43rd ANCOLD Conference
Hobart, Tasmania, 24-29 October 2003

NUMERICAL FLOW ANALYSIS FOR SPILLWAYS

David Ho[1], Karen Boyes[2], Shane Donohoo[3] and Brian Cooper[4]

[1, 2, 3] Advanced Analysis, Worley Pty Ltd, Sydney, Australia.
Email: David.Ho@worley.com.au phone: (02) 9468 3635
[4] NSW Department of Commerce (Dams and Civil)
Email: Brian.Cooper@dpws.nsw.gov.au phone: (02) 9372 7836

ABSTRACT

Many dam structures in Australia were designed and built in the 1950s and 60s with limited hydrological information. As a result existing spillway structures are under-sized for today's revised probable maximum floods (PMF). Potential problems such as the generation of excessive negative pressure over spillway crest under increased flood condition could be encountered. This may cause instability or cavitation damage to the spillway. The raised flow profile may also have adverse impacts on crest bridges and gate structures.

Historically, 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 use of high-performance computers 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 two- and three-dimensional CFD modelling of spillway behaviour under rising flood levels. The results have been validated against published data and good agreement was obtained. The technique has been applied to investigate several spillway structures in Australia.

1 INTRODUCTION

The behaviour of water discharging over spillways has been investigated extensively by the US Army Corps of Engineers Waterways Experiment Station (USACE-WES, 1952) since the early 1950s. A series of hydraulic design charts has been updated and documented. A manual by the USACE (1995) is available for engineers to design spillway profile for a given design flood condition. However, the design charts are only applicable for certain types of spillway profiles and pier configurations covering a limited range of flood levels. In the past, this limitation was overcome by building scaled physical models to study the flow behaviour.

Ideally in a spillway design when operating at its design head, the spillway surface should experience just the atmospheric pressure. When the reservoir level is below the design flood level, the pressure over the spillway will be above atmospheric. When the reservoir level is well above the design head, excessive sub-atmospheric (negative gauge) pressure will occur along the spillway crest that may potentially cause instability to the spillway structure and damage to the concrete face of the spillway due to cavitation. Other adverse effects could occur where the discharging water impacts on the raised gate structures, and on the underside of crest-bridge resulting in an orifice flow.

A majority of dams and spillways in Australia were designed and constructed in the early 1950s and 1960s to cope with the then estimated design floods. Additional hydrological data has been gathered and analysed since. In general it has been found that the revised PMFs for many catchments have increased. In order to select the most optimum upgrade design, many dam owners need to consider the most cost-effective way to analyse the behaviour of spillway flow under increased maximum flood. Up until now, the use of physical scaled model was the only investigation method. Now, the use of numerical methods such as computational fluid dynamic (CFD) analysis is attractive in terms of lower cost and substantially reduced preparation time, and results can be obtained throughout the flow domain rather than at selected monitoring locations.

The recent advances in computer software and hardware technology mean the use of CFD technique in analysing flow over spillways is now possible (Kjellesvig, 1996, Higgs, 1997, Savage and Johnson, 2001). Early difficulties involving moving mesh or grid to track the free water surface and to obtain a converged solution were reported. Nowadays, more efficient CFD codes can solve the Navier-Stokes equations in three-dimensions and free surface computation in a significantly improved manner. Defining complex geometry and meshing in three-dimensions has been simplified. Many CFD codes can import geometry data directly from other drafting and computer-aided engineering software.

As this type of spillway analysis technique was used for the first time in Australia, the need to carry out validation was essential. The fundamentals of verification and validation of CFD simulations have been addressed by the aerospace industry (American Institute of Aeronautics and Astronautics, 1998). The inter-relationship between the real world, mathematical model and computer model has been thoroughly researched and established. Many recommendations in the aerospace guidelines are also applicable for the CFD modelling of hydraulic structures. It should be realised that even a physical scaled hydraulic model is only a mathematical representation of the real structure.

This paper begins by describing the general background of CFD modelling, and in particular, the tracking of free surface flows for spillway simulation. Then the process of conducting an extensive validation exercise by analysing an Ogee spillway profile under various flood levels in both two- and three-dimensions will be described. The computed results were compared with published data in order to provide a level of confidence in applying CFD modelling in future studies. Finally, a number of case studies will be shown to demonstrate the applicability of this analysis technique. In each case study, validation using available information, where possible including earlier physical hydraulic model tests, was carried out to ensure the simulation result was correct and sensible. Some interesting flow behaviours discovered in the computation are also highlighted.

2 NUMERICAL FLOW ANALYSIS

Fluid flow models that are solved numerically can be categorized by the appropriate governing partial differential equations as shown in Table 1. For the majority of hydraulic applications involving water flow, the numerical model will be set up for solving a steady-state incompressible flow in the form of the Navier-Stokes equations. The velocities and pressures in the domain will be solved in the computation for each time step in a transient analysis. The partial differential equation can be difficult to solve because of the inherent nonlinearity, second order form, four-dimensional, and inter-related variables.

Flow model	Governing equation
Ideal, irrotational and incompressible	Euler's momentum equation, and conservation of mass
Viscous, incompressible without inertia effects	Stoke's flow or creeping
Viscous, incompressible with inertia effects	Navier-Stokes equation
Viscous, compressible with inertia effects	Navier-Stokes equation with compressibility terms

Table 1 Different flow models and their governing equations.

