

Investigation of Spillway Behaviour under Increased Maximum Flood by Computational Fluid Dynamics Technique

D.K.H. Ho, K.M. Boyes and S.M. Donohoo

Advanced Analysis Group
Worley, North Sydney, New South Wales, 2060 AUSTRALIA

Abstract

Early dam structures were designed and built with limited hydrological information. As such many existing spillway structures are under-sized to cope with the revised probable maximum flood levels. Potential problems such as the generation of lifting pressure and cavitation over spillway crest under flood condition will be encountered. Historically, scaled physical models have been constructed in hydraulic laboratories to study these behaviours, but they are expensive, time-consuming and there are many difficulties associated with scaling effects. Today, with the advance in computer technology and more efficient computational fluid dynamics (CFD) codes, the behaviour of hydraulic structures can be investigated numerically in reasonable time and expense.

This paper describes the CFD modelling of spillway behaviour under rising flood levels two- and three-dimensionally. The results have been validated against published data and good agreement was obtained. The same technique was applied to investigate a spillway structure on the largest concrete gravity dam in Australia.

Introduction

Since the early 1950s, the US Army Corps of Engineers Waterways Experiment Station (WES) has been studying the behaviour of flow over spillway using physical models [1]. A series of hydraulic design charts are available for engineers to design spillway profile for any given design flood levels [9].

The use of CFD technique in analysing flow over spillway is relatively recent [3,7 & 8]. Early difficulties involving solution convergence and moving mesh/grid to track water surface were reported. Nowadays, more efficient CFD codes can solve the Navier-Stokes equations in three-dimensions, and they also have a number of turbulence models to choose from. Defining geometry and meshing in three-dimensions has been simplified – some codes can even transfer geometry from other drafting and/or computer-aided engineering programs.

In spillway design, the profile is designed such that when water flows over the spillway structure under the maximum flood it will not cause adverse effects such as cavitation at the crest and further downstream. Ideally, the spillway surface should experience just the atmospheric pressure under the design head. When the reservoir level is below this flood level, the pressure over the spillway will be above atmospheric. When the reservoir level is well above the design head, sub-atmospheric (negative gauge) pressure will occur along the spillway crest that may damage the concrete face of the spillway due to cavitation, and undesirably affects other components including gate structures.

A majority of dams and their associated hydraulic structures in Australia were designed in the early 1950s and 1960s to cope with the then design floods. More reliable long-term hydrological data have been gathered and processed since. In many cases, the

revised maximum flood magnitudes have increased. In order to come up with the most optimum remedial design, many dam owners/authorities need to consider the most cost-effective way to investigate the behaviour of spillway flow under increased maximum flood. In the past, the use of physical scaled model was the only investigation method. Now, the use of numerical methods is attractive in terms of lower cost and substantially reduced analysis time.

In this investigation a standard WES spillway profile under various flood conditions was analysed. Both two-dimensional and three-dimensional models were studied. The computed results were compared with the published data for validation purposes and thus have provided a level of confidence in applying the CFD technique in future studies.

Numerical Calculation Methods

The CFD code used for the present study was FLOW-3D, which solves the Navier-Stokes equation by the finite difference method. The algorithm is an extension method based on the SOLA method that was developed at the Los Alamos National Laboratory by Hirt et al [4]. The Volume of Fluid (VOF) method is used for computing free surface motion [5].

All the governing differential equations such as continuity and momentum equations are formulated with area (2D) and volume (3D) porosity functions. This formulation, FAVOR (Fractional Area/Volume Obstacle Representation) is used to model complex geometric regions [6].

Any complex obstacle geometry can be represented using the FAVOR technique. The portion of volume (or area in 2D) occupied by the obstacle in each cell (grid) is defined at the beginning of the analysis. The fluid fraction in each cell is also calculated. The continuity equation, momentum equation, or transport equation of fluid fraction is formulated using the FAVOR function. A finite difference approximation is used for discretisation of each equation. Unlike some finite element/volume or boundary fitting grid methods, this meshing technique does not require re-meshing and would not induce any mesh distortion during transient analysis. Hence an accurate solution algorithm can be applied easily.

