Numerical and Physical Modelling of the Performance of the Pro-vortex Vanes in Shaft Spillways

Miroslav Broucek 1, Ladislav Satrapa 1, Martin Kralik 1, Jiri Soucek 1

1 CTU in Prague, Faculty of Civil Engineering, Thakurova 7, Prague 6, 16629, Czech Republic

miroslav.broucek@fsv.cvut.cz

Abstract. The paper focuses on the analysis of hydraulic conditions in the proximity of the intake part of high shaft spillways equipped with pro-vortex vanes and discusses recent enhancements in modelling of the shaft spillways and compares the acquired results of the performance of the spillway after complete removal or rehabilitation of the vanes in context of capacity and overall hydraulic conditions. Increasing requirements on safety of embankment dams during floods with respect to anticipated effect of the climate change scenarios on parameters of design floods demand further assessment of capabilities of outlet structures to meet the updated needs. Such dam safety assessments often conclude in the need of designing additional measures to improve existing structures. Despite different approach to the evaluation of the uncertainties and subsequent risk assessment the goal of improving safety of large dams remains consistent in the effort of all developed countries. Adjustments of the intake part of shaft spillways can present a valid design option for increasing capacity of the complex spillway / tunnel structure if supported by solid analysis of hydraulic conditions inside these structures. As the governing idea of the pro-vortex vanes is to ensure spiral flow inside the shaft and to minimize the pressure fluctuations the paper presents results from physical model of several designs of the pro-vortex vanes which approximated possible adjustment of tower like shaft spillway of existing large dam in Czech Republic and also results from CFD modelling illustrating the importance of combination of both modelling approaches. For the CFD part, different turbulence models are discussed.

1. Introduction
The shaft spillways often present a technically and economically viable option when considering a design of an outlet structure for large earthfill or rockfill dams. Therefore the hydraulic performance and conditions for different flow regimes near the inlet part, which influence significantly the capacity, has been studied in the past [1], [2] especially with respect to pressure fluctuations of aerated spiral flow in vertical shafts and transition elements. In contrast to the straight spillways for shaft spillways with circular (or another closed shape) the coefficient of discharge depends not only on the actual thickness of the overfalling nappe but also to a ratio of the thickness of the overfalling nappe to the radius of the circular plan [3], [4]. When the cross section of the spillway, i.e. the intake part of the shaft spillway, vary from the original shape presented in the literature the effect on discharge coefficient should be evaluated using physical or numerical models especially in case where local morphological conditions lead to shapes allowing negative pressures [5].
The transition between free flow and submerged region at the inlet part is accompanied with the creation of a vortex which can be stable or unstable based on the local conditions and influence of additional elements near the intake part such as pro-vortex vanes. The transition process is also accompanied with significant pressure fluctuations which might induce unwanted loading on the shaft body and adjacent structures. Especially for dams with clay core, the potential of increased seepage along the shaft or tunnel and subsequent internal erosion evolution is considered unacceptable. When assuming symmetrical inflow conditions, i.e. without influence of the reservoir topology or position of the intake part with respect to dam or banks, and no streamlining elements, the flow in the shaft can be considered axial in free-flow and transition regions, while unstable vortex will arise under submerged conditions. Ensuring stable spiral flow decreases the pressure fluctuations [6] at the cost of decreasing capacity which on the other hand is unacceptable and prevented by anti-vortex elements for spillways passing through solid rock.

The applicability of physical modelling using Froude’s model law and the limits was tested and successfully verified on scaled models by Sikora [7] who also described the coefficient of aeration of the pressure flow in outlet tunnels from shaft spillways.

Decreasing calculating costs, led to significant progress in numerical simulations in the capacity curve estimation [8]. The Two-phase CFD simulations using RANS method with k-ε SST turbulence model and non-homogeneous velocity field can provide sufficiently accurate results [9], also presented in this paper. The remaining issue is selection of correct boundary conditions with respect to reservoir topology, complete geometry of the spillway combined with still demanding calculation costs under unsteady state condition. Assuming steady state, the of pressures fluctuations and possible periodic loading cannot be identified nor quantified as shown in [10] but as the periodic loading with potentially dangerous frequencies can occur during certain discharge range, the inherently unsteady physical modelling cannot be omitted.

2. Physical model

A physical model of a real shaft spillway structure was constructed in the laboratory in order to study the performance of different design and setup of streamlining elements. In this paper the spillway of Jirkov dam in Czech Republic was used as prototype for laboratory model scaled 1:20. The outlet structure of this 55.6 m high rockfill dam with a clay core consists of tower like shaft spillway and two bottom outlets 800 mm in diameter each. The spillway consists of circular (9.4 m in diameter) nappe shaped overflow structure – the inlet part, with four very unusual pro-vortex vanes, followed by vertical shaft (3 m in diameter), a sharp bend, short lowered section, aerated sudden raise of the ceiling profile and an outlet tunnel with 2% bed slope ensuring free-surface supercritical flow conditions. The bottom outlets discharge behind plunger valves into the lowered section of the outlet tunnel from shaft spillway just before the sudden raise of the flow profile.

