Application of computational fluid dynamics to evaluate hydraulic performance of spillways in Australia *

DKH Ho † and KM Riddette
WorleyParsons Services Pty Ltd, North Sydney, NSW

SUMMARY: *In recent years, the design flood estimates of a number of dams in Australia have been revised, requiring their spillways to be upgraded to cope with increased flood discharges. Traditionally, reduced scale physical models were used in hydraulic laboratories to study spillway hydraulic performance. However, these are prone to problems associated with scaling effects, and cannot readily capture behaviours such as cavitation and air-entrainment effects, which can occur in reality. Nowadays, with the advancement in computing technology, the hydraulic performance of spillways can be investigated numerically. Since 2001, a number of spillway upgrade projects in Australia have been performed using computational fluid dynamics. This paper provides an overview of how this technology was applied to these projects. The validation process to ensure the numerical model is reliable, and various analysis capabilities allowing better understanding of the flow behaviour will be described. Current limitations are also highlighted in order that future research and development can resolve them, thus making this numerical modelling technique more robust for flow simulation of not just spillways, but also other hydraulic structures in the future.*

1 INTRODUCTION

In Australia, many dams were designed and built in the past with limited hydrological information. As a result, existing spillways are under-sized to cope with revised probable maximum floods (PMFs), which have been re-evaluated using reliable longer-term hydrological data and improved analysis methods (Green & Meighen, 2006). The increased discharge may potentially cause dam safety problems, such as the generation of excess negative pressure over the spillway crest and along the chute, water impacting on crest structures, reduction of discharge efficiency to mitigate flood, erosion of unlined rock channels and banks, and overtopping of chute walls. The discharge coefficient and head-discharge curve will also need to be re-evaluated for increased flood levels. In the US, the National Performance of Dams Program, run by the Civil and Environmental Engineering Department at Stanford University (2008), has been monitoring dam safety incidents and maintaining a database of records dating back to 1848. It is interesting to note that out of 2570 incidents (taken from the 2007 figures), 196 cases were caused by inadequate spillways and 20 dams failed as a result.

Traditionally, reduced scale physical spillway models have been built in laboratories to study their hydraulic behaviour. However, these models can be expensive and time-consuming to construct and test, and there are some difficulties associated with scaling effects. There are difficulties in representing both the terrain and concrete roughness, and the measuring devices themselves may interfere with the flow behaviour. In turbulent flows, it is difficult to visualise streamlines. Furthermore, results can only be sampled in limited locations. A highly specialised facility is required to study the effect of cavitation, and air-entrainment cannot be examined reliably. As with many spillways built a long time ago, their physical models may have been decommissioned already and new ones have to be built for any upgrade studies.

Today, with the use of high-performance computing, for example, utilising multiple processors in a cluster arrangement, and more efficient computational fluid dynamics (CFD) codes, for example, parallelised algorithm, it is feasible to study the hydraulic behaviour of full-scale spillways. This technology has been well established in the aerospace, automotive and maritime industries worldwide for some time. The use of CFD technology for spillway application

* Paper C09-647 submitted 27/07/09; accepted for publication after review and revision 4/03/10.

† Corresponding author Dr David Ho can be contacted at david.ho@worleyparsons.com.

is quite recent, particularly in Australia. A review will be described in the next section. Throughout this paper, the term numerical or computational modelling or analysis means CFD modelling or analysis, and physical models imply reduced-scale physical models.

The first commercial application of CFD technology to spillway flow in Australia was performed in 2001 for the drum gate upgrade project at Warragamba Dam. A rigorous validation program was conducted in stages to ensure this modelling was reliable. Since then, 16 spillway upgrade projects were performed by the authors, and with the increase in computing power and more efficient use of multi-block meshing technique, much bigger three-dimensional (3D) models involving multi-bay spillways and even multiple spillways in a large reservoir were performed. For most of these projects, where possible, comparison between computed results and those obtained by physical models were carried out to raise the confidence level of the modelling. This validation process should correctly be termed as "pseudo-validation" as the physical models themselves are also an approximate representation of the actual spillways. A true validation can only be achieved by comparing computed results or results obtained from physical models with measurements taken from actual performance of the full-scale spillway under the same initial and boundary conditions.

