3D NUMERICAL MODELLING OF THE CAPACITY FOR A PARTIALLY PRESSURIZED SPILLWAY

Samuel Vingerhagen[1] & Nils Reidar B. Olsen[1]

[1]Department of Hydraulic and Environmental Engineering, The Norwegian Univeristy of Science and Technology,
S. P. Andersens vei 5, 7491 Trondheim, Norway
E-mail: Samuel.Vingerhagen@ntnu.no

Abstract

The commercial CFD program STAR-CCM+ was used to compute the stage-discharge curve for the spillway of the Innerdalen Hydropower Dam in Norway. The spillway consisted of first a free overflow section and then a system of tunnels where the flow could be pressurized. Air entrainment and negative pressures caused the flow physics to be fairly complex.

The computer program solved the Navier-Stokes equations with the k-epsilon and k-omega turbulence model. A volume of fluid method was used to determine the water surface location. Several types of non-orthogonal unstructured grids were tested. The results were compared with experimental values from a physical model study. Parameter tests were done on roughness, turbulence model, time steps and grids. The results showed that the numerical model predicted the free flow section very well, with an accuracy less than 5 %. For higher discharges where the tunnel was limiting the discharge, the accuracy was under 15 %.

Introduction

Computation of the stage-discharge curve for spillways is a common task for hydraulic engineers. Norway has a large number of hydropower reservoirs that regularly are assessed for functionality during floods. Many of the spillways have complex geometries, including tunnels that can be pressurized. The rating curve can then have an unusual shape with high slopes when the tunnels are filled. This flow situation often occurs during the largest discharge the spillway should be designed for.

Determining the capacity of the spillway system has often been done by physical model studies. An alternative approach is to use a three-dimensional numerical model. The current study discusses both methods applied to the spillway of the hydropower dam at Innerdalen in Norway.

Three-dimensional numerical models have been used earlier to compute spillway capacity for a large number of cases. Also, different computer programs have been used. Li et al (2011) used the program FLUENT to compute the spillway capacity of the Canton Dam in USA. DeCesare et al (2011) used the FLOW-3D program to estimate the spillway capacity for the Koman Dam in Albania. The same computer program was used by Jacobsen and Olsen (2010) to compute the capacity of the spillway of the Sysen Hydropower dam in Norway. Feurich and Rutschmann (2005) used Flow3D to assess the performance of the spillway at the Merowe dam. Haun et al (2011) used both Flow3D and SSIIM 2 to compute the water surface profile over a broad-crested weir. All these studies obtained fairly good agreement between the numerical model results and physical model studies. The accuracy of the computed water levels and coefficients of discharge were under 5 %.

The current study focuses on a complex spillway system where the flow is partially pressurized for the largest discharges. Similar to the Sysen dam case (Jacobsen and Olsen, 2010), the Innerdalen dam spillway conveys water from the spillway through rock-blasted tunnels. The tunnels are not completely filled at low discharges, and free surface flow occurs in the whole system. The conveyance of the tunnels is the limiting capacity factor for high discharges, when the tunnels become pressurized.

The complex flow physics leads to a non-smooth stage-discharge curve with is difficult to estimate. Physical model studies have therefore been used for the Innerdalen dam. Although such studies should in principle be fairly straightforward, there are some sources of errors which can affect the result. One problem is air entrainment. Two-phase flow is difficult to scale in a physical model study. The current Innerdalen Dam physical model study used PVC pipes to model the tunnel, which were not transparent. It was therefore difficult to observe the existence and extent of the air entrainment in the system.

The current study uses the computer program STAR-CCM+ to replicate the flow conditions at the Innerdalen Dam, and compare the results with the physical model.

The Dam Innerdalen Spillway

The hydropower reservoir Innerdalsvatnet is located 100 km south of Trondheim in Norway. The reservoir is made with a 57 m tall dam, requiring a considerable spillway capacity to cope with large floods. The spillway is shown in Figure 1. The water will flow over a concrete wall, down into a bay and then into a 45 degree rock-blasted shaft. At the bottom of the shaft, a 200 m rock-blasted tunnel leads the water to a river.