The computation becomes more complex when a turbulence model is introduced in the equation. It should be noted that while the Navier-Stokes equations can be solved exactly for many classical problems, any attempt to model the effects of turbulence could only be done in a statistical approximation manner. The Reynolds-averaged Navier-Stokes equations are commonly used. This is an expanded form of the Navier-Stokes equations that carries the Reynolds stresses terms. Most turbulence models such as mixing-length, k-ε (turbulent kinetic energy dissipation equations), are ways for calculating the Reynolds stresses. An alternative method to Reynolds-averaging in dealing with turbulence is Large Eddy Simulation (LES). The LES equations treat turbulence in terms of space-average at a fixed point in time. Although this technique can capture large-scale eddies accurately (sometimes exactly) in time, small scale eddies will be difficult to represent in the model unless a very fine grid or a sub-grid scale model (Demny et al, 2000) is used. A reasonable LES would require very fast CPUs (Central Processing Units) and enormous data storage, and therefore are still restricted to research institutions.

The treatment of boundary layer and wall roughness adds another complexity to the computation. The boundary layer concept assumes that at high Reynolds numbers the effects of viscosity are confined to the thin layers adjacent to any solid obstacles or boundaries. In general a flow region can be divided into two zones; a boundary layer region where the effects of viscosity are concentrated, and a main flow region where they are negligible (or inviscid). The viscous-inviscid iteration is a method of coupling the two regions of flow. Sometimes calibration may be necessary to determine the appropriate value of "wall function" or "wall roughness" to match the global empirical Manning or Chezy coefficients.

The details and different numerical methods of solving the above-mentioned equations can be found in standard CFD textbooks (Versteeg and Malalasekera, 1995, Abbott and Basco, 1989 and Wilcox, 1993). Typically, the CFD modelling process can be carried out in the following steps:

1. Select the appropriate governing equations – some idea of the Reynolds number would provide a good starting point.
2. Create the flow domain and obstacles that represent no flow regions such as spillway and piers. An appropriate grid or mesh will then be generated.
3. Define the boundary and initial conditions – for example, upstream head and tail-water head.
4. Select the fluid properties, wall roughness and turbulence model.
5. The program will solve for the unknown variables such as pressure and velocity with time at the grid points or at the cell/mesh center.
6. Post-process results to extract the desired information for design.

Besides solving for pressure and velocity in the flow domain, one important aspect in modelling open-channel flow is the accurate tracking of the free surface. Sometimes, multiple free surfaces would be involved in the model, for example, a stream of water entering a plunge pool after leaving a flip-bucket. A well-known computational technique was developed by Hirt (1994), and was implemented in the CFD code, FLOW-3D, which was used for the spillway analyses described in this paper. The program solves the Navier-Stokes equation by the finite difference method. It utilizes a true volume of fluid (true VOF) method for computing free surface motion (Hirt and Nichols, 1981) and the fractional area/volume obstacle representation (FAVOR) technique to model complex geometric regions (Hirt and Sicilian, 1985). The true VOF method tracks the sharp interface accurately and does not compute the dynamics in the void or air regions. The portion of volume or area 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, momentum 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. Furthermore, multi-block meshing technique can be employed to provide higher resolution of simulation in the region of interest if necessary.

The basic algorithm for advancing a solution in one time increment consists of the following three steps (Flow Science, 2000):

1. Compute the velocities in each cell 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.
2. Adjust the pressure in each cell to satisfy the continuity equation.
3. Update the fluid free surface or interface to give the new fluid configuration based on the volume of fluid value in each cell.

A mixture of explicit and implicit solution schemes can be used to solve for the partial differential equations. The selection depends on the complexity of the fluid flow problem in question. Detailed information is described elsewhere (Versteeg and Malalasekera, 1995).

3 VALIDATIONS USING THE OGEE SPILLWAY PROFILE

As part of the Warragamba Dam upgrade project, the use of the CFD analysis to study the flow impact on the control gates at the crest was subjected to extensive validation to ensure the analysis technique was correct. A flowchart highlighting the validation process in a multi-stage approach in the project is shown in Figure 1. The validations of the standard Ogee spillway profile were carried out both two- and three-dimensionally.

It should be noted that a similar validation exercise (Savage and Johnson, 2001) was conducted using the same CFD code, which provides further confidence in the analysis technique.

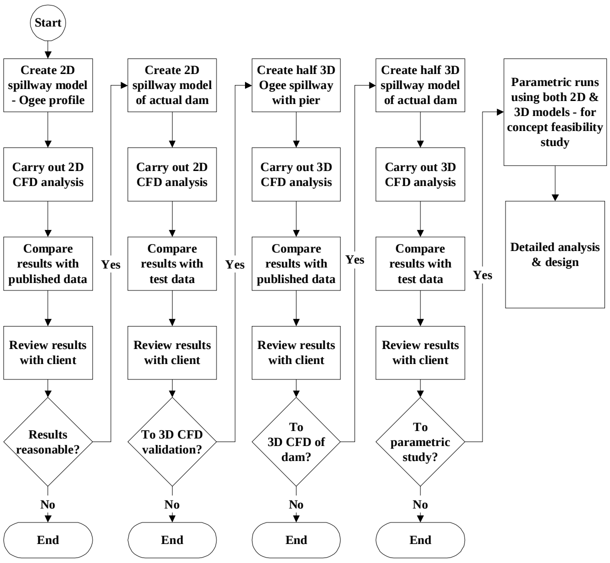

Figure 1 Flowchart showing the validation process.