In a few upgrade project studies, some interesting flow behaviours such as the transition flow regime from a free spillway discharge to an orifice flow through a gated spillway as the upstream water level rises was captured by the CFD model. Although simple two-dimensional (2D) models and half 3D models utilising symmetry can reproduce results published by the US Army Corps of Engineers (USACE, 1995), many spillway upgrade projects required the inclusion of the upstream approach terrain, abutments, multiple bays and crest structures in more complex 3D models in order to capture realistic flow behaviour. These full-scale 3D models can now be analysed more cost-effectively by using meshing techniques such as multi-block meshing, nested meshing and analysis restart from a coarse mesh to a finer mesh using Cartesian structured meshes.

Besides significant cost savings for these upgrade projects, the CFD modelling has led to the investigation and development of an innovative device to mitigate flow impact on radial gates, and concept development to reduce excessive negative pressure over a spillway crest. Other hydraulic structures that the authors have analysed using CFD include fishways, hydro-power penstock, pump intake stations and outfall structures. However, their details are beyond the scope of this paper.

During the course of these investigative studies on spillways, a number of interesting challenges were raised, including shock-wave formation, cavitation, water-structure interaction, air-entrainment, thermal stratification and erosion potential. Although some were overcome satisfactorily, others remain to be resolved. It is hoped that by highlighting the limitations as well as the features of the current CFD technology application to spillway hydraulics, the engineering community (in particular, dam engineers), will be aware of the capability and benefit that this technology can offer. The current limitations highlighted can be the focus for future research and development needs if this technology is to become a reliable and routine analysis tool.

2 CFD TECHNOLOGY IN HYDRAULIC ENGINEERING

2.1 International development and application

A literature search for the use of CFD technology in international applications has revealed, not surprisingly, that it began as in investigative tool at research institutions (Kjellesvig, 1996; Savage & Johnson, 2001), and gradually it has been accepted by the hydraulic/dam engineering community (Higgs, 1997; Yang & Johansson, 1998; Cederstrom et al, 2000; Teklemariam et al, 2002; Gessler, 2005). Since the mid 1990s, there has been an increasing amount of CFD modelling covering a diverse range of applications, such as piers and abutments in open channel flow and sewer overflow structures. For instance, a keyword search for "CFD" in the American Society of Civil Engineers (ASCE) database revealed that there has been a rapid increase in publications related to this technology. It appears this has a strong correlation with the increase in computing efficiency over the last decade.

The research and development and diverse applications of CFD will not be reviewed in this paper. There are numerous papers, for example those published by ASCE, that can be sourced by interested readers.

2.2 Australian development and application

In Australia, computational or numerical modelling of hydraulic performance has also been carried out at research level. For example, Brady (2003a; 2003b) investigated free surface flow for sewer overflow; Barton (2003) studied numerical modelling of vertical slotted fishway; Edwards (2006) compared two different CFD codes and the appropriateness of their use to model elliptical and ogee crest spillways; and more recently Kenny (2007) carried out an extensive parametric/sensitivity study to improve the confidence in CFD modelling of spillways.

In terms of industrial applications, this type of numerical modelling was not mentioned in the Australian National Committee on Large Dams publication on the history of dam technology from 1850 to 1999 (Cole, 2000). There has been no published information on CFD analysis of spillways in Australia until quite recently (Ho et al, 2003; 2004; 2005; Riddette et al, 2006; 2008; Phillips & Riddette, 2007; Hurst et al, 2007; Lesleighter et al, 2008).

2.3 A brief overview

There are a number of textbooks that explain in detail the theory and numerical implementations of CFD technology, for example, Abbott & Basco (1989), Wilcox (1993) and Versteeg & Malalasekera (1995). For hydraulics applications, the governing equations describing the behaviour of incompressible water (constant density, ρ) are the conservation of mass (ie. the continuity equation) and momentum (ie. the Navier-Stokes equation).