Figure 1. Photograph of the spillway structure at the Innerdalen Dam (Vingerhagen, 2011)

Figure 2 shows a sketch of the spillway with the tunnel systems as built in the physical model. Non-transparent PVC pipes were used to model both the shaft and the tunnel. The outlet of the tunnel was not built similar in the prototype and the lab. To prevent the water from spilling on the floor of the lab, a vertical pipe was placed at the end of the tunnel, leading the water down to the reservoir below the laboratory floor. A photograph of this pipe is shown in Figure 3. The effect of this pipe will be discussed in more detail later.

The flood discharge for the spillway was 147 m^3/s for a 1000 year flood and 217 m^3/s for the probable maximum flood.

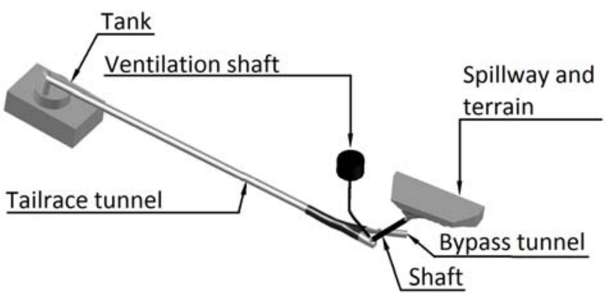

Figure 2. 3D sketch of the spillway system as built in the laboratory.

Figure 3. PVC pipe at the end of the tunnel in the laboratory model (Røneid & Sæter, 2007)

The physical model study was carried out with both smooth and rough pipes. The PVC pipes were originally smooth, but after the first test phase sand particles were glued on the inside of the pipes. The prototype tunnel was rock-blasted with fairly rough walls.

Looking at Fig. 2, an air shaft was included in the physical model. The purpose of this ventilation was to avoid negative pressure in the 45 degree inclined shaft. Negative pressures were observed in the physical model study Stage-discharge curves were made with the air shaft both closed and open. An open air shaft is in the following figures denoted “With air”, and “Without air” means the air shaft is closed. The curves are shown in Fig. 4. It is clear that the capacity of the system at high discharges very much depend on wall roughness and air entrainment.

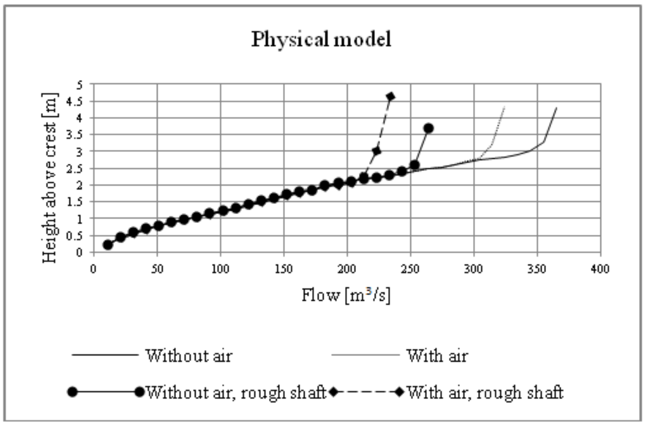

Figure 4: Stage-discharge curves from the physical model test

Results from the numerical model study

The current study used the STAR-CCM+ software, which is a general purpose package that solves the Navier-Stokes equations with a range of turbulence models and discretization schemes. The program uses the volume of fluid method to compute the location of the free surface in a fixed grid. A number of different unstructured grids are available. The current study used mainly a trimmed grid, while a polyhedral grid was used some times in the tailrace tunnel. The trimmed grid is based on an orthogonal hexahedral grid, which is refined in some areas. At locations with complex geometry some non-orthogonal cells are included. Our experience is that the dominantly orthogonal trimmed grid usually gives the best accuracy and stability.

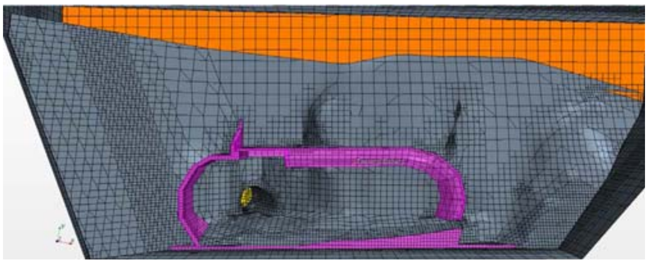

Figure 5. Trimmed grid of the reservoir and spillway seen from above. The concrete construction is shown with pink color. The yellow are is the 45 degree shaft. The orange area is the inlet of the model.