3.1 Two-dimensional Ogee profile

An Ogee spillway model without piers was considered because the measured results were not influenced by any three-dimensional effects as the edges were far from the region of interest. This model represents close to a true two-dimensional flow problem that can be used for validation purposes.

The geometry of the spillway profile was taken from the USACE Hydraulic Design Chart 111-2/1 (USACE-WES, 1952). It has a vertical upstream face and a curved profile 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 crest centerline. The profile downstream of the crest centerline is defined by the following equation:

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

The origin of the x-y coordinates is located at the crest with +ve y pointing downward. An overall view of the grid and spillway obstacle is shown in Figure 2. The grid consists of 95 cells in the horizontal direction and 98 cells in the vertical direction. The aspect ratio was kept to unity where possible especially in the region of interest for solution accuracy and efficient run-time purposes.

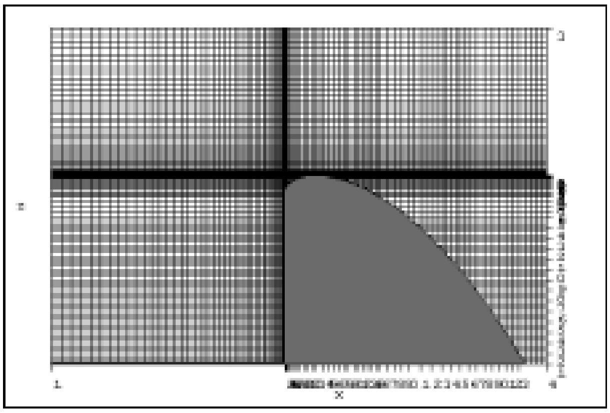

Figure 2 An overall view of the 2-D spillway mesh and obstacle.

The design head was taken as 10m in this validation study. The upstream boundary was 25m away from the crest and the downstream boundary was 22m away from the crest. The bottom boundary was 18m below the crest and the top boundary was 14m above the crest. The following conditions were assigned at the boundaries:

- Upstream boundary: Hydrostatic pressure with zero velocity; fluid height = H = H_d
- Downstream boundary: An outflow boundary
- Bottom upstream: No flow is allowed as an obstacle below blocks this boundary
- Bottom downstream: An outflow boundary

- Top boundary: Symmetry – no influence in this case due to open-channel flow

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 period of time until a steady state was reached. This was determined by inspecting the results such as the net flow rate and the kinetic energy of the system. A constant water density of $1000 kg/m^3$ and dynamic viscosity (at 20^oC) of 0.001kg/m/s was used. This assumed the water is incompressible. A gravity value of $9.81 m/s^2$ was applied in the negative vertical direction. Three different upstream heads (H/H_d=1.33, 1.00 and 0.50) were examined.

The crest pressure distributions for the three heads are shown in Figure 3. These pressures were taken from the cells located close to the spillway surface. Also plotted in the figure are the published data by the USACE. It can be observed that the computed results give 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 localized mesh effect.

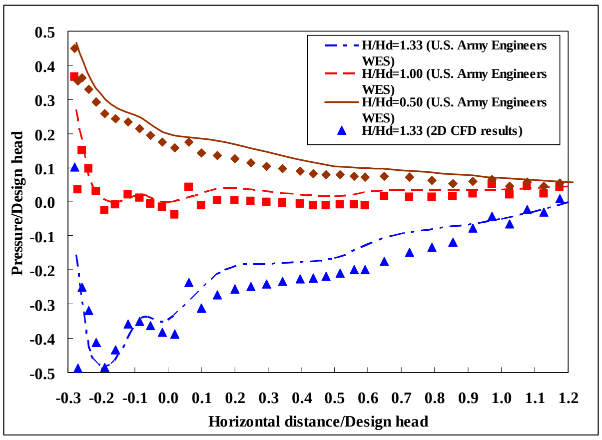

Figure 3 Comparison of crest pressure distributions for three different heads.

For the design head case (i.e. H/H_d = 1.00), the flow generated a small amount of pressure along the spillway as expected even though no aeration was introduced in the simulation. When the head was higher than the design head, negative pressure was generated at the crest as shown in Figure 4.

The computed free surface profiles and the measured data are shown in Figure 5. It can be seen that they are in excellent agreement with each other.

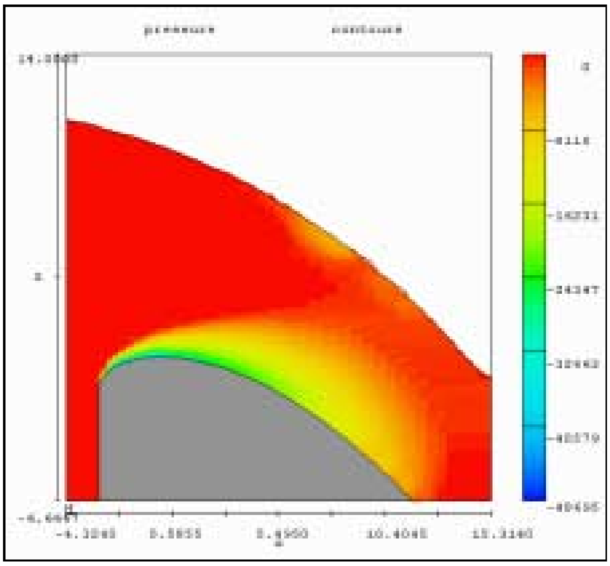

Figure 4 Negative gauge pressure (Pa) contour above spillway crest (H/H_d=1.33).