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \qquad (1)$$

Navier-Stokes equation:

$$\frac{1}{\rho}\frac{\partial p_d}{\partial x}=-\frac{Du}{Dt}+\upsilon\nabla^2 u \qquad (2a)$$

$$\frac{1}{\rho}\frac{\partial p_d}{\partial y}=-\frac{Dv}{Dt}+\upsilon\nabla^2 v \qquad (2b)$$

$$\frac{1}{\rho}\frac{\partial p_d}{\partial z}=-\frac{Dw}{Dt}+\upsilon\nabla^2 w \qquad (2c)$$

where $p_d = p + \rho gh$, a constant hydrostatic condition due to gravity, g; ∇^2 is the Laplace operator; and

$$\frac{Du}{Dt}=\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z} \qquad (2d)$$

These partial differential equations, inherently non-linear, are discretised both in space and time, and they can be solved using a variety of numerical schemes. Due to the complex nature of turbulence, it is often simplified and approximated using an averaged approach (eg. Reynolds-averaged Navier-Stoke). For practical purposes, the Re-Normalised Group k-ε turbulent energy dissipation equation has been rather successful for spillway modelling. Kenny (2007) examined the k-ε model, k-ω (shear stress transport) model and the Reynolds Stress Model, and found that there is insignificant difference in the prediction of water flow rate for an ogee spillway discharge.

In solving these equations, the main variables the analysis computes are velocity (a vector quantity), fluid fraction, pressure and temperature (scalar quantities) throughout the domain as a function of time for the given initial and boundary conditions. Other parameters such as flow rate, vorticity, turbulence and viscous energies, cavitation index, stream power, and distribution of Reynolds number and Froude number can be derived from these variables. In many cases, only the steady-state flow condition is of interest; but in other cases, for example, water releasing from a closed gate, the transient flow behaviour may need to be examined. Therefore, depending on the purpose of the analysis, the appropriate time-stepping scheme should be selected.

It is important to note that just like any other numerical modelling technique (eg. finite element analysis (FEA)), the need for validation against actual performance or theoretical solution, if available, is essential. The verification and validation of models are discussed further in section 3.3 of this paper.

3 SPILLWAY UPGRADE PROJECTS IN AUSTRALIA

Table 1 lists a number of spillway upgrade projects carried out by the authors to date that utilised CFD modelling to investigate hydraulic performance. They are listed in chronological order with the initial projects being mostly 2D or half symmetrical 3D models using tens of thousands of cells (figure 1). With increasing computing power, improved software features and modelling experience, more complex geometries involving multiple bays, and extensive upstream and downstream topography were included in models consisting of several million cells (figure 2). A variety of spillway types were analysed, for both existing and proposed structures.

3.1 Methodology

A general methodology is summarised in the flowchart in figure 3. It should be noted that there will be variation between projects depending upon the purposes of analysis and project requirement. Similar to other numerical modelling, such as

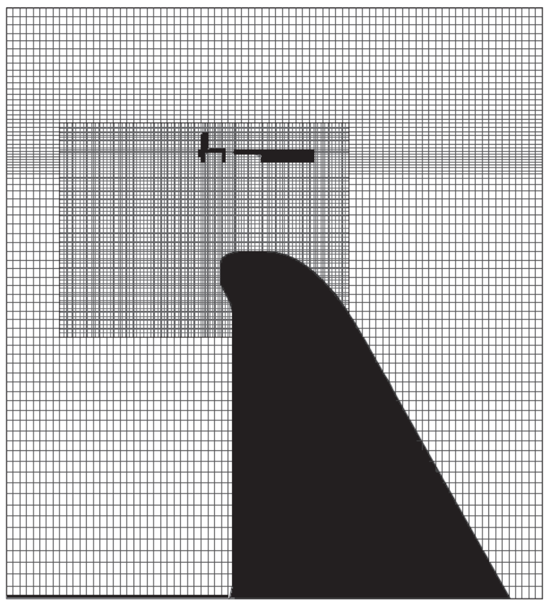

Figure 1: A 2D geometry and grid of the Hume Dam spillway.

Table 1: Summary of spillway upgrade projects using CFD modelling (see table 2 for validation details).

