DOI: 10.14264/uql.2020.606

Numerical modelling of flow in parallel spillways

P.A.M. Hedberg[1], J.G.I. Hellström[1], P. Andreasson[1,2], A.G. Andersson[1], K. Angele[2] & L.R. Andersson[1]
[1]Dept. of Fluid Mechanics
[1]Luleå University of Technology
Luleå, Sweden
[2]Vattenfall Research and Development
Älvkarleby, Sweden
E-mail: mikael.hedberg@ltu.se

ABSTRACT

Mathematical modelling of single spillways is well documented in literature. For parallel spillways however, there is a lack of documented, verified, and validated cases. Here, in this article, ANSYS-CFX is used to simulate the flow over three parallel ogee-crested spillways. For mesh size verification, a grid convergence study is performed by Richardson extrapolation. The turbulence model chosen for this simulation is the k-ε model and the volume of fluid method is used to simulate the water-air interface. This article details the models ability to accurately predict flow distribution at the spillways, and the water levels. The mesh is kept relatively coarse at the channel inlet with increased mesh density at the spillways. The results are validated against experimental data from Vattenfall AB, R&Ds laboratories. The geometry and boundary conditions of the experiment are tailored for CFD. The flow rate of each spillway is measured separately with high accuracy, and for several different inlet volumetric flows. The simulation results lie within the error estimates of the measuring tools used in the experiments, within ±1%. The volume flow rate differences between the three outlets is very small, within ±1%.

Keywords: *Validation & Verification, CFD, Spillways*

1. INTRODUCTION

In Sweden hydropower has existed for more than a century, with the majority of the existing plants having been built in the middle of the twentieth century (Yang et al. 2019). With the advent of climate change, and the potential for increased severity of rainfalls the dimensional flow conditions need to be reevaluated for most dams in Sweden. At present physical scale models are often used to evaluate flow characteristics. These models provide accurate results while being costly and time consuming. Computational Fluid Dynamics (CFD) is today a standard tool for use in hydropower industry, and is seen as a potential solution to the problem of evaluating spillway capacity for a large number of dams. There exist several best practice guidelines on how to perform CFD with respect to quality, in Europe ERCOFTACs recommendations (Casey & Wintergerste 2000) and (Sommerfeld et al (2008) are commonly used. For the case of a single spillway, mathematical modelling is well documented (Akoz 2009), (Akoz & Oner 2009), (Salazar et al 2013), (Chatila & Tabbara 2004), (Fadaei-Kermani & Barani 2014). The majority of larger dams in Sweden utilizes more than one spillway in parallel, (two or more gated openings in the dam located in close proximity) in order to provide sufficient spilling capacity (Yang et al. 2019). This study investigates the case of three parallel spillways, in order to evaluate how accurate CFD can predict the spillway flows, and water levels. The spillway flows are measured by flow volume in dm^3/s, while the water levels are compared by creating a surface by use of the volume of fluid method. The water level in the experiment showed 1 mm increase per 1 dm^3/s of inlet flow volume. This relation is used as a criterion for judging the validity of the simulated water levels. For the outlet flows the distribution of the flow volume is the important criteria. As the results of the experiment shows that the flow is very evenly distributed through the three outlets seen in Figure 1, with minor differences. While the simulations show that outlet 1 has the lowest volume flow rate with a relatively large margin, then followed by outlet 3, and with outlet 2 showing the largest volume flow rate. Regarding distribution between spillways, the simulation results are considered to be validated if the percentage of flow through the outlet lies within 1 % of the experimental flow, adjusted for the differing inlet flow. The experimental data presented were gathered for validation use.

