V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal, 14–17 June 2010

SIMULATION OF FREE SURFACE FLOW IN A SPILLWAY WITH THE RIGID LID AND VOLUME OF FLUID METHODS AND VALIDATION IN A SCALE MODEL

Anders G. Andersson[†,*], Kristoffer Lundström[†], Patrik Andreasson[†,††], T. Staffan Lundström[†]

[†]Division of Fluid Mechanics
Luleå University of technology, SE-971 87 Luleå Sweden

[††]Vattenfall Research and Development, SE-814 70 Älvkarleby Sweden

[*]Corresponding author e-mail: aneane@ltu.se

Key words: Volume of fluids, Rigid lid, CFD

Abstract. *Simulations on the spilling from a dam were performed and compared to experimental results from a physical scale model. Both mechanical and acoustic methods to measure the velocity were used. The model has three gates leading into the spillway that can be maneuvered separately. At first two of the gates were closed and the inlet flow was high enough to get a fully wetted outlet at the third gate. This case was simulated with a rigid lid approximation since the water surface was considered to be plane. The water surface level was taken from the scale model. In the second case, all three gates were open resulting in a free water surface through all the gates to the spillway. This case was simulated with the Volume of Fluids method were both water and air phase were considered. Water levels, velocities and the shape of the water surface were compared between simulations and experiments. The simulations capture both qualitative features such as a vortex near the outlet and show good quantitative agreement with the experiments.*

1 INTRODUCTION

The estimation of spillway capacity in a design phase of a dam is costly. Scale model attempts are often used to get results with good accuracy but at a high cost. There are semi-empiric models that give faster answers but the cost of safety margins often exceeds that of scale attempts. To use CFD-models to estimate spillway capacity can be an alternative that may provide high accuracy at a lower cost, however validations of CFD-models are still necessary. In this particular case simulations on a scale model dam were performed with the free surface of the water in focus and the results were compared to experiments. Both a rigid lid approach and the Volume of fluids (VOF) method were applied. The latter method has been used to simulate flow over spillways on a model scale by 2D [1, 2] and 3D set-ups [3], showing good conformity between simulations and experiments.

2 EXPERIMENTAL

The flow in a down-scaled model (1:50) of the Höljes dam located in the river Klarälven in the central part of Sweden was studied. The reservoir was constructed in concrete while the spillway and the gates were built in stainless sheet metal. The flow was driven by a large pump system and measures were taken to obtain a uniform flow into the model. The velocity profile was measured at two depths in a single measuring plane, perpendicular to the flow, upstream of the gates, see figure 1. An acoustic measuring probe was used giving all three velocity components. The sampling rate was set to 25 Hz for a period of 24 s in each measuring point and time averaged results were used for comparison to simulated results. The probe was mounted on a ladder on top of the model, see figure 2. This setup enabled stable and precise measurement at points located 0.3 m apart (corresponding to one step on the ladder).

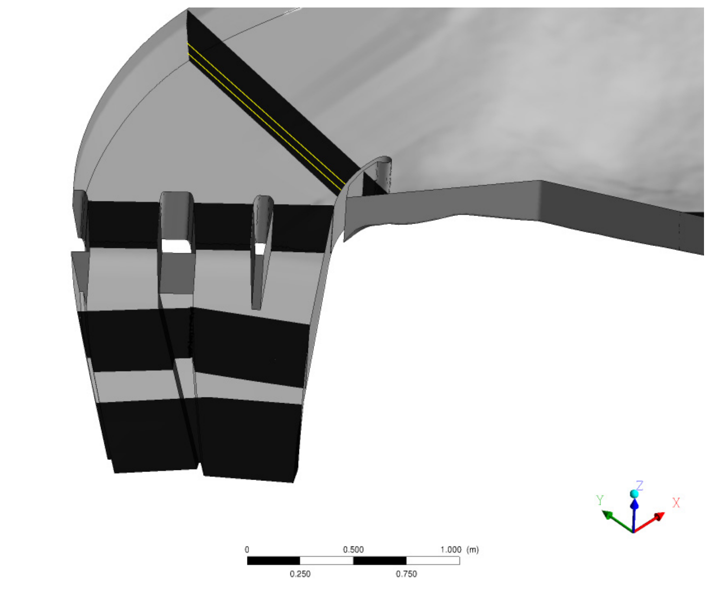

Figure 1: Measuring plane with lines and the three measuring sections

The water level profile was measured at three sections in the spillway indicated in dark grey in figure 1 with a point meter with 0.01 m spacing between measuring points.