Dam, location	Spillway type	Chute	Number of bays	Gate type	Crest type	Bridge piers (BP) or divider walls (DW)	CFD geometry model	Validation	Re-evaluate rating curve for higher flood	Orifice or partial orifice flow	Overtopping or submerged flow	Pressure, velocity and flow depth distributions	Impact on gates	Impact on pier/bridge structures	Erosion assessment	Shockwave simulation	Hydraulic jump	Plunge pool hydraulics	Flip bucket behaviour	Apron or dissipator
Warragamba, NSW	Gated	Short	5 (drum gate at centre)	Drum and radial	Non-standard	BP	Local 3D model using symmetry	✓				✓	✓							
Hume, NSW	Gated	Short	29	Vertical-lift	Non-standard	BP	Local 2D and 3D models using symmetry	✓	✓	✓	✓	✓		✓					✓	✓
Buffalo, VIC	Gated	Short	3	Vertical-lift	Ogee	BP	Half 3D model	✓				✓								
Wivenhoe – existing main spillway, QLD	Gated	Short	3	Radial	Ogee	BP	Full 3D and half 3D models	✓	✓	✓		✓	✓				✓	✓		
Wivenhoe – auxiliary spillway, QLD	Fuse plugs	Short	3	None	Elliptical	DW	Full 3D model		✓					✓						
Goulburn Weir, VIC	Gated	Short	9	Radial	Non-standard	BP	Half 3D model	✓		✓	✓	✓	✓	✓						
Blowering, NSW	Uncontrolled	Long	1	None	Ogee	None	Full 3D model	✓	✓		✓	✓			✓	✓				
Tullaroop, VIC	Uncontrolled	Long	1	None	Ogee	None	Full 3D model	✓	✓		✓	✓		✓	✓	✓				
Tallowa – proposed dam upgrade, NSW	Gated	Short	21	Radial	Ogee	BP	Local 3D model using symmetry		✓			✓					✓	✓		✓
Copeton, NSW	Gated	Short	9	Radial	Elliptical	BP	One bay 3D model		✓	✓		✓	✓							
Pykes Creek, VIC	Uncontrolled	Short	1	None	Ogee	None	Full 3D model		✓			✓			✓		✓	✓		✓
Lake Manchester – proposed spillway upgrade, QLD	Uncontrolled	Short	1	None	Trapezoidal	None	Full 3D model	✓	✓			✓				✓		✓	✓	
Hinze, existing Stage 2 spillways and proposed Stage 3 spillway upgrade, QLD	Uncontrolled	Short (stepped) and long	3 (Stage 2/3) and 1 (Stage 1)	None	Slotted ogee	BP	Full 3D model	✓	✓		✓	✓			✓			✓		✓
Catagunya, TAS	Uncontrolled	Short	1	None	Ogee	None	Full 3D model	✓	✓		✓	✓			✓				✓	
Wyangala, NSW	Gated	Short	8	Radial	Elliptical	BP	Full 3D model	✓	✓	✓	✓	✓	✓							
Keepit – proposed spillway upgrades, NSW	Gated and fuse plugs	Short	6 (existing), 3 each (fuse plugs)	Radial (existing), none for fuse plugs	Ogee (existing), sharp crested (fuse plugs)	BP and DW	Full 3D large-scale model	✓	✓			✓								

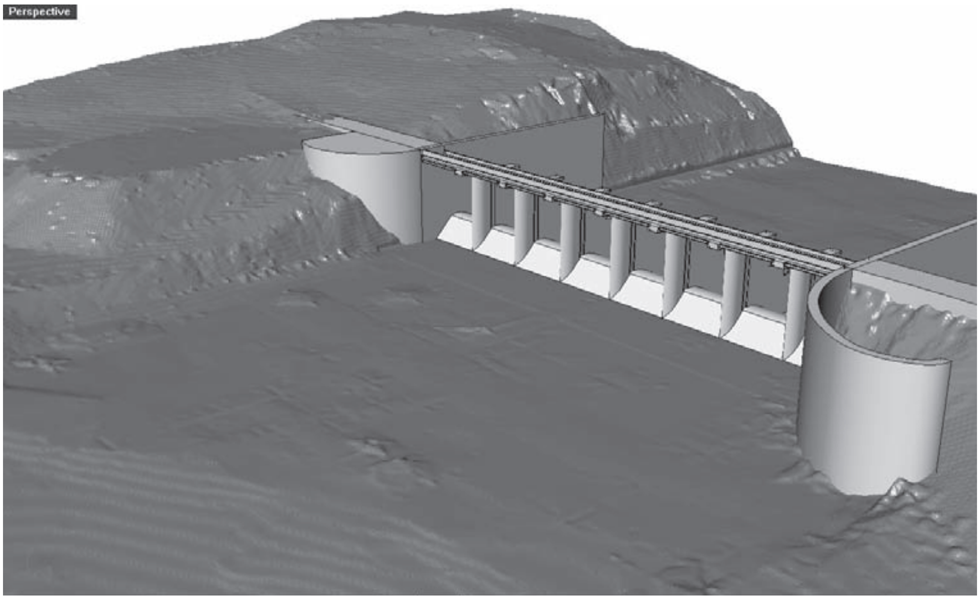

Figure 2: A 3D geometry model of the Wyangala Dam Spillway showing the upstream approach and downstream topography.

