from 44100 to 200100. Here, the Froude and Reynolds numbers are defined as follows
\[ Fr = \frac{U_m}{(gh)^{1/2}}, \]  
\[ Re = \frac{U_m h}{v}, \]

where \( U_m \) is the mean bulk velocity, \( h \) is the height of the water level, \( g \) is the gravity constant and \( v \) is the kinematic viscosity. The numerical simulation of flow over the overflow walls can be found in [4] and an extensive numerical simulation of a pumping station with an overflow-based discharge object is described in [5].

Siphons represent also widely used discharge installation type, with possible higher hydraulic efficiency compared to overflows. But in this case, a siphon-breaking valve is required to prevent back-flow and allow venting at start-up. Such a valve can be very simple, but also quite sophisticated construction can be used to prevent dangerous effects during the reverse-flow regime [6]. There are some numerical simulations of the siphon breaker operation [7-8], but they are related mostly to the applications in research reactors. CFD simulations of a complete pumping system with the siphon outlet can be found in [9], treating the flow inside the system as the single-phase one, or in [10], with the volume of fluid (VOF) multiphase model considering also the influence of the free water level. Very complex and advanced study can be found in [11], which uses VOF model to predict transient phenomena during the stopping process in the axial-flow pump system with the siphon outlet.

Concerning the intake objects in this study, they are based on the suction elbows, which provide more uniform flow in front of the pump compared to the installations with the suction bells [12-15].

2. Test case

The aim of this paper is to find an optimal design of the pumping station using the CFD analysis of multiphase flow with the free water level. Numerical modeling has become recently a tool which can prove the functionality and design parameters of hydrodynamic machines as well as hydraulic systems. Using CFD is even more important when guaranteeing a functionality of a new pump or turbine station because any changes and reconstructions of the suction and discharge objects of the station are extremely time and money consuming and it is practically impossible to provide model tests of a full station in the hydraulic laboratory, including all interactions of hydrodynamic machines with other hydraulic components. Unfortunately, it is very difficult to compare the numerical results with data measured in real pump stations as such data are rare and hard to obtain. That is why the laboratory tests with the scaled transparent models of discharge objects have been done, aimed to compare CFD simulations with flow visualizations and PIV measurements.

In this study we tried to compare numerical simulations and measurements in discharge objects with the arrow-shaped overflow wall and with the welded siphon (figures 1-2) from the point of their critical regimes. The range of Reynolds numbers was approximately between 35000 and 50000. Experiments have been done in the hydraulic laboratory of the Centre of Hydraulic Research.

2.1. Test case setup

All calculations in the discharge objects were carried out by means of the CFD software ANSYS CFX release 18 [16], solving the Reynolds-averaged Navier-Stokes equations. The numerical solution of free-surface flow including the gravity effects was accomplished by means of the VOF method based on the evaluation of the volume fraction of each fluid. A non-homogenous model of multiphase flow has been applied with different velocities for the water and air fractions. All calculations were fully unsteady with the time step 0.01 s. The high-resolution scheme (which is of the second order) was used. The time derivative was treated with the second-order backward Euler scheme. The computational grids represented about 10 mil. nodes. All these calculations were done with the Scale-Adaptive Simulations (SAS), but some additional numerical simulations of flow were carried out also with the DES (Detached Eddy Simulations) scale resolving simulations. There, the time step decreased to \( \Delta t = 0.002 \) s and the computational grids of these simulations increased to about 17 mil. nodes. For the DES model, the high-resolution scheme was blended with the central differencing scheme according to the evaluated DES limiter.
Figure 1. Test discharge objects with arrow-shaped overflow (left) and welded siphon (right).

Figure 2. Test circuit with the siphon-based discharge object.

2.2. Test case results

The description of all experimental results and all their comparisons is out of the scope of this paper. So, for the numerical simulations with the overflow, just comparisons of measured and calculated height and shape of the free water level close to the overflow crown are shown, as the maximum height of the water level above the overflow crown is crucial for the pump performance optimization.

All the presented CFD results were obtained with the SAS scale resolving simulations. The overflow wall height is 0.244 m, the width and length of the transparent test section are 0.264 m × 4.25 m.

Two water levels at the test section outlet are considered. The first one labelled as LWL (Low Water Level) is 0.095 m above the floor at the flow rate 11.4 l/s, the second one (High Water Level) is 0.213 m above the floor at the flow rate 10.5 l/s. Figure 3 shows the calculated water level height, measured from the floor, for both numerical simulations. A detailed comparison with experiments can be found in figure 4, showing the water height in the symmetry plane (Section 1) and in the sections which are located 0.05 m, 0.1 m and 0.132 m from the symmetry plane (Sections 2-4). The agreement is quite good, though the numerical simulations slightly over-predicted the water level height just above the overflow crown.