2. EXPERIMENTAL SETUP

The flow into the model seen in Figure 1 begin about 20 meters from the honeycomb, the honeycomb inlet can be seen in Figure 2. The flow is measured by an electromagnetic flow meter, which has an accuracy of ±0.5% according to the calibration documents. A tank with a volume of 6 m^3 is located below the outlets, and a redirection ramp seen in Figure 3 is used to lead the water from the

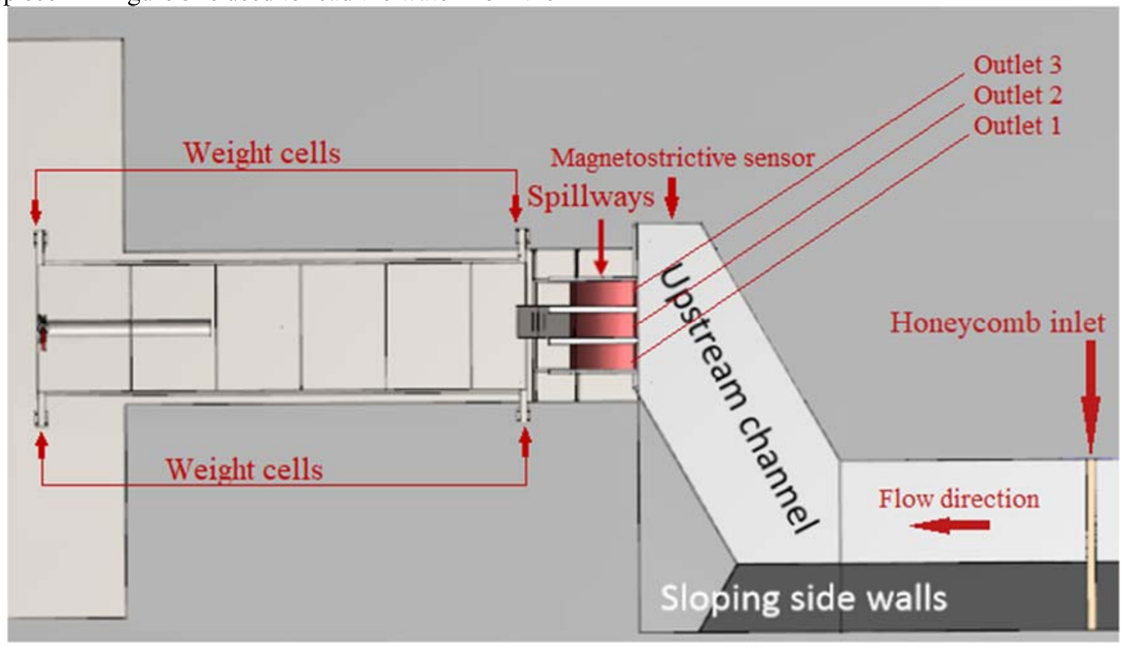

Figure 1: Sketch of experimental setup, showing honeycomb inlet, magnetostrictive sensor for water level measurement, 3 parallel spillways, and water tank with weight cells. The flow direction is from right to left.

different outlets into the tank for measuring. The tank is suspended on 4 weight cells shown in Figure 1, calibrated to measure with an uncertainty of 0.1%.

Figure 2. Picture showing the honeycomb used to smooth the flow, as well as parts of the channel leading from the pump.

The left corner of the model, seen in Figure 3, has a magnetostrictive sensor for measuring of the water level, the location is also shown in Figure 1. Labview is used to gather the data from the different sensors. The flow is then measured by calculating the gradient of a curve fit of the combined mass data from the 4 cells that the tank is suspended upon which can be seen in Figure 1. This dynamic weighing method is well established. The water level measured shows an increase of about 10mm for every 10 dm^3/s of inflow increase over the first case at 90 dm^3/s, and can be seen in Table 3 where it is compared to the simulated water levels.

Figure 3. The three spillways, redirection ramp- and part of the measuring tank.