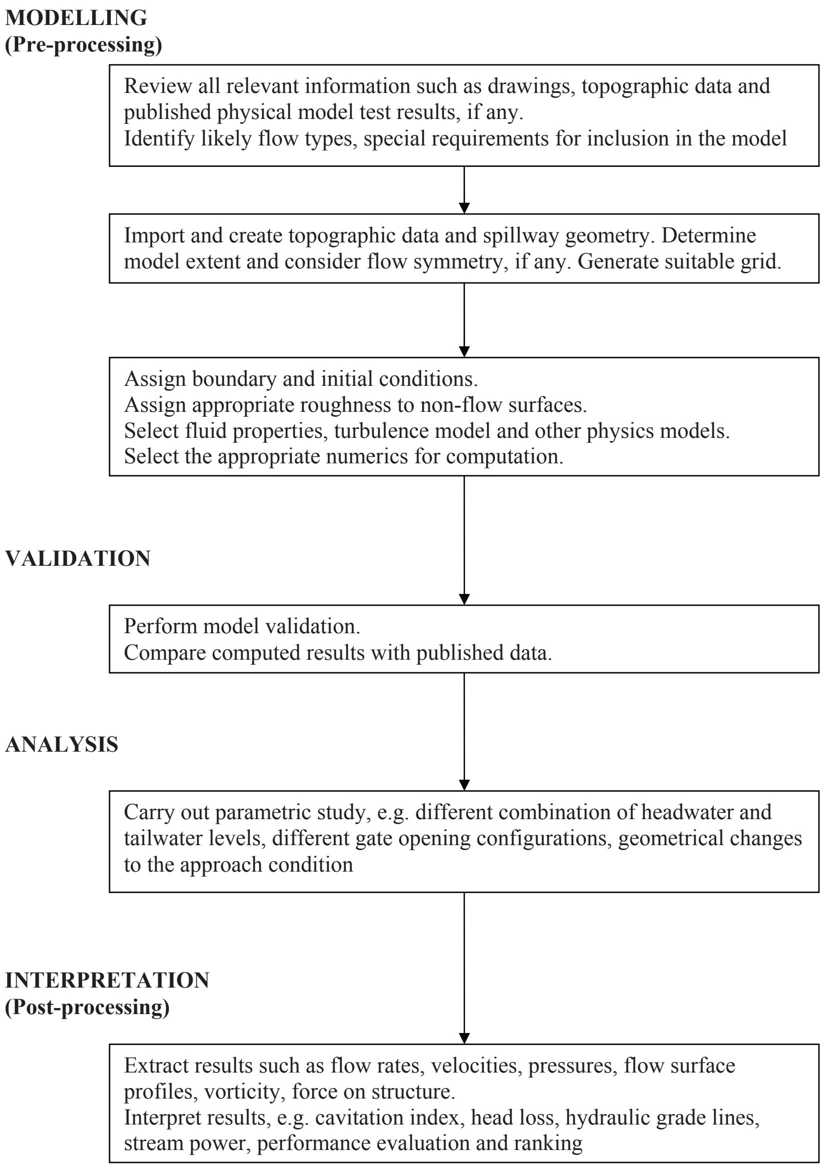

Figure 3: Flowchart showing a general methodology.

FEA, CFD analysis involves the three well-known stages: (i) pre-processing, (ii) analysis and (iii) post-processing (result extraction and interpretation). In addition, validation was also an important aspect for spillway modelling. Where possible, the models were set up so that the results can be compared against published data obtained by using the appropriate theory or previous physical model tests. This provided confidence in the model before embarking on extensive parametric runs.