Figure 3. Water level height measured from the floor. LWL (left) and HWL (right). CFD simulation.
Figure 4. Comparison of calculated and measured water level height in four longitudinal sections. LWL (top) and HWL (bottom).

Considering the CFD simulations inside the object with siphon, the most crucial is the start-up phase, when the siphon is filled with water before the siphon effect is reached. This phase is highly important for the pump performance optimization. The siphon outlet cross-section is 0.16 m × 0.16 m, the width and length of the transparent test section are 0.304 m × 4.25 m and the flow rate is 13.8 l/s.

Figure 5 shows four phases during filling the siphon with water (water is in blue color). The first one (Phase a) represents the instant when water starts to fill the upper part of the siphon but still has not reached the siphon outlet. In the second phase water jet enters the free water level in the discharge tank. In the third phase water fills the whole cross-section of the siphon bend and enters the valve. At this instant the valve should be closed to prevent air sucking. Then, the remaining air is quickly released and the siphon effect starts (Phase d). Approximately the same phases are visualized during the experiment in figure 6.

3. Pumping station
The real simulation of the whole pumping system represents a virtual project of the pumping station with a set of six axial-flow pumps, each of them installed in a basin between concrete walls. The width of the basin is 4.2 m (figure 7). The water level can change from 4.25 m to 4.62 m above the intake basin floor at its lowest altitude. Each pump is equipped with a suction elbow and its flow rate is
Q = 8.5 m³/s for the low water level at the suction and high-water level at the discharge object. There the water level can change from 3 m to 3.2 m above the discharge basin floor at its lowest altitude. The angle between the axis of the pump and the ground floor is 45°. Each discharge piping runs to the tank with the welded siphon or alternatively, with the welded diffuser and a V-shaped overflow walls (figures 8-9). Quite complicated shape of the ground floor can be found in the discharge tank with overflow so as to suppress large vertical structures. 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.

Figure 5. Filling siphon with water (in blue): a) exhausting air, b) first interaction of water jet with water level, c) phase with water entering siphon-breaking valve, d) phase after closing valve. CFD.

Figure 6. Visualization of siphon start-up. The same phases as in figure 5.

Figure 7. Arrangement of pumping station with 6 inclined axial-flow pumps and siphon outlets.
3.1. Case setup
The CFD simulations follow the setup from chapter 2.1. Nevertheless, while all the metal surfaces were treated as the smooth non-slip walls, the 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 (199 rpm) and number of blades of the pump (4 in the impeller, 7 in the stator). The impeller diameter is 1.8 m. Computational grids represent about 16 mil. nodes. All these calculations in the complete pump station passage were done with the SAS turbulence model.

3.2. Case results
For both types of installation, a transient analysis has been done for the full range of parameters, which enabled to compare in detail overall efficiency as well as technical demands required for the pumps. Additional transient analysis of the station start-up phase has been done when using the siphon-based outlet.

Figure 10 shows the water volume fraction in the symmetry plane of the pump station for both installation types, for low water level at the inlet and high-water level at the pump station outlet. The corresponding free water level shape can be seen in figure 11. It has been found, that the complicated shape of the floor in the discharge object with overflow wall significantly improved the flow structures in front of and behind the wall, though there is still a strong vortex in the middle of the rear part of the V-shaped overflow wall. This phenomenon is very similar to the vortical structure described in [5].

Figure 8. Installation with welded siphon in the discharge object.

Figure 9. Installation with V-shaped overflow wall in the discharge object.

Some stages of the start-up regime for the station with siphon can be seen in figure 12. During this procedure (especially between Stages d and e), the pump delivery head highly exceeds the delivery head necessary for the regime with the presence of the siphon effect. The calculated characteristic curves of the pump corresponding to the full range of operation regimes (resulting from the expected variation of water levels) are shown on the right-hand side of each graph in figure 13. On the left-hand side of the graphs there are two values corresponding to the siphon start-up regime. The theoretical
values of the delivery head and power input correspond just to the highest altitude of water when the siphon is filled. In reality, the necessary delivery head taken from the CFD analysis is even higher, as it also includes increased hydraulic losses caused by a very complex flow pattern of water and air mixture in the siphon during start-up.

![Figure 10](image1.png)

**Figure 10.** Water volume fraction in the symmetry plane of pumping station. Installation with siphon (top) and overflow (bottom).

![Figure 11](image2.png)

**Figure 11.** Free water level shape for LWL at the station inlet and HWL at the outlet. Installation with siphon (left) and detail of water level in the discharge object with overflow (right).

4. Conclusions
The two presented installation types of the discharge object have both their advantages and disadvantages. The overflow wall is the simplest construction tool enabling to control easily the flow and water level and to guarantee that back-water could not be transmitted upstream. But the overflow edge must be long enough to guarantee reasonable flow conditions, which makes the construction works more expensive. Also complicated shape of the basin floor can increase expenditures significantly. The operating range of the pump does not depend on the water level in the discharge object, so it is possible to set the pump operation very close to its best efficiency point (BEP). On the other hand, it decreases the overall efficiency of the station, especially at lower water levels in the discharge object.