The volume of fluid method gave fluid velocities in all cells, also the air above the water surface. Figure 6 shows velocity vectors for the flow field over the concrete structure of the spillway. Higher velocities are shown at the top and downstream of the spillway, which is in accordance with laboratory studies. The profile of the water level also corresponds well to observations.

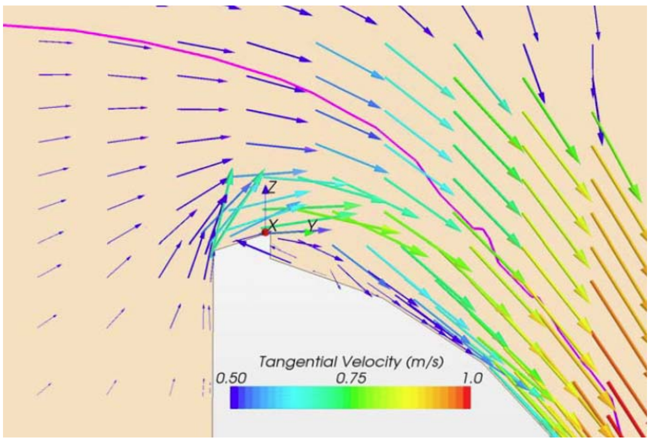

Figure 6. Velocity vectors over the spillway. The pink line shows the water level.

The rating curve was obtained from STAR-CCM+ by introducing a macro, recording the water level at a point in the reservoir upstream of the concrete construction in Fig. 1. The corresponding discharges were also written to a file, enabling the construction of spillway rating curves. These are given in Figs. 7a, b and c. Each of the curves includes the results of a parameter sensitivity test. The parameters that were tested were the time step, turbulence model and the grid size. The figures show considerable effect of the parameters for high discharges. The results are independent of the parameters for lower discharges where the water level is below 2 meters above the spillway crest. The reason is that the flow over the concrete structure is limiting the discharge for low water levels. The water is flowing with free surface through the tunnels. For water levels higher than 2 meters above the spillway crest, the tunnels are filled with water and the water level downstream of the concrete structure is fairly high. The limiting factor for the spillway system is then the capacity of the tunnels. And this capacity is not so straightforward to determine.

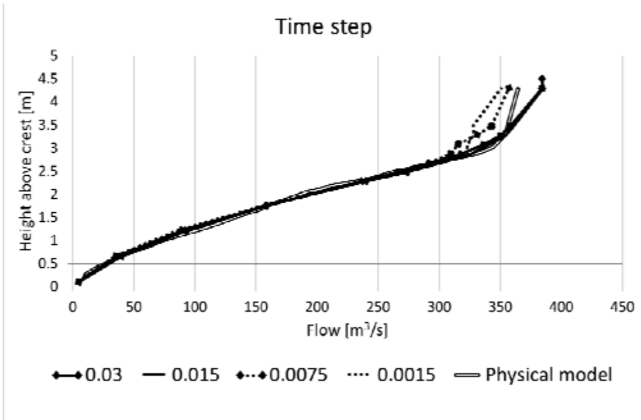

Figure 7a. Rating curves from parameter sensitivity tests: time step. Situation without air and smooth walls.

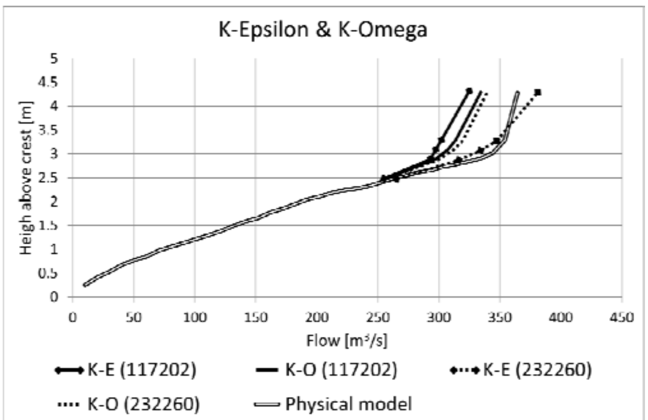

Figure 7b. Rating curves from parameter sensitivity tests: turbulence model. Situation without air and smooth walls.