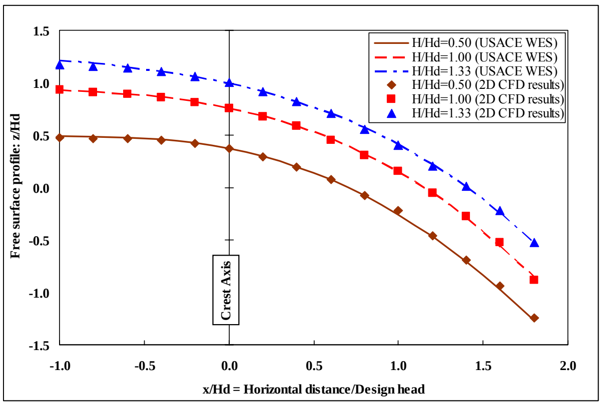

Figure 5 Comparison of upper nappe profiles for the 3 heads.

The empirical relationship between the flow rate and upstream head for a sharp-crested weir/spillway is given by:

$$Q = C\,L\,H^{1.5} \qquad (2)$$

where Q = discharge
C = discharge coefficient
L = effective length of weir crest
and H = measured head above the crest excluding the velocity head.

The discharge coefficient according to Rehbock is approximately given by Chow (1959):

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

where h = height of weir (ft).

Note that equation (2) is units dependent and equation (3) is in imperial units.

The computed discharge and average horizontal velocity at the spillway crest for each case are shown in Table 2. The discharge based on equation (3) and the corresponding average velocity for each head is also shown in the table for comparison purposes. The computed values over-estimated the empirical predictions by about 10 to 20%. This is probably due to laminar and invisicid flow assumption used in the computation. In addition the spillway wall was assumed to be perfectly smooth. Therefore any energy loss at the spillway boundary has not been accounted for. The effect of turbulence flow and boundary roughness will be the subject of future validation study.

H/H_d	Empirical predictions			CFD predictions			
	Q/L	D	V	Q/L	Diff (%)	V	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 ($m^3/s/m$)
D = water depth at crest (m)
V = average horizontal velocity at crest (m/s)

Table 2 Comparison of discharge and average velocity (2D ogee spillway).

3.2 Three-dimensional Ogee profile

The geometry of the spillway is the same the two-dimension model except a Type 2 pier (Chow, 1959) and abutment were included in the obstacle model. By exploiting symmetry condition, a half three-dimensional model was analysed (see Figure 6). A design head, H_d, of 10m was also used as before. The pier dimensions are: width = 0.2 H_d; upstream radius = 0.1 H_d and bay width = 1.078 H_d. The same water properties, boundary and initial conditions were used as in the two-dimensional analysis. The steady state results were extracted for comparison purposes.

The crest pressure distributions along the pier for different heads are shown in Figure 7. The computed results compare reasonably well with the trend and magnitude. In general the predicted pressures are more negative than the published data.

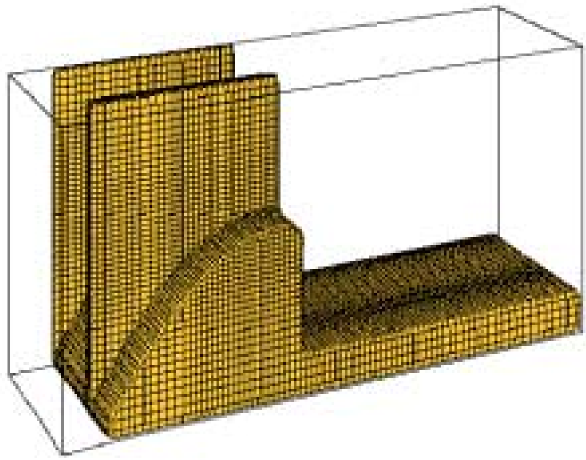

Figure 6 A view of the ogee spillway and Type 2 piers in the 3D CFD model.

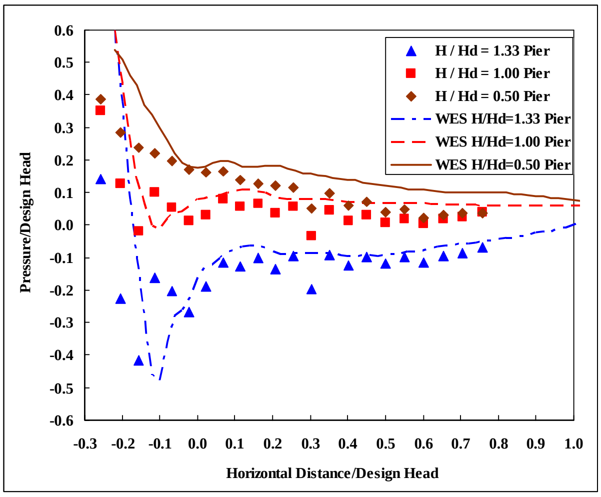

Figure 7 Comparison of crest pressure next to pier (3D model).