Siphons are in principle more efficient from the hydraulic point of view and their discharge objects can be usually shorter compared to the discharge objects with overflow wall. But during the station start-up regime, pumps must work at quite low flow rates, far from their BEP. The start-up regime also requires a large reserve in the power output of the engines, which can (together with a siphon-
breaking/venting valve) increase the installation price. It is also more difficult to reach BEP during the station operation.

Figure 12. Start-up of siphon-based pumping station.

Figure 13. Characteristic curves for power input $P$, delivery head $H$ and pump overall efficiency $\eta_{pa}$ corresponding to the full range of operating regimes including the siphon start-up.
From the point of view of the long-term operation and the eco-design requirements, at the presented conditions the station with the siphon outlet is more efficient, though it requires at least simple automatic valves.

But generally, in any case the correct CFD analysis can significantly increase the efficiency and reliability of a new design of the pumping station and to optimize both the construction works and the correct pump and motor selection.

Acknowledgments
This research was financially supported by the Ministry of Industry and Trade of the Czech Republic under the grant project FV30104 “Suction and discharge objects of pump and turbine stations.

References
[1] Příhoda J, Zubík P, Šulc J and Sedlář M 2011 Modelling of supercritical turbulent flow over an inclined backward-facing step in an open channel, Proc. 17th Int. Conf. Engineering Mechanics, pp 495-8
[2] Příhoda J, Zubík P, Šulc J and Sedlář M 2012 Modelling of critical regime in free-surface flow over an inclined backward-facing step, Proc. AEnMFiMaeE, pp 212-217
[3] Příhoda J, Zubík P, Šulc J and Sedlář M 2012 Experimental and numerical modelling of turbulent flow over an inclined backward-facing step in an open channel, Communications 4A, pp 6-12
[4] Larsen T, Nielsen L, Jensen B and Christensen E D 2008 Numerical 3-D modelling of Overflows, Proc. 11th Int. Conf. on Urban Drainage, pp 1-7
[5] Sedlář M, Příhoda J and Vyroubal M 2019 Modeling flow phenomena in pump station with overflow, Proc Int. Conf. AMMSo 2019, pp 204-10, doi:10.12783/dccse/amms02019/30127
[6] Shomayramov M, Norkulov B, Rakhmanov J, Tadjtyeava D and Suyunov J 2019 Experimental researches of vacuum breakdown devices of siphon outlets of pumping stations, E3S Web of Conf. 97, 05009, doi.org/10.1051/e3sconf/21199705009
[7] Lee K Y, Yoon H G and Park D K 2017 CFD analysis of a decay tank and a siphon breaker for an innovative integrated passive safety system for research reactor, Sci. and Tech. of Nuclear Inst. 3106278, pp 1-9, doi.org/10.1155/2017/3106278
[8] Ramajo D E, Corzo S E, Alberto P, Moscicaro C, Saez M and Nigro N M 2017 Numerical simulation of siphon breaker of an open-pool type research reactor, Mecanica Comp. XXXV, pp 1809-25, doi:10.1177/1687814017723280
[9] Zhu H, Zhu G, Lu W and Zhang Y 2012 Optimal hydraulic design and numerical simulation of pumping systems, Proceneda Engng., 28, pp 75-80, doi:10.1016/j.proeng.2012.01.685
[10] Sedlář M, Šoukal J and Krátký T 2016 Numerical analysis of hydraulic systems with free water level, Proc 7th Int. Symp. on Fluid Machinery and Fluids Engineering
[11] Liu Y, Zhou J and Zhou D 2017 Transient flow analysis in axial-flow pump system during stoppage, Advances in Mech. Engng., 9 (9), pp 1-8, doi:10.1177/1687814017723280
[12] Okamura T, Kamemoto K and Matsui J 2007 CFD prediction and model experiment on suction vortices in pump sump, Proc. 9th Asian Int. Conf. on Fluid Machinery, pp 1-10
[13] Shukla S N, Kshirsagar J T 2008 Numerical prediction of air entrainment in pump intakes, Proc. 24th Int. Pump Users Symposium, pp 29-33
[14] Bayeul-Lainé A C, Bois G and Issa A 2010 Numerical simulation of flow field in water-pump sump and inlet suction pipe, Proc. 25th IAHR Symp. on Hydr. Machinery and Systems, pp 1-8
[15] Kim C G, Kim B H, Bang B H and Lee Y H 2015 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(4) 042044, doi:10.1088/1757-899X/72/4/0420.
[16] ANSYS, Inc. 2017 ANSYS CFD Documentation, Release 18.0