The basic algorithm for advancing a solution in one time increment consists of the following three steps [2]:

- Step 1: Compute velocities using the initial conditions or previous time-step values for all advective, pressure, and other accelerations based on the explicit approximations of the momentum (Navier-Stokes) equations.
- Step 2: Adjust pressures to satisfy the continuity equation.
- Step 3: Update the fluid free surface or interface to give the new fluid configuration based on the volume of fluid.

Two-Dimensional Spillway Model

A WES spillway model without piers is considered. This was selected for the analysis because the measured results were not influenced by any three-dimensional effects as the edges were far from the region of interest. Hence, this model represents close to a true 2D flow problem that can be used for validation purposes.

The geometry of the spillway profile is as per the Hydraulic Design Chart 111-2/1 [9]. It has a vertical upstream face and a curve, which is defined by three radii ($R=0.04H_d$, $R=0.20H_d$ and $R=0.50H_d$; H_d is the design head) in front of the centreline of crest. The profile downstream of the crest centreline is defined by the following equation,

$$(x/H_d)^{1.85} = 2H_d^{0.85}(y/H_d) \quad (1)$$

The origin of the x-y coordinates is located at the crest with +ve y pointing downward.

An overall view of the mesh and obstacle is shown in figure 1. Figure 2 shows a close-up view of the spillway crest. The mesh consists of 95 cells in the x (horizontal) direction and 98 cells in the z (vertical) direction. The aspect ratio was kept to unity wherever possible especially in the region of interest for solution accuracy and computational speed purposes. For this mesh, the maximum aspect ratios were 2.3 and 2.5 in the x and z directions respectively. Note that the z-direction was used in place of the y-direction as defined in equation (1).

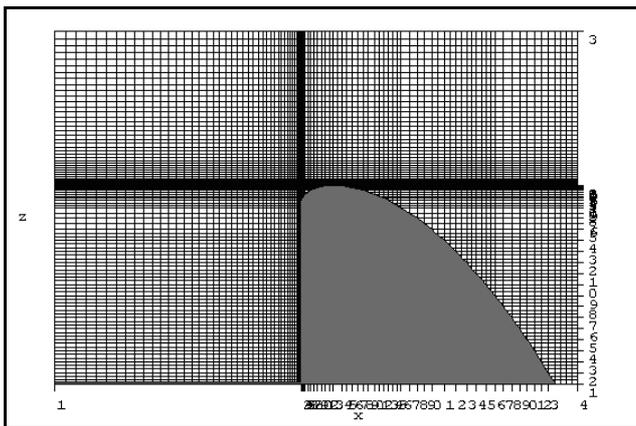


Figure 1. Overall view of the 2-D spillway mesh and obstacle.

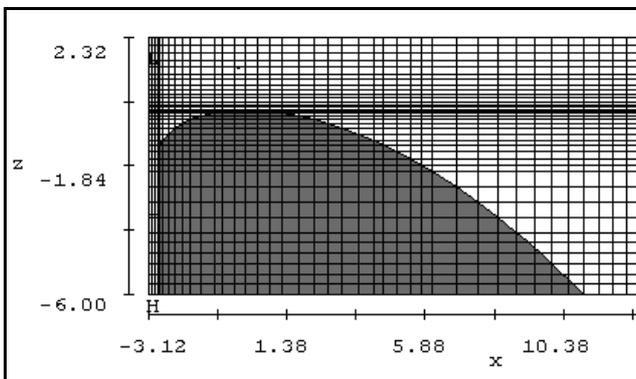


Figure 2. Close-up view of the spillway crest.

The design head was taken as 10m for the present investigation. The left boundary (upstream) was 25m away from the crest and the right boundary (downstream) was 22m from the crest. The bottom boundary was 18m below the crest and the top boundary was 14m above the crest. The following boundary conditions were assumed:

Upstream boundary:	Hydrostatic pressure with zero velocity; fluid height = H ;
Downstream boundary:	An outflow boundary;
Bottom upstream:	No flow – blocked by obstacle below;
Bottom downstream:	An outflow boundary;
Top boundary:	Symmetry – no influence in this case because of gravity.