The free surface profiles adjacent to the pier are shown in Figure 8. The computed profiles compare reasonably well with the measured data. The influence of the piers is also captured by the computation.

The empirical discharge based on USACE (1995) is given by:

$$q = Q/L_e \qquad (4)$$

where $Q = C\ L_e\ H_e^{1/2}$ (5)
C = discharge coefficient
L_e = effective length of weir crest
and H_e = total specific energy above the crest.

The effective length takes into account of the abutments and piers and it is given by:

$$L_e = L - 2(nK_p + K_a)H_e \qquad (6)$$

where L = net length of crest
n = number of piers
K_p = pier contraction coefficient
and K_a = abutment contraction coefficient.

The discharge for each head is summarized in Table 3. The computed discharges for different heads are in good agreement with those predicted by the empirical method.

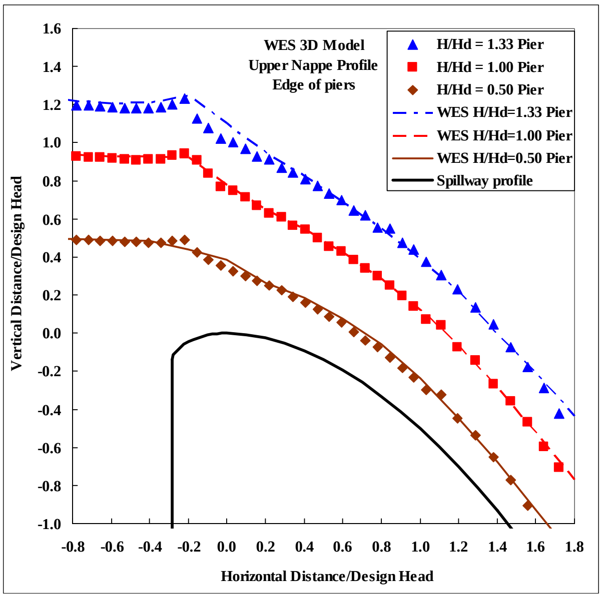

Figure 8 Upper nappe profiles next to pier.

H/H_d	Empirical predictions	CFD predictions	Difference (%)
1.33	115.1	113.2	1.7
1.00	70.2	70.1	0.1
0.50	22.6	21.7	3.9

Table 3 Comparison of discharge $m^3/s/m$ run (3D ogee spillway).

4 APPLICATIONS

As with any other projects that involved extensive numerical computation, a considerable effort was spent on validating the CFD model for each of the following case studies - despite the fact that the validation performed for the Ogee spillway was quite satisfactory and encouraging. Detailed description of the modelling process for each case will not be covered in the paper due to space constraint. The basic modelling approach is similar to the one used for the Ogee spillway validation work. Only special features will be described. In all cases, the revised PMFs have increased quite considerably. The Buffalo Dam Spillway has a standard Ogee profile. Neither the Warragamba Dam Existing Spillway nor the Hume Dam Spillway has a standard Ogee profile.

4.1 Warragamba Dam Existing Spillway, New South Wales

As part of the Warragamba Dam upgrade program, interim raising of the dam crest and construction of an auxiliary spillway have been completed. However, the impact of the higher flood level on the existing gates at the crest of the existing spillway is still need to be assessed. Any excessive negative pressure generated over the spillway crest could potentially create a large enough suction to lift the drum gate out of its chamber. Also, the raised free water surface is likely to impinge on the radial gates' arms and trunnions just beyond the spillway crest.

The analysis process (Worley, 2001 & 2003a), which incorporated several validation stages, is outlined in Figure 1. Both two- and half three-dimensional models were prepared in the investigation. The pressure distributions along the spillway crest for a number of flood levels were compared with those obtained from physical scale model (1:100) tests. A typical comparison is shown in Figure 9. The computed discharge was also checked against the design head-discharge chart. It was discovered that the piers do not have a strong three-dimensional influence on the crest pressure over the drum gate spillway. This is probably due to the small depth of water to crest length aspect ratio. Also, the use of various turbulence models does not show any significant difference in crest pressure. However, the discharge is reduced slightly when compared to that using a laminar inviscid flow model.

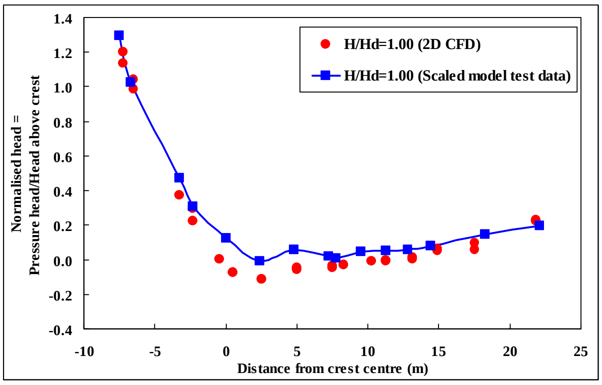

Figure 9 Comparison of crest pressure distributions.