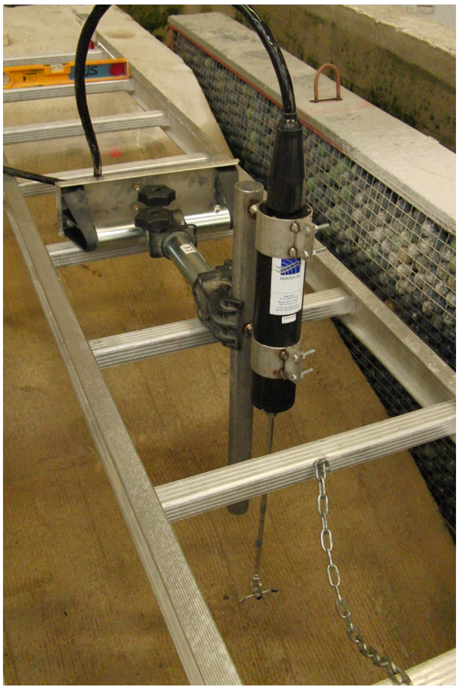

Figure 2: Acoustic measuring probe mounted in place

The discharge q [m^3/s] through each gate was approximated by dividing each gate into segments and summarizing the discharge of all segments with the following formula:

$$\sum q_n = a_n \cdot \frac{(U_1 + U_2)}{2} \tag{1}$$

where a_n is the area of the n:th segment of respective gate and $U_{1,2}$ are the velocities, as measured in the middle of each segment with a handheld hydrometric paddle-wheel at two depths d_1 and d_2 see figure 3.

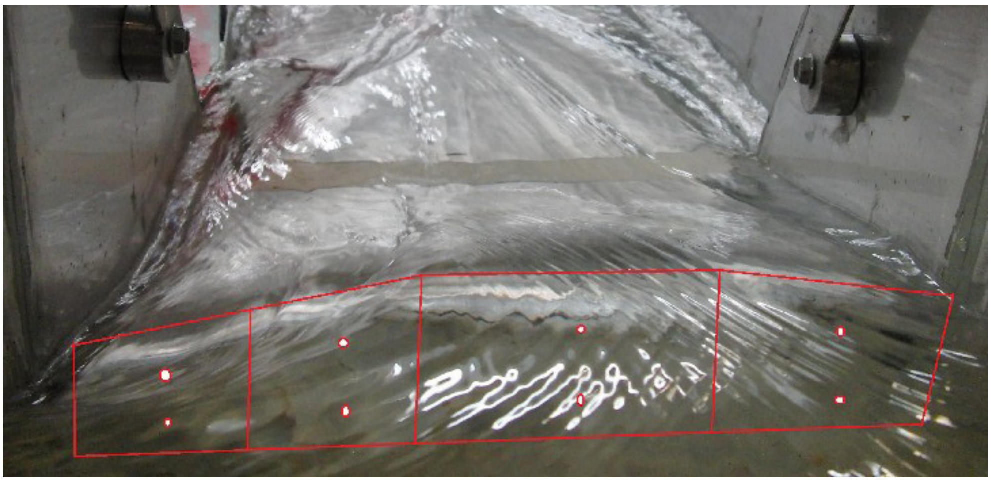

Figure 3: Approximation of surface level and measuring points

3 NUMERICAL SETUP

The geometry for the simulations was created by laser scanning the reservoir in the physical down-scaled model and the spillway area was created from 2D drawings. The point cloud obtained by the laser scanning was used to create a bottom surface in the software Imageware 13. The spillway entrance and the spillway segments were modeled in NX5 from UGS, see figure 4. The numerical grids for the rigid lid and free surface (VOF) model were generated in Ansys Icem CFD as tetrahedral elements with prism elements close to the wall to improve the y+ value. A global smoothing of the mesh with regard to the aspect ratio and the minimum angle was applied to improve mesh quality.