3. NUMERICAL SETUP

The model is made from CAD blueprints of an experimental setup at Vattenfall R&Ds laboratory in Älvkarleby. ICEM CFD 19.0 was used to create the geometry and to discretize the computational domain into a mesh. The experiment was created for validation purposes, breakpoints were then measured to validate that the built experiment and the model share measurements. As can be seen in Figure 1 the model consists of an inlet channel that widens past three parallel outlets. The outlets were scaled to be at a width of 300 mm. Outlet 1 in Figure 1 is 297 mm wide at the entrance and 304 mm at the exit, the other outlets differed 1 mm at most. The walls are made of stainless steel, and are simulated as smooth walls. The volume of fluid model (Hirt and Nichols 1981) is used to simulate the water-air interactions. For boundaries, the roof and outlet were left as openings to match the conditions of the experiment. The inlet conditions were set with a step function to separate water and air, with an inlet flow volume ranging from 90 to 130 dm^3/s, the profile of the water flow is assumed to be uniform and calm. Both water and air were kept at a temperature of 25ºC in the simulation. As initial conditions for the numerical model a steady state solution was performed. The simulated time was chosen to be 120 seconds to allow the simulation to establish transient behavior which based on observations done during the experiment should take at least 60 seconds. The time used for the dynamic weighing was 25-30 seconds, adding 30 more seconds should result in good data. Then transient simulations were run with a timestep of 0.05 seconds. The data taken from the simulation is the last 500 timesteps resulting in the average flow over 25 simulated seconds. This was done to compare with the experimental measurements which measure diverted water for up to 60 seconds. The first order k-ε turbulence model (Launder and Spalding 1974) was used with a scalable wall function. A High Resolution advection scheme used for the solver.

The mesh put emphasis on the flow at the outlets by giving elements at the spillway parts an order of magnitude denser mesh, identifying it as the critical part for a simulation giving good results. It was generated by filling the model with tetrahedral volume elements, and then replacing the internal volume elements with hexahedral elements by utilizing ICEMs hexacore function, to reduce the amount of nodes.

The solver chosen for this study was the commercial software Ansys CFX 19.2, its governing equations are the Reynolds-averaged Navier-Stokes Eq (1), and the continuity equation Eq (2) for incompressible flow.

$$\frac{\partial U_i}{\partial x_i} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \nu \nabla^2 U_i - \frac{\partial}{\partial x_j}\left(\overline{u_j u_i}\right) \tag{1}$$

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{2}$$

Where U is the mean velocity, P is the pressure in the fluid, ν is the viscosity, ρ the fluid density, $\overline{u_j u_i}$ is the Reynolds stresses. Refinement of the mesh where the water surface was expected to lie was considered, but it could be counterproductive to the purpose of using CFD to predict water level, as it uses information from the experiment to predict the outcome. Automatic mesh refinement was also considered but not applied.

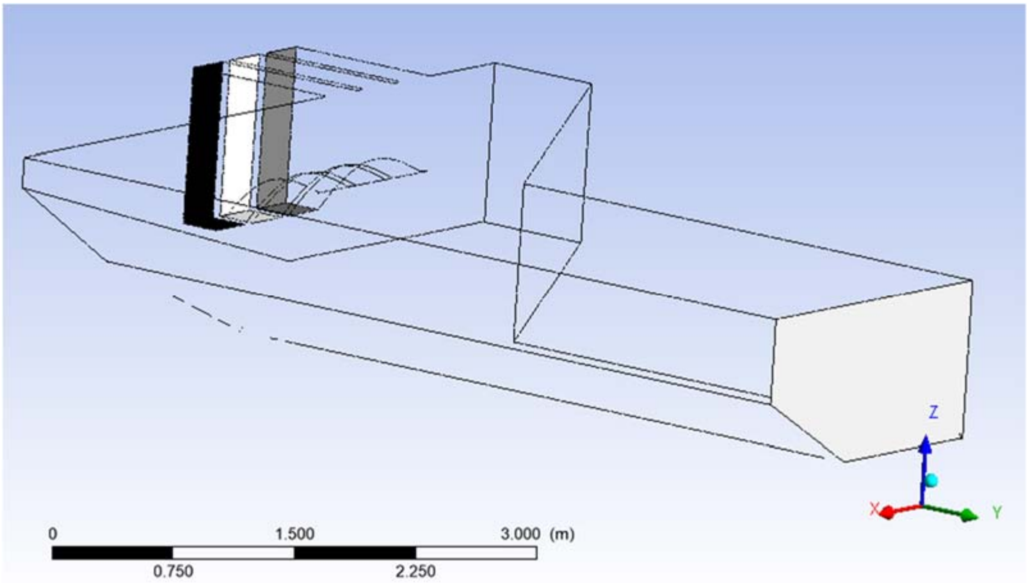

Figure 4. Outlet 1 shown with black, outlet 2 in white, outlet 3 in dark grey, and inlet shown in light grey.