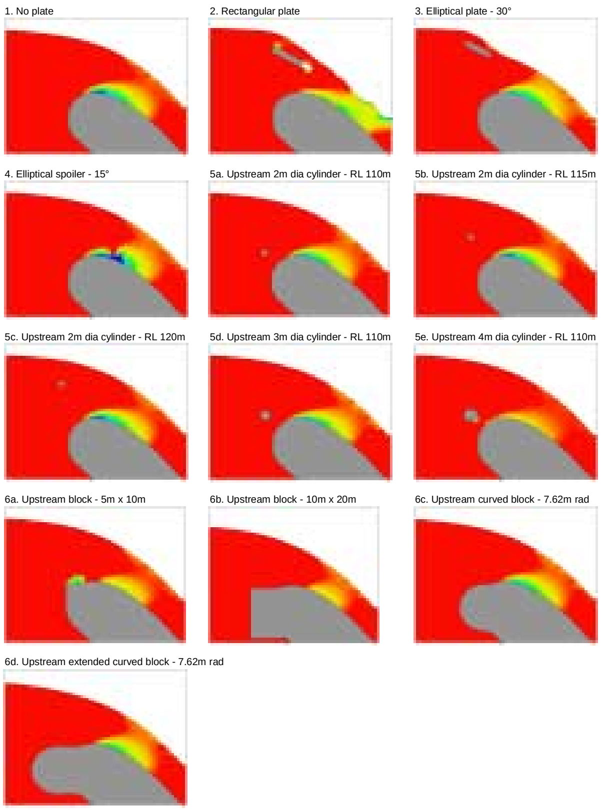

Figure 10 Influence of various flow splitters, baffles and blocks on crest pressure. Positive pressures are shown in red.

The free surface profile predicted for the radial gate spillway under the revised PMF was found to impact on the radial gate arms. This was further investigated in a simplified two-dimensional CFD analysis to determine the likely drag and lift forces on the partially submerged arms.

The CFD analysis was able to quantify the loads on the gate structure. In addition, it allowed the designers to study various concepts to improve or reduce the pressure distribution on the crest by reshaping the drum gate/spillway profile or installing different forms of obstacle just upstream of the crest as shown in Figure 10 (Worley, 2003c). It should be noted that none of these concepts were practicable from the engineering point of view.

4.2 Hume Dam Spillway, New South Wales

The rating curve for the Hume Dam Spillway was re-assessed to take into consideration of the effect of the overhead bridge when the flood level is high enough to cause over-topping (Worley, 2003b). The geometry details of the spillway crest, piers, the overhead bridge and the parapet wall were presented in the model using the multi-block meshing technique. A close-up side view of the crest top and a view of water discharging over the spillway in three-dimension are shown in Figure 11. A number of upstream heads were analysed.

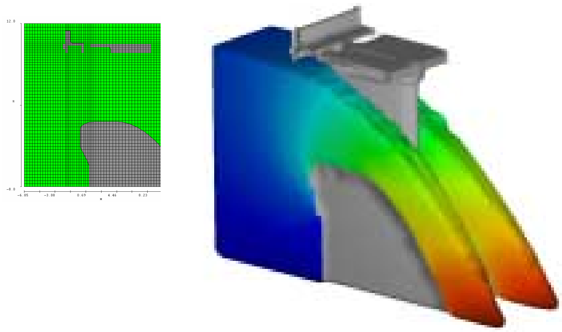

Figure 11 Geometric details at the crest. Side view of mesh (left). Water velocity magnitude contour plot (right).

Pressure measurements to study spillway cavitation were carried out on a 1:50 physical model as reported by Gourlay (1962). These results were used to validate the computational results as shown in Figures 12 and 13. The computed pressure distribution along the centerline of a spillway bay compares well with the measured data. Along the side of the pier, the computed results under-predicted the pressure but the trend was captured. It can be seen that the pressure immediately behind the pier does not agree with the measurement because water was not present in this region in the model. It was reported (Gourlay, 1962) that there was a large pressure fluctuation at this location with the maximum positive pressure head of 4.57m and a minimum of just below atmospheric.

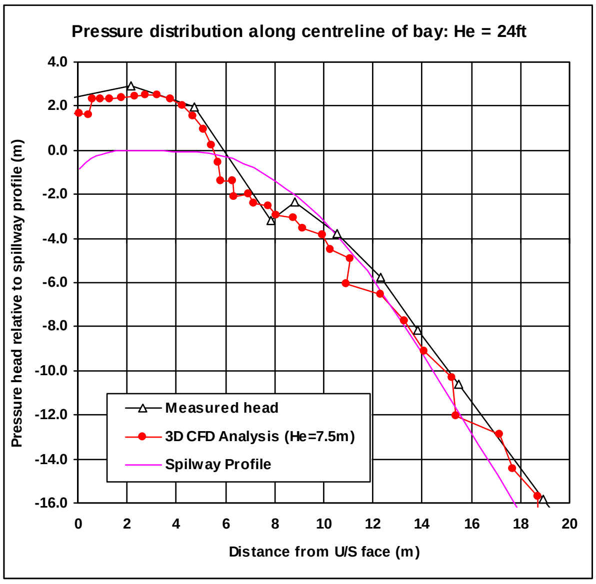

Figure 12 Comparison of pressure distribution along the centerline of bay.