The cross section of the inlet part is non-vacuum free shaped overflow section for nappe thickness 0.5 – 1.1 m. The current pro-vortex vanes are only 0.4 high but 9.0 m long, see following figures.

In order to study the effect of the complex structure the laboratory model was constructed including the outlet tunnel and several meters of the channel downstream of the dam toe. Furthermore new potential designs of pro-vortex vanes with more common shape were tested for pressure fluctuations and hydraulic performance.
Figure 1. Shaft spillway of a Jirkov dam: a) intake part – prototype; b) laboratory model

2.1. Tested alternatives
Apart from the current setup with four specific vanes following alternatives were tested.

Table 1. Setup alternatives tested at the laboratory model

| Alternative | Description |
|-------------|-------------|
| Var 1       | current state (i.e. four atypical pro-vortex vanes and sharp bend at the bottom of the shaft) |
| Var 2       | no streamline elements present (current pro-vortex vanes removed and sharp bend kept) |
| Var 3       | no streamline elements at the intake part but additional hydraulically smooth transition added at the bottom of the shaft |
| Var 4a      | six new pro-vortex vanes (see the following figure) with two vanes parallel with the outlet tunnel axis |
| Var 4b      | six new pro-vortex vanes rotated by 90 degrees with respect to Var 4a i.e. no vane is parallel with the outlet tunnel axis |
| Var 5a      | five new pro-vortex vanes (see the following figure) with one vane parallel with the outlet tunnel axis |
| Var 5b      | five new pro-vortex vanes rotated by 36 degrees with respect to Var 5b so no vane is parallel with the outlet tunnel axis |

Alternatives Var 4a, 4b, 5a and 5b were tested also with the hydraulically smooth transition added at the bottom of the shaft and are denoted “ + smooth bend” further in the text.
2.2. Measurement

The experiments were conducted under the steady inflow into the large upper reservoir in the laboratory for 10 different discharges corresponding with floods with different reoccurrence intervals from $Q_1$ to $Q_{10000}$ or full capacity, i.e. submerged regime at the intake part.

The model was equipped with 16 pressure probes with measuring frequency 1 Hz to 1 kHz. Four probes were positioned along the bottom part of the vertical shaft and 6 probes on the bottom just downstream of the bend.

The discharge was measured by induction flowmeter at the inlet pipe to the upper reservoir. Also the water level was measured in the reservoir and along the outlet tunnel and in the channel downstream.

The pressure fluctuations were assessed from two points of view. Firstly, the fluctuations were assessed with respect to the probability of exceedance for particular discharge. It is important to stress out, that the results vary significantly along the shaft and the width of the outlet tunnel bottom. Therefore, the comparison must take into account position of the probe, overall magnitude of the pressure fluctuation and the probability of occurrence of the particular discharge conditions. Secondly, the possible oscillations caused by fluctuations with dominant frequencies were analyzed as such loading can potentially damage the structure and increase the risk of internal erosion.
3. Numerical modelling

The aim of the numerical model was to analyse the discharge – water level relationship under different mesh density and turbulence model. The results were compared with the physical model and results commented for intake part without streamlining elements.

ANSYS CDF with RANS method and k-ε and Shear Stress Transport (SST) turbulence models were used for numerical modelling. The non-structured mesh generated by the Delaunay algorithm with significant refinement around the intake part, boundary of the domain and water surface was applied for discretization of the model. Axisymmetric geometry of the spillway was taken into account and the 3D model was created by rotating cross section, see figures 4 and 5.

![Figure 4. a) solved domain and b) detailed view on the intake part of the shaft](image)

![Figure 5. Cross section of the mesh](image)

3.1. Boundary conditions (BC) and turbulence models parameters

To represent the inflow into the solved domain the Inlet-Total Pressure BC with defined hydrostatic (depth governed) distribution was used. The free water level surface was defined via Opening pressure BC with average relative pressure 0 kPa, which allows for water level variations.
The only discharge option was through the shaft itself and was defined as Averaged static pressure BC with average relative pressure 0 kPa. Finally the Rough Wall BC was used to approximate the influence of the concrete structure. All the boundary conditions are clear from following figure.

![Figure 6. Boundary conditions: a) inflow, b) discharge, c) free water surface and d) rough walls](image)

Following table summarizes the constants used in both turbulence models. The SST model uses k-ε model in close proximity to the obstacles (walls) and k-ω model in other parts of the domain.

|                     | k-ε model | k-ω model |
|---------------------|-----------|-----------|
| C_{\epsilon}        | 1.44      | 0.09      |
| C_{\omega}          | 1.92      | 0.555     |
| C_{\mu}             | 0.09      | 0.075     |
| \sigma_{\epsilon}  | 1.0       | 2         |
| \sigma_{\omega}    | 1.3       | 2         |

4. Results and discussions
Following figures present selected results from both numerical and physical model. At first the problem of flow regimes and their boundaries was analysed from the physical model. It is clear from the following rating curve of the outlet structure, that while the relatively higher hydraulic smoothness of the improved bend can be seen to have rather positive impact in terms of capacity and submergence
occurrence, the streamlining elements reduce the capacity in the free-flow region by approximately 4 - 5%. Also the current pro-vortex vanes cause earlier transition to the submerged regime. The newly designed pro-vortex vanes in Var 5b setup, however, allow for higher discharges before transition into submerged regime occurs.