4. GRID CONVERGENCE STUDY

The volume flow rate through the outlets were used as a parameter of interest for the grid convergence study. The average flow for the final 25 seconds of simulation was used as the variable of interest for the three different outlets. The complete data can be seen in Table 1. The values for the apparent order of accuracy of the calculations p, are high for the High Resolution advection scheme used, but according to (Celik et al. 2008) a p value above the formal order of the scheme used is not to be taken as unsatisfactory. The extrapolated errors are all low, and the grid convergence index is also low for all the outlets. The data used for the extrapolation is from the case with an inlet volume of 120 dm^3/s. The value for p used was 1, to not underestimate the error. The grid chosen to be used is the N_3 mesh. The numerical uncertainty calculated with the grid convergence method, CGI_{fine}, for the chosen grid is below 1% for all three outlets.

Table 1. Richardson extrapolation with regards to the flow through the different outlets.

Richardson extrapolation	ϕ= Flow through outlet 1 [dm^3/s]	ϕ= Flow through outlet 2 [dm^3/s]	ϕ= Flow through outlet 3 [dm^3/s]
N_1,N_2,N_3 [mesh nodes]	67948,197849,1322910	67948,197849,1322910	67948,197849,1322910
r_{21} [grid size number]	1.428	1.428	1.428
r_{32} [grid size number]	1.884	1.884	1.884
ϕ_1 [dm^3/s]	39.2847	40.3617	40.2868
ϕ_2 [dm^3/s]	40.0391	40.0945	39.8162
ϕ_3 [dm^3/s]	39.7898	40.2292	39.994
p [local order of accuracy]	5.4314	4.1740	4.9692
ϕ_{ext}^{32} [dm^3/s]	40.1963	40.0096	39.7041
e_a^{32} [approximate error]	0.62 %	0.34 %	0.45 %
e_{ext}^{32} [extrapolated error]	0.39 %	0.21 %	0.28 %
GCI_{fine} [uncertainty]	0.8805 %	0.4751 %	0.6315 %

5. RESULTS AND DISCUSSIONS

A comparison is made of the differences in the inlet flows, and how it changes the surface level. For every increase of flow by 10 dm^3/s the water level increases by about 10 mm, these measurements are from the magnetostrictive sensor shown in Figure 1, the same difference in water level can be observed in the simulations. The simulated water level used for comparison is acquired by creating an isosurface through the cells with a volume of fluid fraction equal to 0.5, a height contour is then projected unto this surface, examples for four different cases can be seen in Figure 5. This agrees with the values of the water level given by the magnetostrictive sensor in the experiment, the averaged values from the sensor can be seen in Table 3. For the flow in each outlet, comparing the simulated data to the experiments both show that the differences are tiny. The simulations show larger and more consistent differences. The percentages of the total flow can be seen in Table 2 below, the experimental data is from Vattenfall AB, R&Ds laboratory. The discrepancies that can be observed, for example the experiment case for outlet 2 for an inlet flow of 100 dm^3/s deviates by more than 2%. This is could be a result from problems occurring during the experiment as it is a large system and if the pumps shut off from lack of water, it takes some time before it is noticeable without looking at the data graphs.

Table 2. Outlet flow as percent of inlet flow, for simulated cases the stated inlet flow volume is used. For the experiments, measured average inlet flow is used. Values in parentheses show difference between simulation and experiment.