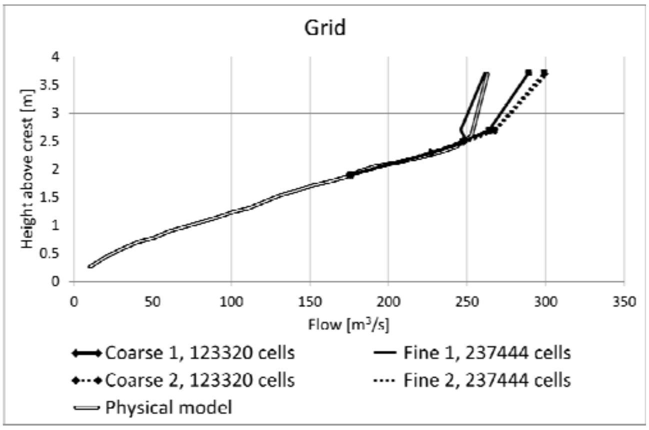

Figure 7c. Rating curves from parameter sensitivity tests: grid size. Situation without air and rough walls.

The main question looking at the results in Fig. 7 is the reason for the discrepancies between the different parameter variations. The most likely explanation can be found when looking at the computed air concentrations from the CFD model. Fig. 8 a, b and c shows the air concentrations in two different locations of the model. Fig. 8a and b shows the concentrations at the outlet of the vertical pipe at the downstream end of the model. The negative pressure inside the pipe sucks air into the outlet of the pipe. This reduces the cross-sectional area, causing added energy loss in the pipe. The size of the contractions depend on how much air that enters the system. From the figures, this seems to be an unstable situation where only small changes in the flow field will make considerable enlargements of the air pockets.

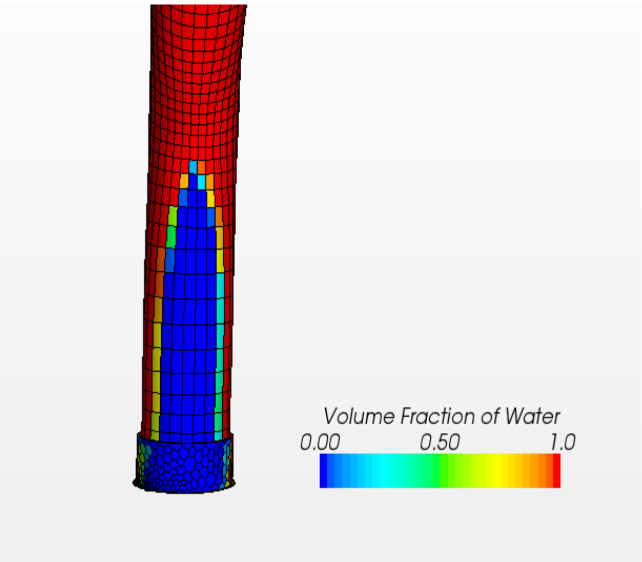

Figure 8 a. Air concentrations at the vertical pipe outlet for situation without air and smooth walls.

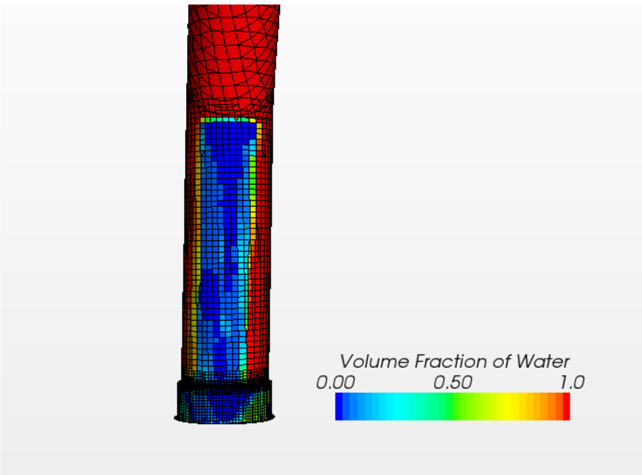

Figure 8 b. Air concentrations at the vertical pipe outlet for situation without air and smooth walls.