**Figure 7.** Rating curve of the spillway structure for Var1, Var 2 and Var 5b setups

**Figure 8.** Probability of exceedance of pressure fluctuation at probe no. 3 for 100 year flood discharge
The figure 8 presented above clearly demonstrates the effect of streamline elements, even the atypical current ones, on the pressure fluctuation. Also the effect of the hydraulically smooth transition element at the bottom of the shaft is well demonstrated. The removal of the current pro-vortex vanes would lead to increased capacity, see figure 7, but without implementing hydraulically smooth bend, the pressure fluctuations will rise. Implementation of more typical design of pro-vortex vanes will ensure spiral flow mode along the shaft and significantly reduce the pressure fluctuations even further.

Figure 9. Frequency analysis of measured pressure fluctuation at probe no. 13 for 10,000 year flood discharge

There is, however, a risk of oscillations. As presented in figure 9, some of the probes at certain discharges shown dominant frequencies and therefor a significant dynamic loading on the structure. It should be stress out that the FFT analysis only discovered dominant frequencies at certain probes and only for very high discharges which are rather unlikely to occur during the remaining service life of the dam. The benefit of the lowered fluctuations is evident event for discharges with higher probabilities of occurrence, e.g. $Q_{20}$.

The results from the numerical models as presented in figure 10 demonstrate good agreement between both physical and numerical model. The geometry scaling effect was insignificant and the difference between turbulence models is approximately 10% with physical model results directly between different turbulence models. The numerical calculations were still rather demanding and it is quite clear that gaining similar pressure fluctuation results for all tested alternative setups and discharges would present an enormous task if compared with relatively less time consuming experiments in the laboratory due to inherently unsteady flow conditions.
5. Conclusions
Specific local conditions and design decisions lead to construction of complex outlet structures for which the general recommended values of discharge coefficients are not applicable. The hydraulic interaction between the inlet part, shaft and outlet tunnels also presents a challenge when assessing the capacity curves without help of modelling and can end up with significant under or overestimation of the spillway capacity thus adding undesirable uncertainties to the safety assessment.

Physical modelling of the shaft spillways remains a valid option for description of hydraulic conditions in the entire outlet structure system from both time and economy point of views despite recent computing capabilities. The physical model allows analysis of the results of any topological adjustments in a real time, which is especially useful for optimization of shape of designed elements, while every change in topology of the numerical model requires new calculation process. The limits of the physical models are well defined and the numerical models can provide answers to those questions beyond such limits.

Acknowledgment(s)
The financial support of the project No. VI20192022121 provided by the Ministry of the Interior of the Czech Republic is gratefully acknowledged.

References
[1] W E. Wagner “Morning-Glory Shaft-Spillways. Determination of Pressure – Controlled Profiles” Proceedings ASCE, vol. 80 (432), 1954.
[2] A. Gardel, “Les évacuateurs de crues en déversoirs circulaires” Bulletin Technique de la Suisse Romande, (27), 1949.
[3] K. Haindl and J. Cabelka “Turbulent pressure fluctuation in a complex hydraulic structure” Proceedings of the 9th IAHR General Meeting, pp. 56–65, 1961.
[4] M. Broucek, L. Satrampa, M. Zukal and M. Kralik “Existing Shaft Spillway Enhancement Based on Physical Modelling” Proceedings of the International Symposium on “Appropriate...
technology to ensure proper Development, Operation and Maintenance of Dams in Developing Countries”, pp. 311-320, 2016.

[5] L. Tanchev “Dams and Appurtenant Hydraulic Structures 2nd Ed.” CRC Press, Leiden, NL, 2014.

[6] P. Novak and J. Cabelka “Models in hydraulic engineering physical principles and design applications” Pitman Publication, London, UK, 1981.

[7] A. Sikora “Zavzdušnenie šachtových priepadov” Práce a štúdie VÚVH Bratislava, SR (in Slovak language), 1965.

[8] J. Ackers, F. Bennett and J. Khan “Hydraulic design issues in raising the morning glory spillway of Chaffey Dam” IPENZ Proceedings of Technical Groups 39. Multiple Use of Dams and Reservoirs: Needs, Benefits and Risks, 2013.

[9] S. E. Rafiee and M. M. Sadeghiayad “3D cfd energy analysis of the performance of a counter flow vortex tube” Int. J. Heat and Technology, vol 32 (1-2), pp. 71–77, 2014.

[10] E. Skarupova and P. Nowak “Physical and mathematical modelling for aided overspill redesign” Proceedings of the Hydro 2016 - Achievements, opportunities and challenges, 2016.