Inlet flow volume:	Outlet 1 Simulated	Outlet 1 Experiment	Outlet 2 Simulated	Outlet 2 Experiment	Outlet 3 Simulated	Outlet 3 Experiment
90 [dm^3/s]	33.16	33.58 (-0.41)	33.45	33.64 (-0.22)	33.41	33.52 (-0.12)
100 [dm^3/s]	33.15	32.67 (0.48)	33.47	31.09 (2.23)	33.34	33.81 (-0.45)
110 [dm^3/s]	33.18	32.98 (0.19)	33.49	33.76 (-0.26)	33.34	33.24 (0.15)
120 [dm^3/s]	33.18	33.56 (-0.38)	33.54	33.12 (0.4)	33.35	33.18 (0.12)
130 [dm^3/s]	33.23	33.60 (-0.27)	33.56	33.93 (-0.3)	33.24	33.82 (-0.54)

For outlet 1, the outlet closest to the inlet channel the flow is lowest for all cases followed by the outlet furthest from the inlet leaving the middle outlet with the highest flow. The differences between outlet 2, and outlet 3 are very small and will be hard to validate. The differences between outlet 2, and outlet 1 are larger. Overall the simulations reflect the experiments well. It is unfortunate that the experiment did not show larger differences between the different outlets as the differences are within the margin of error of the instruments used. Larger differences would have been more interesting as a validation case. In the future changes could be made to the experiment to try to increase the differences between them.

The differences between the flow cases are compared for the water levels, as the simulated model is based on CAD blueprints which included supports that lifts it 250mm from the floor, it does not show the same level as the experiment. The simulated water level is recorded by creating an isosurface through the cells with a volume of fluid fraction equal to 0.5. A contour of the height is then created on this surface showing the height of the surface and how it changes in the model, the largest variations in this surface were 5mm (magnitude from lowest value to highest), excluding the areas close to the outlet and inlet. The largest contour area was chosen as the average water level to be used for comparison with the experimental data. The water level is shown for some of the cases in Figure 5. For the simulated cases the threshold of the outlets lie at 600mm, while in the experiment the threshold is at 350mm. Adjustments to the data are made to compare them and can be seen in Table 3 below. Comparing the adjusted simulation with the adjusted experiment the results are similar.

Table 3. Simulated and adjusted water level, and measured water level.

Simulated inlet flow volume:	Experiment inlet flow volume	Simulated water level:	Adjusted simulation:	Experiment water level:	Adjusted experiment:
90 [dm^3/s]	89.47 [dm^3/s]	780 [mm]	180 [mm]	528.17 [mm]	178.17 [mm]
100 [dm^3/s]	99.54 [dm^3/s]	790 [mm]	190 [mm]	538.37 [mm]	188.37 [mm]
110 [dm^3/s]	109.18 [dm^3/s]	800 [mm]	200 [mm]	549.60 [mm]	199.60 [mm]
120 [dm^3/s]	119.47 [dm^3/s]	810 [mm]	210 [mm]	558.76 [mm]	208.76 [mm]
130 [dm^3/s]	129.73 [dm^3/s]	820 [mm]	220 [mm]	569.29 [mm]	219.29 [mm]