The model had three outlets that could be maneuvered separately. For the first case studied only one of the gates was partly opened giving a fully wetted outlet, i.e. the free surface was located above it. The spillway was not included in the numerical models for this case and the surface was modeled with as well a rigid lid with zero friction as a free surface with the VOF method. The rigid lid approximation is likely to be valid given that the deformation of the water surface is less than 10% of the depth of the channel [4]. In the second case all gates were kept open giving a free surface into the spillway. This case was exclusively simulated using the VOF method. The VOF method introduces the volume fraction field F, which for each element in the computational grid contains the fraction of that elements volume that is occupied by a specific fluid, see [5, 6]. In this case a volume fraction value of one is defined as a pure water element and a value of zero is a pure air element. The interface between the two fluids is then considered to be all elements between zero and one volume fraction.

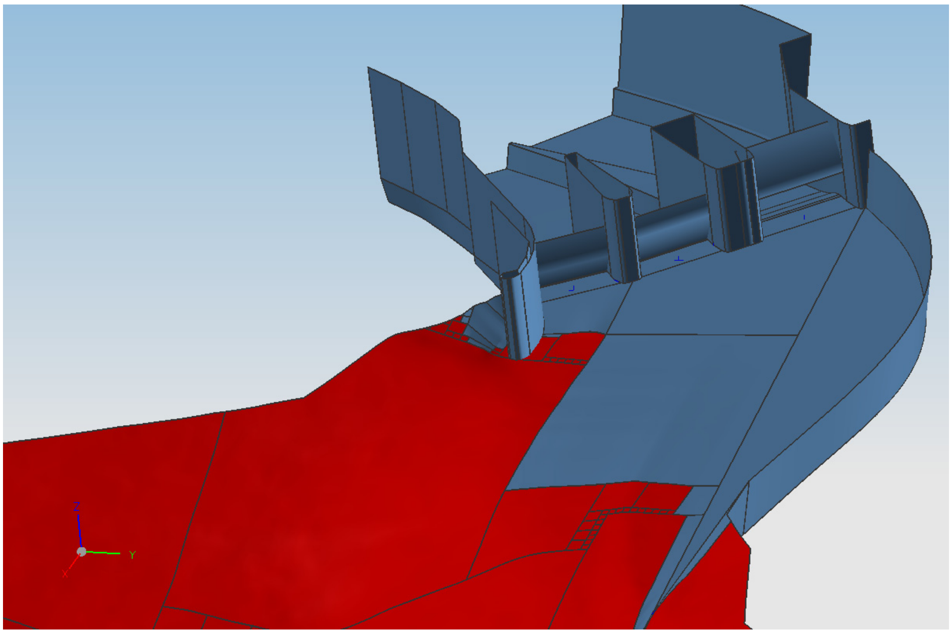

Figure 4: Geometry with one gate open

The commercial software Ansys CFX12 was used for all simulations. Computationally demanding simulations were run on a parallel solver with double precision on a 64-bit Linux cluster, which has proven to provide excellent parallelization [7]. Reynolds-averaged Navier-Stokes equations were solved. The turbulence models used was k-ε with scalable wall functions and *SSG*, which is a Reynolds stress model. The High Resolution scheme was used for both flow and turbulence equations. The High resolution Scheme uses a close to second order solution in areas with low variable gradients and in areas where the gradients change sharply it will be close to a first order solution to prevent over- and undershoots and maintain robustness [8]. The convergence criteria for the RMS residuals were set to 10^{-6}. The nodes with the highest maximum residuals were all located near the reservoir wall and were considered to have no effect on the solution. Velocity and pressure was monitored in several points in the domain to guarantee stable conditions. Two flows rates, Q, were used, 0.034 m^3/s for the case with one gate open and 0.097 m^3/s when all gates were fully open. The inlet boundary condition was approximated as a plug profile with a given velocity. The surface level of water was given an initial value close to the water surface level measured in the physical down-scaled model. The surface is then allowed to adjust itself during the calculation. The bottom surface was modeled both as a smooth surface and with a roughness length of 3 mm.

A grid dependence study was carried out for a case with free surface, all gates open, spillway present, Q = 0.097m^3/s, a surface roughness of 3mm and with the k-ε turbulence model on four meshes, N1-N4 with 1.5M, 2.8M, 5.3M and 8.8M nodes, respectively.