The initial condition was set up such that a volume of fluid with a head of H was located at the crest of the spillway. The transient flow analysis was carried out for a total time period of 15s when a steady state was reached. This was determined by inspecting the results such as flow rate and kinetic energy of the system. A constant water density of 1000 kg/m^3 was used. This assumed the water is incompressible. A gravity value of 9.81 m/s^2 was applied in the negative z-direction. Three different heads ($H/H_d=1.33$, 1.00 and 0.50) were examined.

Crest Pressure Distribution

The sequence of flow over the spillway is shown from figures 3 to 5. These figures show the pressure contours (in Pa) and velocity vectors (in m/s) at different time steps for the $H/H_d=1.33$ case.

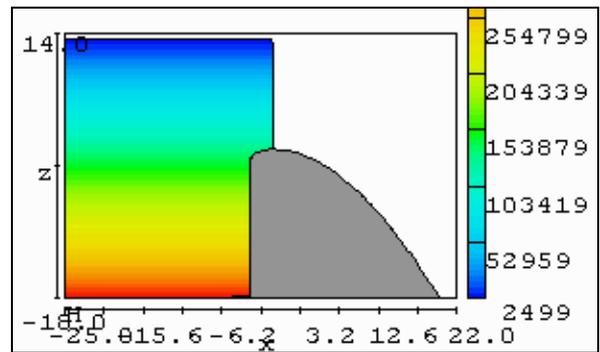


Figure 3. Pressure contours and velocity vectors at $t=0$ ($H/H_d=1.33$).

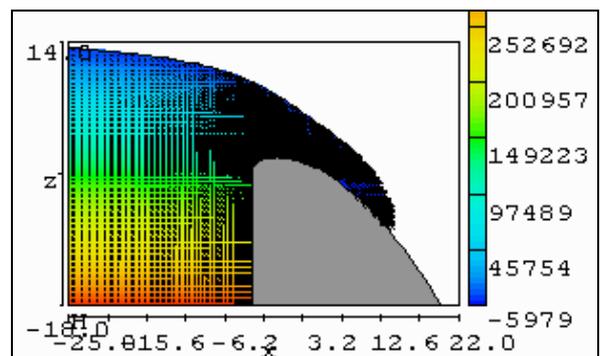


Figure 4. Pressure contours and velocity vectors at $t=1.5\text{s}$ ($H/H_d=1.33$).

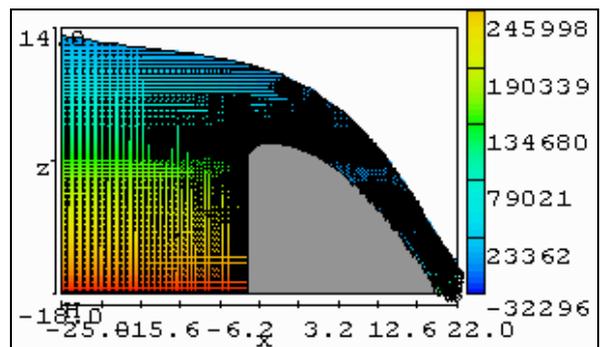


Figure 5. Pressure contours and velocity vectors at $t=15\text{s}$ ($H/H_d=1.33$).

The gauge pressure distributions along the crest at steady state are shown in figure 6. These pressures were taken from the cells located close to the obstacle (spillway). Also plotted in the figure are the measured data. It can be observed that the computed results gave a slightly higher negative pressure, but the general trend and magnitudes are in good agreement with the measured data. Some pressure oscillations can be seen and they are probably attributed to localised mesh effect.

For the design head case ($H/H_d=1.00$), the flow generated close to zero pressure along the spillway as expected; even though no aeration was introduced in the numerical simulation. When the head was higher than the design head, negative gauge pressure occurred. And when the head was lower than the design head, positive pressure was generated. Figure 7 shows the pressure contours at the crest of the spillway at steady state for $H/H_d=1.33$. A region of negative pressure above the crest can be observed.