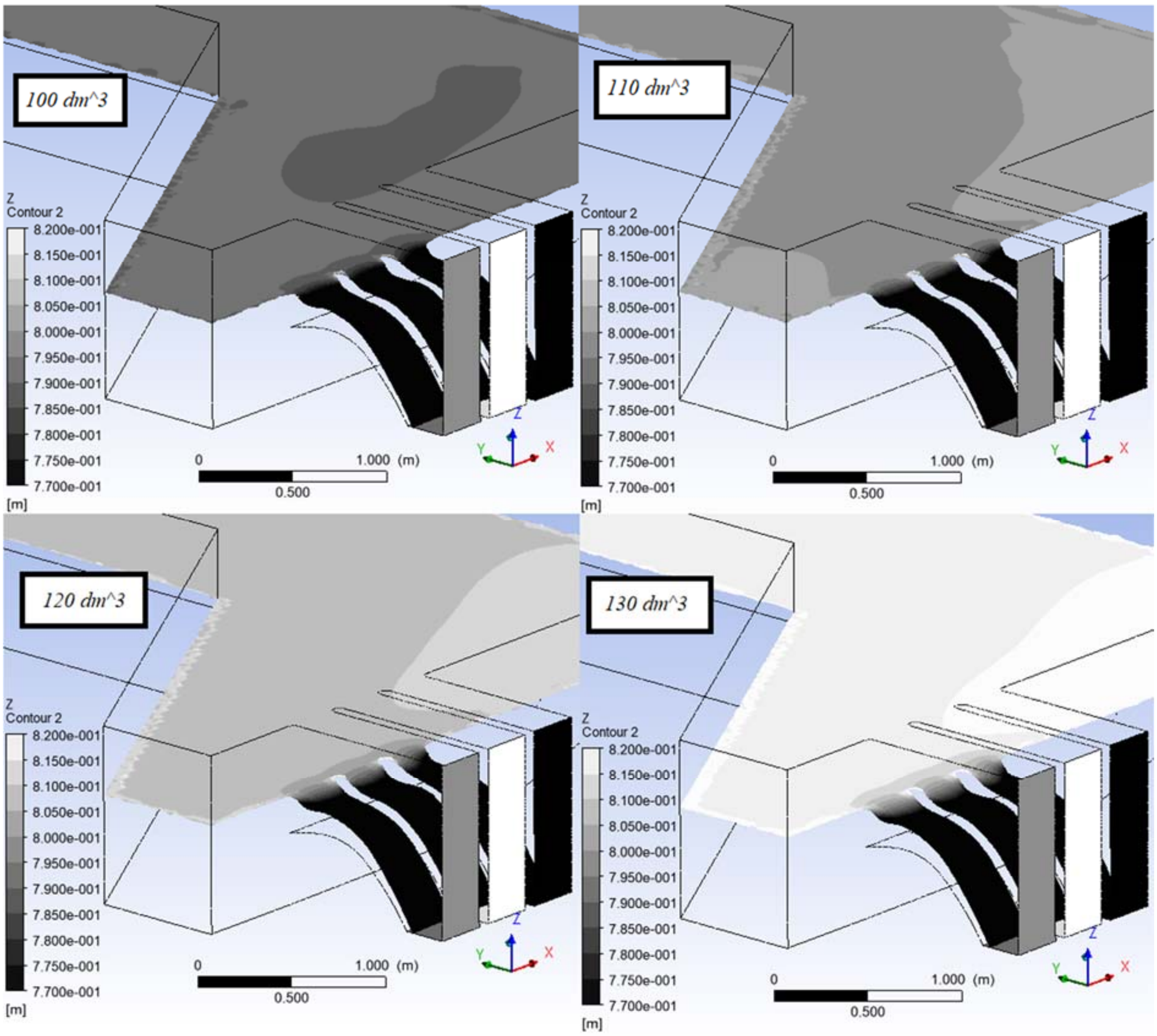

Figure 5. Greyscale contour on isosurface of 0.5 volume fraction showing the 10 mm change observed in the experiment. Inlet flow is written in the corresponding quadrant.

The results are good and indicate that CFD simulations with the k-ε turbulence model can work well to capture the flow over parallel spillways. Looking at the streamlines in the simulation an interesting fluid movement is shown as the water moving through outlet 3 moves beneath the other outlets, this and velocity streamlines can be seen below in Figure 6.