3.2 Code selection

The CFD code, FLOW-3D, developed by Flow Science, Inc., was selected primarily for its ability to accurately model free surface flow, which is essential for modelling the open-channel flow behaviour that is commonly found in spillway flows. It utilises a true volume of fluid (tru-VOF) method for computing free surface motion (Hirt & Nichols, 1981) and the fractional area/volume obstacle representation (FAVOR) technique to model complex geometric regions (Hirt & Sicilian, 1985). The tru-VOF method tracks the sharp interface accurately and does not require computation of the dynamics in the void or air regions. If air-water interaction is deemed not important, the "single fluid" approach allows faster run-time. There are codes that require both water and air to be present in the analysis domain. This modelling approach may lead to excessive computation because both water and air flows are tracked in the analysis. Furthermore, the interface between the two fluids may not be sharply resolved.

The ability to model wall roughness (Souders & Hirt, 2003), air-entrainment (Hirt, 2003) and cavitation were also important considerations in selecting the code. The use of structured Cartesian grids or meshes meant the meshing process could be done very efficiently. The mesh was overlaid on the imported non-flow geometry and the FAVOR technique was used to determine the void or flow region within each cell (see figure 4). With finer grid spacing, the higher resolution of the non-flow region (obstacle) was achieved. The use of multi-block grids enabled larger domains to be modelled and the use of nested mesh blocks enabled more flow details to be captured in regions of interest. The code assumes the "law of the wall" (Rodi, 1980) to mimic the flow behaviour close to obstacles. As most real-scale discharges involve highly turbulent flow, the need to accurately capture boundary layer flow was of less importance.

Although the code solves the momentum and mass-transport equations in a transient, time stepping manner (ie. explicit scheme), only the steady-state flow condition was of interest for the upgrade projects. Generally, the results were examined after a dynamic "steady-state" was reached. This was deemed to be achieved when the flow rate and energy levels in the domain reached a steady-state.

A validation exercise on a standard ogee crest spillway was successfully conducted by Savage & Johnson (2001) using the same code, which provided further confidence in the analysis technique. An independent study was performed by Edwards (2006) where the discharge coefficient, C_d, as a function of head for an elliptical crest spillway was investigated using two CFD codes (CFX and FLOW-3D). Figure 5 shows both codes gave similar C_d values for a range of head levels, and they compared well with the experimental data reported by Maynord (1985). Note that the CFD results tend to, but not always, overestimate the discharge in comparison with physical model data.

All projects described in this paper were carried out by the authors using FLOW-3D. Caution is advised when applying the present findings to other CFD codes.

3.3 Model validation

Validation of numerical models is extremely important and therefore it formed one of the analysis tasks in most of the spillway upgrade projects. An on-going effort to carry out validation against published or experimental data remains essential. This is to ensure modelling correctness and to provide a high confidence level in its application. The American Society of Mechanical Engineers (2006) addressed the fundamentals of verification and validation of computational simulations and so did the American

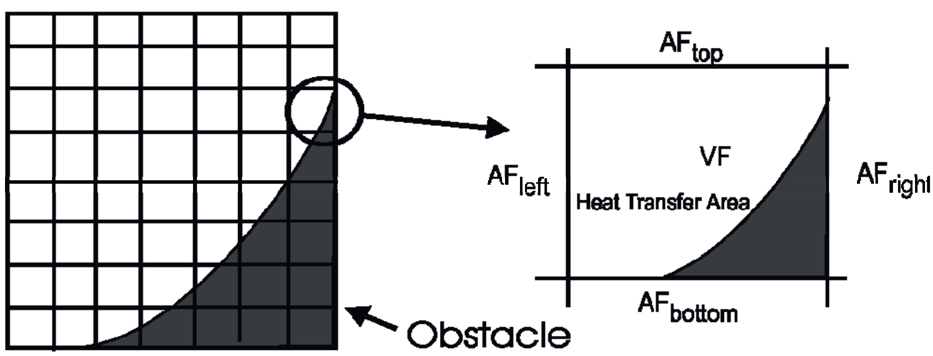

Figure 4: FAVOR representation of a non-flow region, where VF is Volume Fraction and AF is Area Fraction (Flow Science, 2008).