4 RESULTS

The absolute velocity along a line at a depth of 57 mm in the measuring plane defined in figure 1 was compared for the meshes N1-N4 in the grid study. The main result is that the velocity calculated with the coarsest grid differs significantly from the other grids and then only close to the walls, see figure 5 where also the absolute differences between the finest grid and the other grids are presented.

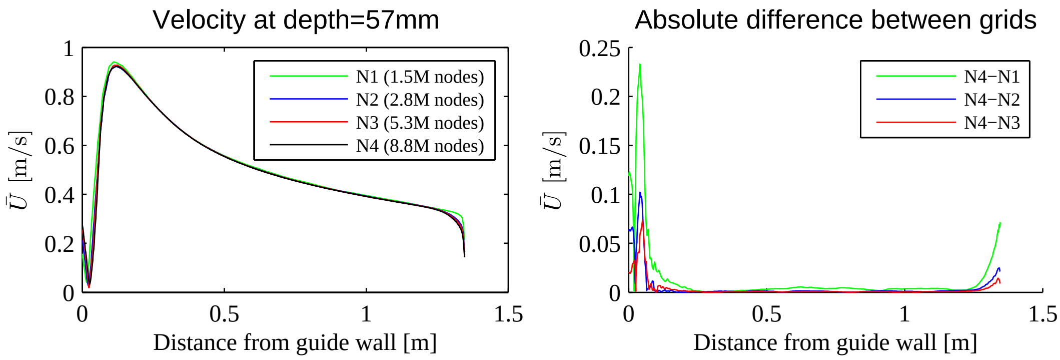

Figure 5: Mesh dependence of velocity profile

Special attention was also given to the diffusion at the interface between the two fluids. The absolute volume of the elements with a volume fraction of water between 0.1 and 0.9 was calculated for all meshes. Richardson extrapolation was used on the three finest meshes according to [9]. The apparent order p obtained was 1.52 and the extrapolated value was V = 0.059 m^3, see figure 6. A mesh adaption methodology implemented in Ansys CFX was also applied with regard to the volume fraction at the interface between the fluids. The mesh is then refined in areas where the selected parameter has large variations [8] i.e. the volume fraction at the water surface. The volume of diffuse elements was then decreased but the overall mesh quality was impaired resulting in inadequate numerical convergence which affected the final results. Hence mesh adaption was not used in the final simulations.

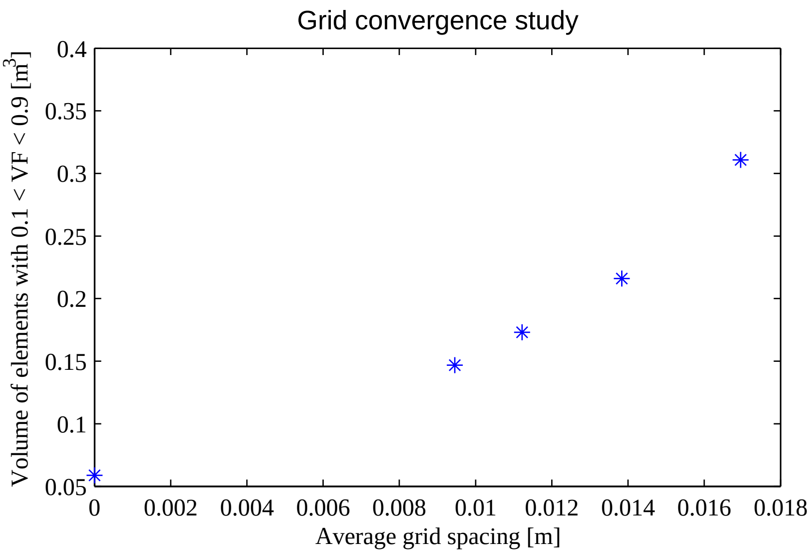

Figure 6: Richardson extrapolation of numerical diffusion

Given the results of the grid study, the results from the coarsest mesh were considered too crude and were discarded. The rest of the results in this study are therefore generated using the 2.8M nodes mesh which was giving high enough accuracy with reasonable computational time.

For the case with one outlet open, the water depth in the rigid lid model was set to be the same as in the scale model i.e. Z = 1.665 m. The surface level calculated with the VOF-model was evaluated in two cross-sections of the reservoir. The maximum difference in water level was ~5 mm and the average value in the cross sections was Z = 1.659 m. A clear qualitative feature of the flow is a vortex that is created at the left edge of the opened gate. This feature is captured in the simulations both with rigid lid approach and VOF method; figure 7 shows the water surface defined as an isosurface with a volume fraction of 0.5 at the outlet for the VOF simulation and the same position in the experiments.