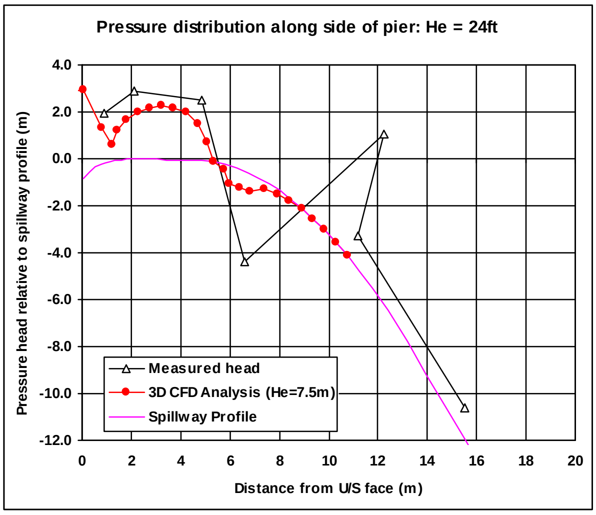

Figure 13 Comparison of pressure distribution along the pier.

One of the objectives in the investigation was to determine the flow rate when the water level was high enough to catch the underside of the bridge to create an orifice flow through the spillway bay. The head-discharge relationship is shown in Figure 14. It can be observed that there is good agreement between the CFD results and the test data for the spillway flow. There is only a slight reduction in discharge for the orifice flow and therefore the curve does not diverge sharply from the $q \approx C_d H^{1.5}$ curve. By considering the "theoretical" discharge using simplified empirical weir flow and orifice flow equations and assuming 100% efficiency (i.e. $C_d = 1.0$), it was shown that the transition was indeed not abrupt. It should be noted that the discharge coefficient could have a range of 0.5 to 0.98 depending on the types of orifice and "mouthpiece" (Featherstone and Nalluri, 1982). This can be back-figured from the CFD analysis provided the details of the orifice geometry have been sufficiently modelled.

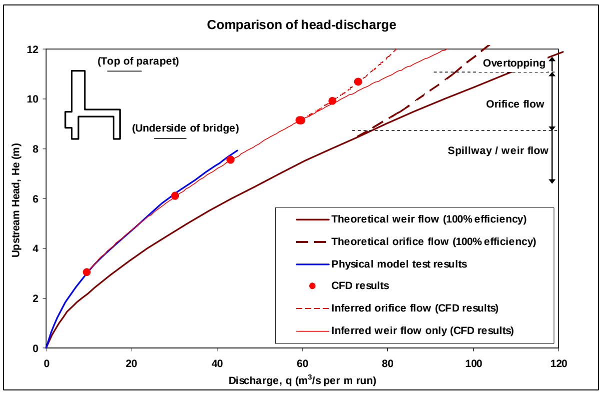

Figure 14 Comparison of head-discharge relationship.

An additional benefit from the CFD modelling is the ability to take snapshots of the water flow and free surface formations. In this investigation, the presence of the "bow wave" at the upstream end of the pier was captured. Research done by USACE (1995) indicated that certain pier configurations can actually increase the discharge (i.e. a negative contraction coefficient). In order to obtain a much finer flow detail, the mesh size around the piers would have to be very fine with a corresponding increase in computation time.

4.3 Buffalo Dam Main Spillway, Victoria

For the Buffalo Dam Main Spillway, it was found that the revised flood level was several times the original head that it was designed for. The upgrade design needed to consider the overall stability of the spillway, the possibility of relocating the crest bridge and the revised discharge. Although it is an Ogee spillway profile, the flood levels are beyond those considered in the design manual. Therefore a series of CFD analysis of the half three-dimensional model of the spillway was carried out (Worley, 2002).

Several two-dimensional calibration analyses were performed to optimise the mesh density and the distance to the upstream and downstream boundaries. Once it was optimised and the results were validated against published data of the design head case, the model was extended to three-dimensions. The final model consists of half of the central bay and one full outer bay with one pier (Type 2 round-nose) in between. The training wall, upstream approach to the spillway and the vertical gate slots in the pier were included in the model. The existing crest bridge was not modelled. It was assumed the pier extended well beyond the revised water surface. A symmetry boundary condition was assigned to the centreline of the central bay. A viscous turbulence flow (k-ε) model was used. A smooth wall boundary layer, which corresponds to a good quality concrete surface finish, was assumed.

The discharges for various heads were analysed and they gave an excellent agreement with those published by USACE as shown in Table 4.

H/H_d	USACE predictions	CFD predictions	Difference (%)
1.00	13.5	13.9	3.0
1.33	22.0	22.1	0.5

Table 4 Buffalo Dam Main Spillway - Comparison of discharge $m^3/s/m$ run.

The analysis provided pressure distribution around the spillway so that the overall stability of the structure could be assessed. The analysis also revealed that there was potential for cavitation damage just upstream of the spillway crest for the highest flood level. Although a cavitation model is available in the CFD code, it was not used in this case because spillway instability would be the dominant "failure" mode at this flood level.

By inspecting the free surface profile for the high flood level (Figure 15), it could also be determined whether the crest bridge would be impacted upon.