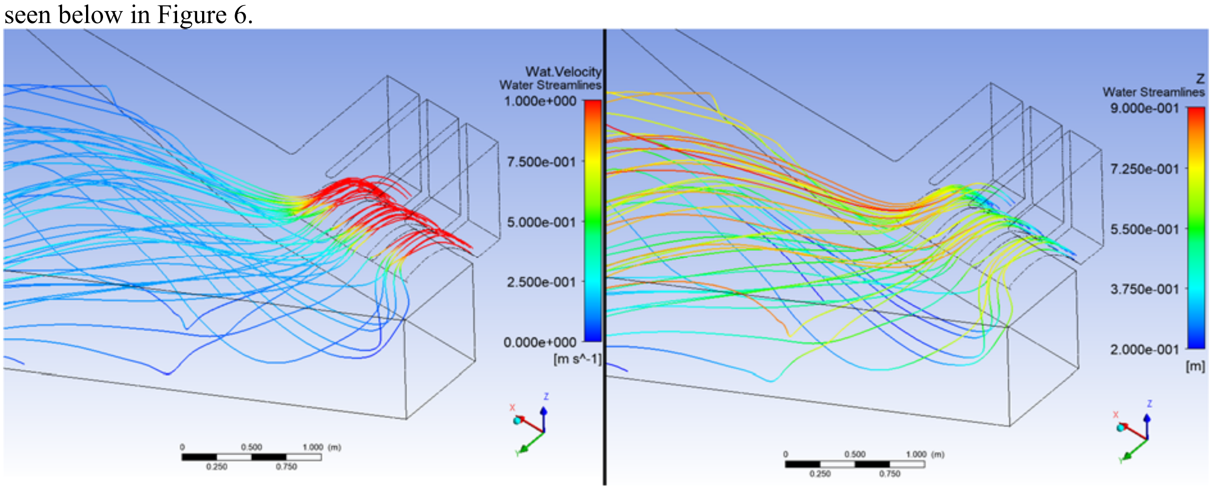

Figure 6. Streamlines showing velocity and height.

6. CONCLUSIONS

CFD simulations of spillways seem to be very robust with regards to the distribution of flow, and the volume of fluid method can predict changes in the water surface levels, for the simulations done the water level lies within 1% of the surface of the experiment. Based on what has been simulated, CFD could be used instead of physical models for the specific purpose of investigating volume flow rate for parallel spillways. However, in this particular setup, the flow approaching the spillways was calm with small velocity head. A more pronounced differentiation in approach conditions would differentiate flow distribution more. This is a more challenging case, for which CFD-performance remains to be investigated. Internal flow and time dependent variations have not been looked at, and would likely require finer meshes to fully resolve, better models, and more advanced measuring techniques. Future work could be done with adaptive meshing to see what benefits it could bring compared to computational costs.

7. ACKNOWLEDGMENT

The research presented has been carried out as a part of "Swedish Hydropower Centre / SVC". SVC has been established by the Swedish Energy Agency, Energiforsk and Svenska Kraftnät together with Luleå University of Technology, Chalmers University of Technology, The Royal institute of Technology and Uppsala University.

8. REFERENCES

Akoz MS, Kirkgoz MS, Oner AA, (2009), Experimental and numerical modeling of a sluice gate flow, *Journal of Hydraulic Research*, 47(2), 167-176.

Akoz MS, Oner AA, (2009), Numerical modeling of flow over a chute spillway*, Journal of Hydraulic Research*, 47(6), 790-797

Casey M, Wintergerste T, (eds.), (2000)*, Special Interest Group on Quality and Trust in Industrial CFD: Best Practice Guidelines*, European Research Community on Flow, Turbulence and Combustion (ERCOFTAC).

Celik IB, Ghia U, Roache PJ, Feritas CJ (2008). Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *Journal of Fluids Engineering,* 130(7), 338-344

Chatila, J. Tabbara M, (2004). Computational modeling of flow over an ogee spillway, *Computers & Structures* 82, 1805-1812.

Fadaei-Kermani, E, Barani GA, 2014, Numerical simulation of flow over spillway based on the CFD method, *Scientia Iranica*, 21(1), 91-97.

Hirt CW, Nichols BD (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *J. Computational Physics*, 39(1), 201–225

Launder BE, Spalding DB (1974). The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering,* 3(2), 269-289

Salazar F, Morán R, Rossi R, Oñate E, (2013), Analysis of the discharge capacity of radial-gated spillways using CFD and ANN – Oliana Dam case study, *Journal of Hydraulic Research*, 51(3), 244-252

Sommerfeld, B, Van Wachem R, Oliemans R, (eds.), (2008)*, Special Interest Group on Dispersed Turbulent Multi-Phase Flow: Best Practice Guidelines*, European Research Community on Flow, Turbulence and Combustion (ERCOFTAC).

Yang J, Andreasson P, Teng P, Xie Q (2019). The Past and Present of Discharge Capacity Modeling for Spillways—A Swedish Perspective, *Fluids*, (1), 10