Figure 7: Vortex at the outlet as simulated with the VOF method to the left and as observed in the physical down-scaled model.

The velocity in the previously defined measuring plane was evaluated quantitatively at different two depths. The VOF gives a smoother velocity profile which can be seen in figure 8.

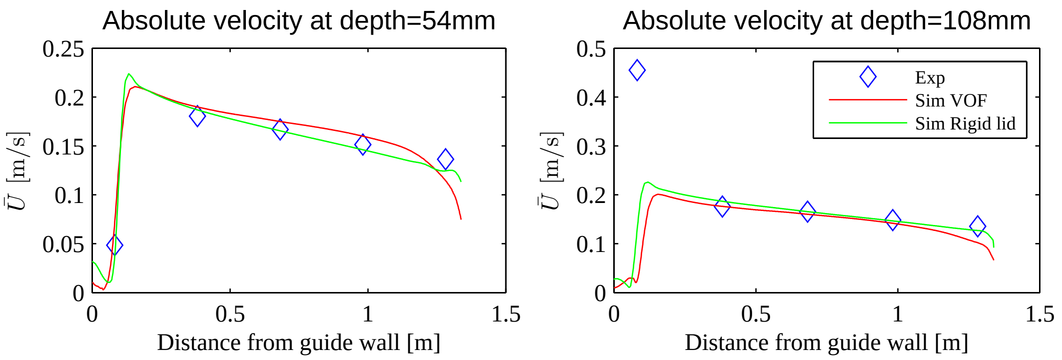

Figure 8: Comparison between the Rigid lid approach and and the VOF method

Both models show good agreement with the experiments but seem to under predict the velocity close to the guide wall closer to the bottom. The reason for the high velocity closest to the guide wall in the measurements is that there is a pulsating motion of the water surface between the guide wall and the dam body in the scale model. This pumping motion causes a periodic increase in the velocity close to the bottom of the channel close to the guide wall. To capture such a feature in simulations, transient analysis must be applied.

For the case with all three outlets open, the simulations show good qualitative resemblance to the scale model, the separation zone at the guide wall is captured as well as the behaviour of the water surface through the outlets as seen in figure 9.

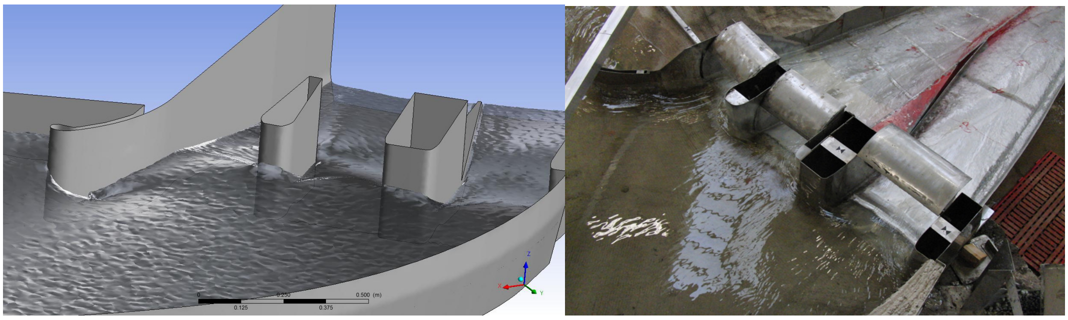

Figure 9: Visualization of water surface as obtained with the VOF method to the left and as observed in the physical down-scaled model to the right

The velocities in the measuring plane are shown in figure 10. The two turbulence models show identical behavior except close to the guiding wall where the *SSG* model gives a higher maximum velocity.