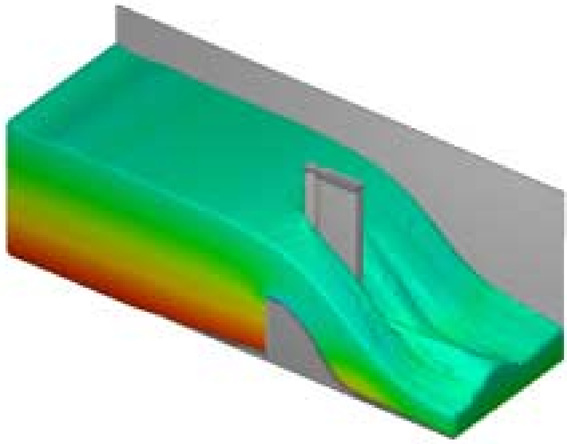

Figure 15 Free water surface at steady state spillway discharge.

5 CONCLUSIONS

The basic background of CFD modelling and its application to analyse spillway flow have been discussed. The methodology has been validated against published data by analysing the standard Ogee spillway in two- and three-dimensions. The benefits of using this new but viable technology for analysing flow in hydraulic structures in Australia have been demonstrated in three real world applications. Currently, extensive CFD modelling is carried out on the existing spillway at Wivenhoe Dam, Queensland.

This technological development has matured to the point that it can be used by hydraulic engineers for practical design purposes. As with physical hydraulic model testing, it is essential that in the development of a CFD model, there is a thorough understanding of the behaviour of the spillway when estimating the input parameters and interpreting the output results.

6 ACKNOWLEDGEMENT

The authors wish to thank the NSW Department of Public Works and Services (now the NSW Department of Commerce), Sydney Catchment Authority, Murray Darling Basin Commission and Goulburn-Murray Water for permission to publish some of the analysis results in this paper.

7 REFERENCES

Abbott, M.B. and Basco, D.R. 1989. *Computational Fluid Dynamics: An Introduction for Engineers*, Wiley, p.425.

American Institute of Aeronautics and Astronautics. 1998. *Guide for the Verification and Validation of Computational Fluid Dynamics Simulations*. AIAA G-077-1998, p.19.

Chow, V.T. 1959. *Open-Channel Hydraulics*, McGraw Hill.

Demny, G., Rettemeier, K., Forkel, C. and Kongeter, J. 2000. 'A Numerical Model for the 3D-Simulation of Flow Through the Intake of Water Power Stations', *Proceedings 28th IAHR Biennial Congress*, Graz, Austria, 22-27 August.

Featherstone, R.E. and Nalluri, C. 1982. *Civil Engineering Hydraulics*, Granada. p.371.

Flow Science, Inc. 2000. *FLOW-3D User's Manual*.

Gourlay, M.R. 1962. 'Some Hydraulic Investigations for the Raising of Hume Reservoir', *Civil Engineering Transactions*, The Institution of Engineers, Australia. March, 61-70.

Higgs, J.A. 1997. *Folsom Dam Spillway Vortices Computational Fluid Dynamics Model Study*, Memorandum Report, Water Resources Research Laboratory, Water Resources Services, Denver Technical Center, Bureau of Reclamation, US Department of the Interior, Denver, Colorado, February.

Hirt, C.W. 1994. 'Weir Discharge and Counter Currents', *Proceedings 1st International Conference on Hydroinformatics*, Delft, 19-23 September.

Hirt, C.W. and Nichols, B.D. 1981. 'Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries', *J. Comp. Phys.* 39, No.201.

Hirt, C.W. and Sicilian, J.M. 1985. 'A Porosity Technique for the Definition of Obstacles in Rectangular Cell Meshes', *Proc. 4th Int. Conf. Ship Hydro.*, National Academy of Science, Washington, DC, September.

Kjellesvig, H.M. 1996. 'Numerical Modelling of Flow over a Spillway', *Hydroinformatics'96*, Balkema, Rotterdam. 697-702.

Savage, B.M. and Johnson, M.C. 2001. 'Flow over Ogee Spillway: Physical and Numerical Model Case Study', *J. Hydraulic Engineering*, ASCE, 127(8), 640-649.

US Army Corps of Engineers Waterways Experiment Station. 1952 – revised in subsequent years. *Corps of Engineers Hydraulic Design Criteria*.

US Army Corps of Engineers. 1995 *Hydraulic Design of Spillway, Technical Engineering and Design Guides as adapted form the US Army Corps of Engineers*, No. 12 ASCE.

Versteeg, H.K. and Malalasekera, W. 1995. *An Introduction to Computational Fluid Dynamics The Finite Volume Method*, Prentice Hall, p.257.

Wilcox, D.C. 1993. *Turbulence Modeling for CFD*, DCW Industries, Inc. p.460.

Worley. 2001. *Warragamba Dam Drum Gate – CFD Analysis Stage 1*. Report No. 087-17063-RP001. April. (Not published).

Worley, 2002. *Buffalo Dam CFD Analysis – 3D Model of Main Spillway*. Report No. 251-03926-RP001. December. (Not published).

Worley, 2003a. *Warragamba Dam Drum Gate – CFD Analysis Stage 2: Validation of 3D Model*. Report No. 087—17063-RP002. January. (Not published).

Worley, 2003b. *Hume Dam Spillway – CFD Analysis*. Report No. 251-00436-RP001-4. March. (Not published).

Worley, 2003c. *Warragamba Dam Drum Gate – Stage 3: Solution Concepts Study*. Report No. 087-17063-RP003. April. (Not published).