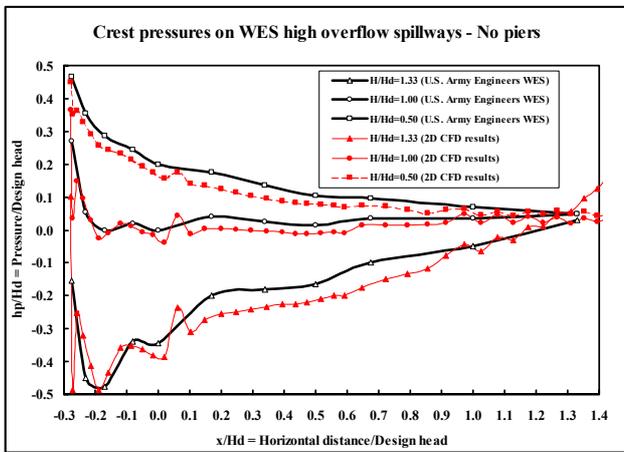


Figure 6. Crest pressure distribution for 3 different heads. Comparison with measured results – normalised with the design head.

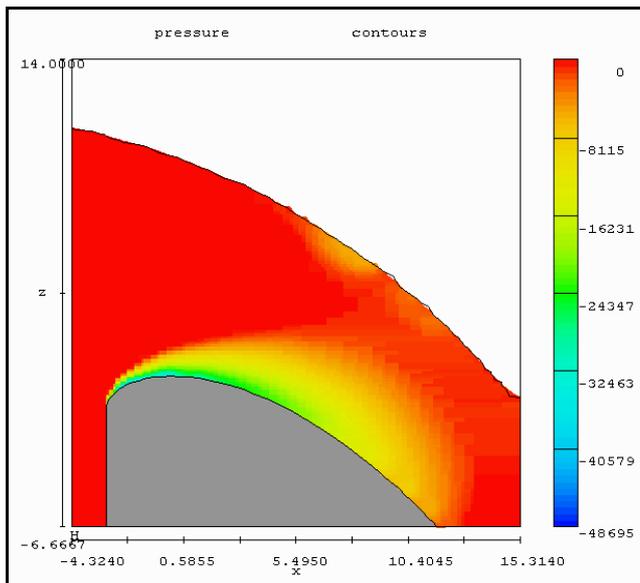


Figure 7. Negative gauge pressure distribution (Pa) above spillway crest. ($H/H_d=1.33$).

The computed upper nappe profiles and the measured data are shown in figure 8. It can be observed that the computed profiles are in excellent agreement with those measured.

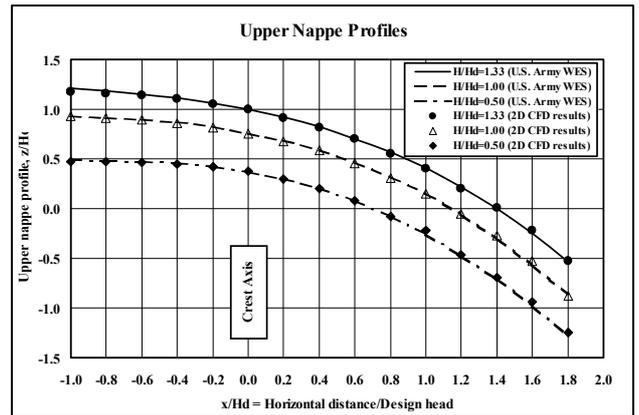


Figure 8. Upper nappe profiles. Comparison with measurement.

Velocity Distribution and Discharge

The velocity vectors over the crest at steady state for $H/H_d=1.33$ are shown in figure 9. The computation assumed a perfectly smooth wall and the flow was inviscid (no viscosity); therefore non-zero velocity at the boundary was computed. The effect of turbulent flow will be the subject of future study. Based on experimental results, the discharge over sharp-crested weir/spillway can be expressed as,

$$Q = CLH^{1.5} \quad (2)$$

where Q = discharge (ft^3/s)
 C = discharge coefficient
 L = effective length of weir crest (ft)
and H = measured head above the crest excluding the velocity head (ft).