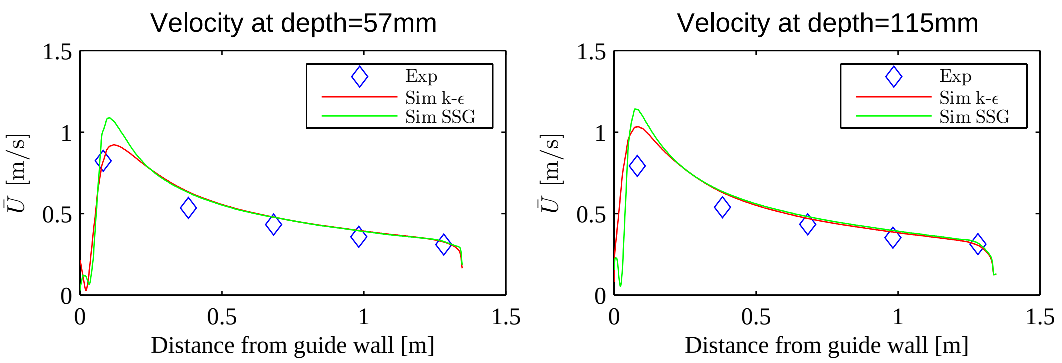

Figure 10: Comparison between *k-ε* turbulence model and *SSG* turbulence model

The simulated water level in the spillway is close to the measured water level. In figure 11 the surface profiles for the simulation and the measured surface profiles are shown. The simulation over predicts the water depth going through the gates and slightly under predicts the water depth in the spillway. In the physical down-scaled model the water is flowing over the dividing wall close to the outlet which is not captured in the simulations. The measured profiles also show a larger influence of cross-waves than the simulations. This might be an effect of the numerical diffusion at the surface that smears out small deformations of the surface in the spillway.

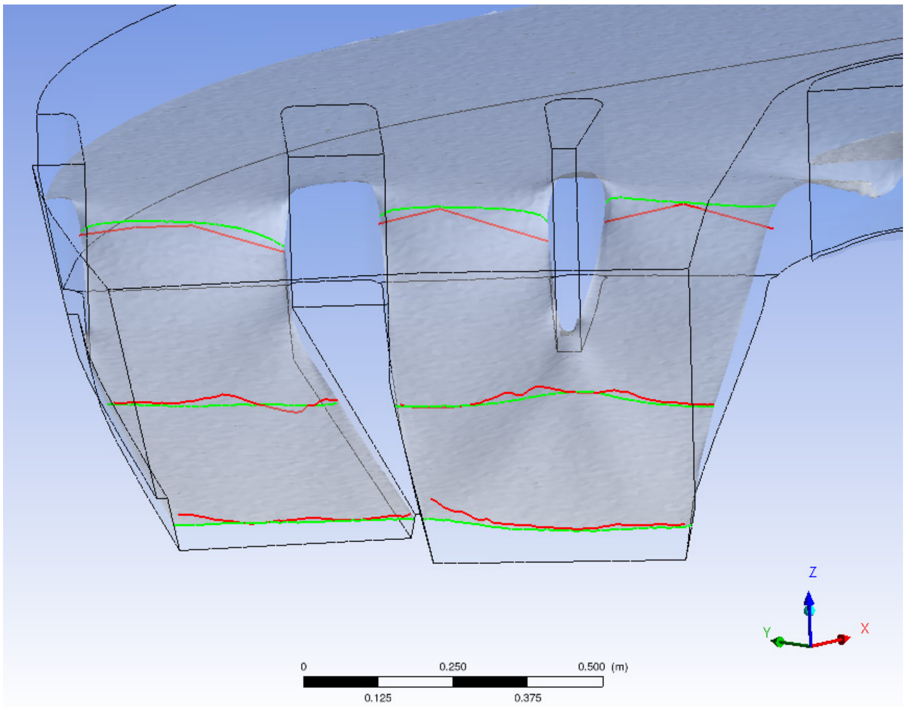

Figure 11: Surface levels in spillway

The water surface with the two different turbulence models matches very closely except at the separation at the guiding wall as seen in figure 12. Unfortunately the measurements do not supply enough data in this region to conclude which model gives the best results.