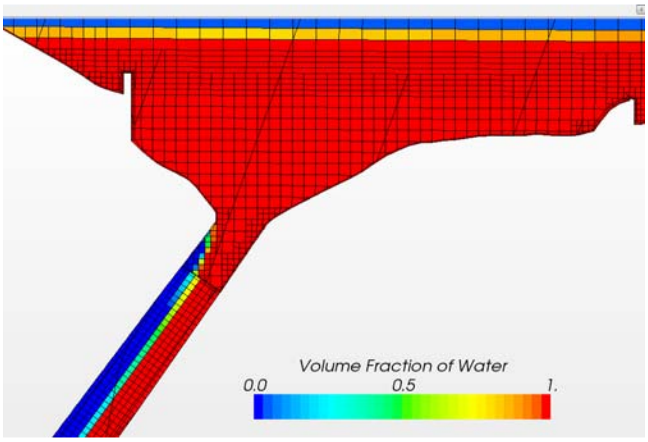

Figure 8 c. Air concentration at the top of the 45 degree shaft for situation with air and smooth walls.

Conclusions

The numerical model is able to compute the stage-discharge curve of the free flow spillway with a high degree of accuracy. The stage-discharge curve was then independent of the tested time steps, turbulence models and grid sizes. However, the accuracy of the numerical model is reduced when spillway system is partially pressurized. The energy loss in the tunnel is then limiting the capacity. The problem with modeling this situation is most likely instabilities due to air entrained into in the tunnel. The different amount of air gives varying sizes of air pockets in the tunnel, leading to different singular losses in the tunnel. Still, the accuracy for this situation was under 15 % (Fig. 7).

Another conclusion for the current study is that it was easier to understand the flow physics in the CFD model than in the physical model. This was because it was not possible to observe the flow field and the air bubbles in the non-transparent plastic pipes of the physical model. The 3D CFD color graphics gave very nice information about the air concentrations, pressures and velocity vectors.

Acknowledgements
We want to thank Trønderenergi AS for allowing us to use the results from the physical model study. We also want to thank Mr. Morten Skoglund and Mr. Å. Sæter for providing information about the physical models study.

References

De Cesare, G., Daneshvari, M ., Federspiel, M., Malquarti, M., Chauvin, GE. and Schleiss, A (2011) *Physical and numerical modeling of the spillways and plunge pools of Koman Dam in Albania*. La Houille Blanche, No. 3, pp. 48-55, June.

Feurich, R. and Rutschmann, P. (2005) *Efficient design optimization and investigation by combining numerical and physical models*, 32nd Congress of the International Association of Hydraulic Engineering and Research, Seoul, Korea.

Haun, S., Olsen, N. R. B. and Feurich, R. (2011), *Numerical modelling of flow over a trapezoidal broad-crested weir*, Engineering Applications of Computational Fluid Mechanics, Vol. 5, No. 3, pp 397-405.

Jacobsen, J. and Olsen, N. R. B. (2010) *3D numerical modelling of the capacity for a complex spillway*, Water Management, Vol. 163, Issue WM6, pp. 283-288.

Li, S. H., Cain, S. Wosnik, M., Miller, C., Kocahan. H. and Wyckoff, R. (2011) *Numerical modelling of probable maximum flood flowing through a system of spillways*, Journal of Hydraulic Engineering, Vol. 137, No. 1, pp. 66-74.

Røneid, Ø. and Sæter, Å. (2007) *Dam Innerdalen – Hydraulic analysis and physical model test of spillway*, MSc Thesis, Department of Hydraulic and Environemental Engineering, The Norwegian University of Science and Technology. (in Norwegian)

Vingerhagen, S. (2011) *Numerical modeling of the spillway at Innerdalen Dam*, MSc thesis, Department of Hydraulic and Environmental Engineering, The Norwegian University of Science and Technology. (in Norwegian).