The discharge coefficient according to Rehbock is approximately give as

$$C = 3.27 + 0.40 (H/h) \quad (3)$$

where h = height of weir.

The computed discharge and average velocity (velocity in the x direction at the centreline of the crest) for each case are shown in table 1. The discharge according to equation (2) and the corresponding average velocity are also shown in the table for comparison purposes. It can be seen that the computed values over-estimated the empirical results by about 10 to 20%. This could be related to the inviscid flow condition used in the analysis.

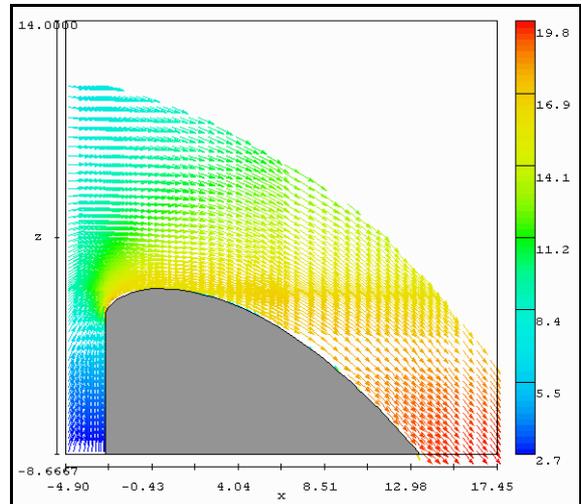


Figure 9. Velocity vectors (m/s) over spillway crest ($H/H_d=1.33$).

H/H _d	Empirical Results			Computed Results			
	Q/L	D	Vel.	Q/L	Diff. (%)	Vel.	Diff. (%)
1.33	95.6	10.0	9.6	112.3	17	10.9	14
1.00	61.0	7.5	8.1	70.7	16	9.4	16
0.50	20.9	3.8	5.5	22.9	10	6.1	12

Q/L (m³/s/m), D = depth over crest (m), Vel.= average velocity (m/s).

Table 1. Comparison of discharge and average velocity.

Three-Dimensional Spillway Model

A WES spillway model with piers is now considered. Three-dimensional effects will be caused by the presence of the piers. A rather coarse mesh was used for this analysis. The geometry is similar to the two-dimensional case except a pier and an abutment were included in the model. A symmetry condition was applied to both the z-x boundary planes. A constant hydrostatic head was maintained on the upstream z-y boundary plane. Water was allowed to flow out through the downstream z-y and x-y boundary planes. A view of the model at steady state is shown in figure 10.

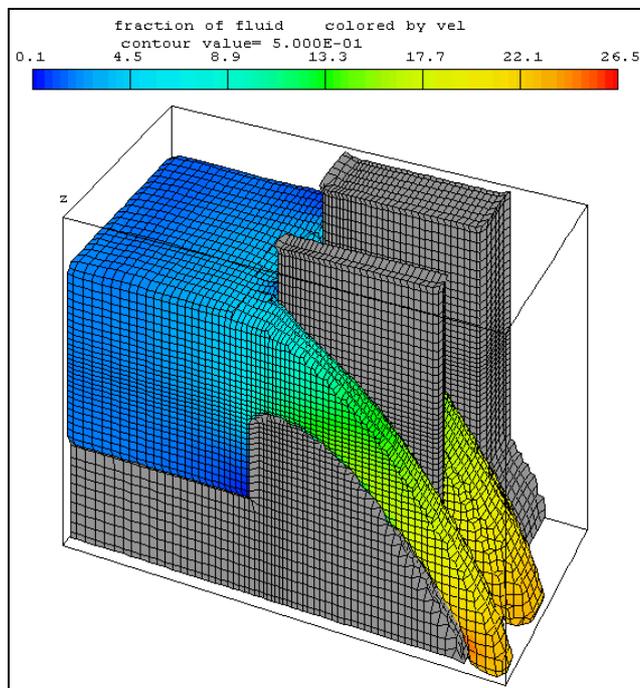


Figure 10. Three-dimensional model showing water fraction and velocity (m/s) flowing over the spillway at steady state – H/H_d=1.33.