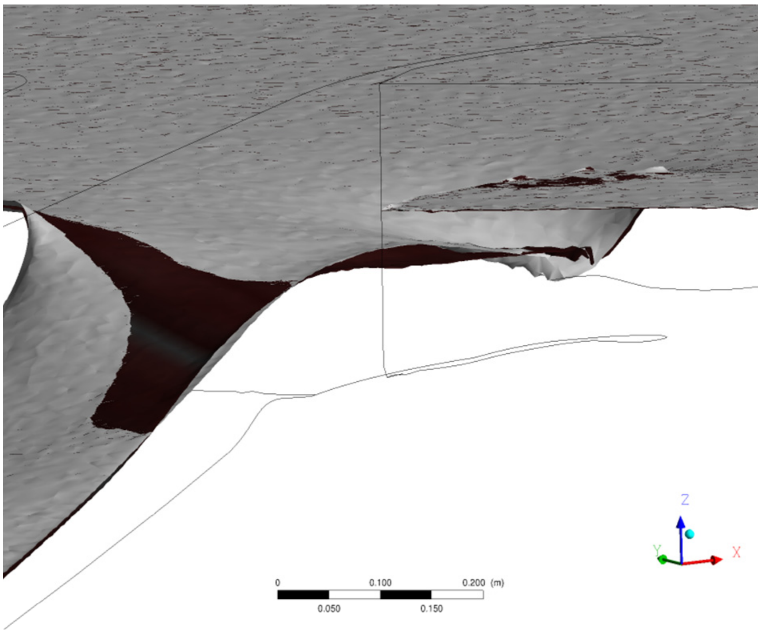

Figure 12: Water surface for k-ε (light) and SSG (dark)

The discharge through each outlet was calculated and compared to the measurements. Both simulations get good conformity for the left gate but the volume flow through the middle gate is higher compared to the right gate than for the measured discharge, see Table 1.

	Left [m^3/s]	Middle [m^3/s]	Right [m^3/s]
Measured	0.0312	0.0291	0.0365
Simulation k-ε	0.0305 (-2.24%)	0.0314 (7.90%)	0.0349 (-4.38%)
Simulation SSG	0.0309 (-0.96%)	0.0311 (6.87%)	0.0347 (-4.93%)

Table 1: Discharge through spillway gates and deviation of simulations compared with measurements

5 CONCLUSIONS

- Simulations show good qualitative agreement with scale model attempts, vortexes and surface deformations are captured well with the volume of fluids method.
- The two turbulence models show identical behavior except in regions with separation.
- The mean water levels in the spillway obtained from the simulations are captured well but show a smoother shape of the surface than measurements. The simulations under predicts important physical features such as cross-waves in the spillway.
- The distribution of discharge through the different gates for the simulations is close to that of the measurement in the scale model but the simulations over predict the discharge through the middle gate compared to the right gate.

6 ACKNOWLEDGEMENT

The research presented was carried out as a part of "Swedish Hydropower Centre - SVC". SVC has been established by the Swedish Energy Agency, Elforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University. www.svc.nu. It was also partly sponsored by Vattenfall Vattenkraft AB.

REFERENCES

[1] B. M. Savage and M. C. Johnson, Flow Over Ogee Spillway: Physical and Numerical Model Case Study. *Journal of Hydraulic Engineering.* **August**, pp. 640-649 (2001)

[2] M. Tabbara, J. Chatila and R. Awwad, Computational simulation of flow over stepped spillways. *Computers and Structures.* **83**, pp. 2215-2224 (2005)

[3] D. Ho, K. Boyes, S. Donohoo and B. Cooper, Numerical Flow Analysis for Spillways. *43rd ANCOLD Conference.* (2003)

[4] J. F. Rodriguez, F. A. Bombardelli, M. H. Garcia, K. M. Frothingham, B. L. Rhoads and J. D. Abad, High-resolution Numerical Simulation of Flow Through a Highly Sinuous River Reach. *Int. Water Resources Management* **18** 177–199, (2004)

[5] C. W. Hirt and B. D. Nichols, Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. *Journal of Computational Physics.* **39**, pp. 201-225 (1981)

[6] R. Scardovelli and S. Zaleski, Direct Numerical Simulation of Free-surface and Interfacial Flow. *Annu. Rev. Fluid Mech.* **31**, pp. 567-603 (1999)

[7] J. G. I. Hellström, B.D. Marjavaara, T.S. Lundström," Redesign of a Hydraulic Turbine Draft Tube with aid of High Performance Computing" *Advances in Engineering Software,* **38** (5), 338-344 (2007)

[8] Ansys, *CFX12 Documentation*

[9] I. B. Celik, Procedure for Estimation and Reporting of Discretization Error in CFD Applications. *Internal Report, Mechanical and Aerospace Engineering Department West Virginia University, Morgantown WV (USA)* (2005)