Crest Pressure Distribution (Three-Dimensional Case)

Preliminary inspection of the results show there is relatively good agreement between the analysis results and the published data. Higher negative pressure occurred close to the pier than at the centre of the bay.

Further investigation would study the effect of mesh density, wall roughness and the use of turbulence model.

Application to a Spillway Structure

In view of the excellent agreement between the computed results and published data in the two-dimensional analysis, a concrete gravity dam central spillway in New South Wales, Australia, was analysed under increased flood levels using the same technique. It is not an Ogee spillway and therefore there are no published design charts that can provide an accurate behaviour of this hydraulic structure. A physical model had been constructed and the behaviour of the spillway under various flood levels had been

carried out. The computed crest pressure distributions along the centreline of the spillway are shown in figure 11. The measurements from the physical model tests are also plotted for comparison. It can be observed that relatively good agreement was obtained.

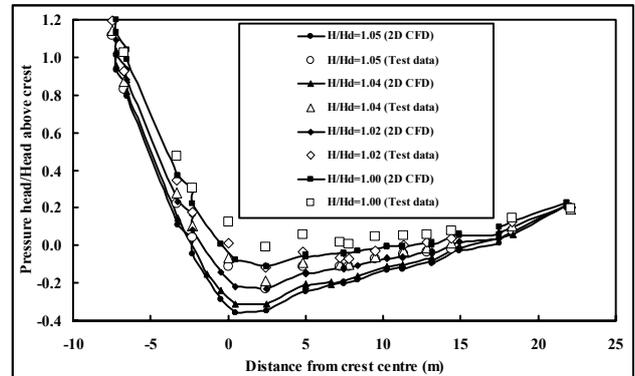


Figure 11. Normalised crest pressure distribution for various flood levels. Comparison of results between CFD analysis and physical model tests.

Conclusions

The use of CFD analysis in studying the behaviour of floodwater flowing over spillway in two- and three-dimensions has been demonstrated. The results were validated against those reported by US Army Corps of Engineers, Waterways Experiment Station. Preliminary analysis of a spillway structure shows the analysis captured the correct behaviour as measured by physical testings. In the present analysis, the computed results overestimated the velocity and hence underestimated the pressure distribution along the spillway. Further investigation will include mesh sensitivity, the influence of turbulence flow, non-uniform upstream flow and adjacent pier structures.

References

- [1] Chow, V.T. *Open-Channel Hydraulics*, McGraw Hill, 1959.
- [2] Flow Science, Inc. *FLOW-3D User's Manuals*, 2000.
- [3] Higgs, J.A., Folsom Dam Spillway Vortices Computational Fluid Dynamic Model Study, *Memorandum Report*, Water Resources Research Laboratory, Water Resources Services, Denver Technical Center, Bureau of Reclamation, United States Department of the Interior, Denver, Colorado, February, 1997.
- [4] Hirt, C.W., Nichols, B.D. and Romero, N.C., SOLA-A Numerical Solution Algorithm for Transient Fluid Flows, *Los Alamos Scientific Laboratory Report LA-5852*, 1975.
- [5] Hirt, C.W. and Nichols, B.D., Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, *J. Comp. Phys.* 39, No.201, 1981.
- [6] Hirt, C.W. and Sicilian, J.M., A Porosity Technique for the Definition of Obstacles in Rectangular Cell Meshes, *Proc. 4th Int. Conf. Ship Hydro.*, National Academy of Science, Washington, DC, Sept., 1985.
- [7] Kjellesvig, H.M., *Numerical Modelling of Flow over a Spillway, Hydroinformatics '96*, Balkema, Rotterdam, 1996.
- [8] Savage, B.M. and Johnson, M.C., Flow over Ogee Spillway: Physical and Numerical Model Case Study, *J. Hydraulic Engineering*, ASCE, 127(8), 2001
- [9] US Army Corps of Engineers, Waterways Experiment Station, *Corps of Engineers Hydraulic Design Criteria*, (1952 – revised in subsequent years).