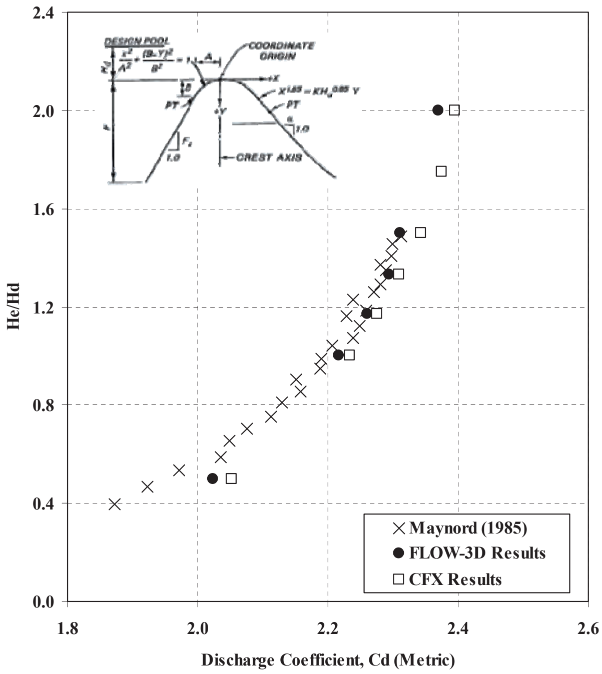

Figure 5: Discharge coefficient as a function of head for an elliptical crest spillway at $P/H_d = 2.0$. Results comparison based on two CFD programs (Edwards, 2006).

Institute of Aeronautics and Astronautics (AIAA, 1998) regarding CFD simulations. The inter-relationship between the real world, the mathematical model and the computer model has been carefully researched by academics, software developers and practitioners. It should be emphasised that both computational and physical models are approximate representations of the real behaviour.

According to the AIAA verification and validation guidelines (AIAA, 1998), for the purpose of CFD modelling the following terms have been defined. The term "model" means a representation of a physical system or process (intended to enhance our ability to understand, predict or control its behaviour). The term "modelling" is defined as the process of construction or modification of a model. The term "simulation" is defined as the exercise or use of a model (ie. a model is used in a simulation).

The definition of mathematical model and computer model is as follows. A "mathematical model" or "conceptual model" consists of all the information, mathematical modelling data and mathematical equations (eg. partial differential equations) that describe the physical system or process of interest. A "computer model" or "computerised model" is an operational computer program (ie. code) that implements a mathematical or conceptual model.

Furthermore, the definition of verification and validation is as follows. "Verification" is the process of determining that a model implementation accurately represents the developer's conceptual description of the model and the solution to the model. "Validation" is the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model.

The inter-relationship between the real world, mathematical model and computer model is represented in figure 6. A proper validation process (see figure 7) will involve comparing the computed results with those taken from the actual structure under comparative flow conditions. Unfortunately, for spillways, extensive monitoring and measurement at the site is unusual because of the high costs involved, and extreme events are rare. In practice, even under relatively high flood discharging through spillways, the hydraulic behaviour is rarely quantitatively measured, only qualitatively observed from videos or photographic evidence, if at all. Therefore a proper validation is difficult to achieve. Unlike the aircraft or the automobile industries where full-scale prototypes can be instrumented and tested extensively, spillway performance is usually inferred from physical models. Unfortunately, as physical models are also approximate representations of the real world, they too suffer from the same problems faced by the computer models, ie. the physical model results cannot be compared with real world performance, especially for extreme events such as PMF discharge.

Due to the lack of actual performance measurement, the current validation practice for spillway models can only be regarded as "pseudo-validation", as shown in figure 8. Bearing this in mind, the use of the term validation will mean "pseudo-validation" throughout this paper.

In many of the upgrade projects carried out by the authors, where possible, at least one flood level was analysed so that validation against physical models or published design charts such as those by the US Army Corps of Engineers/Water Experiment Station (USACE, 1952; 1995) could be performed. It should be noted that the information provided by design charts was based on limited physical model tests confined to a certain spillway crest shape and upstream approach condition, and hence may only provide rough validation for non-standard crest shapes or unusual approach conditions. The available data generally included the following parameters suitable for validation purposes:

- discharge quantity and/or discharge coefficient as function of upstream head – this is a measure of how efficient the spillway is in passing flood water
- pressure distribution along the spillway – this is to examine the potential for cavitation damage due to excessive sub-atmospheric pressure
- flow surface profile – this becomes important when the flood water may interfere with other structures, such as bridges at the crest or raised gates, or it may overtop chute walls.

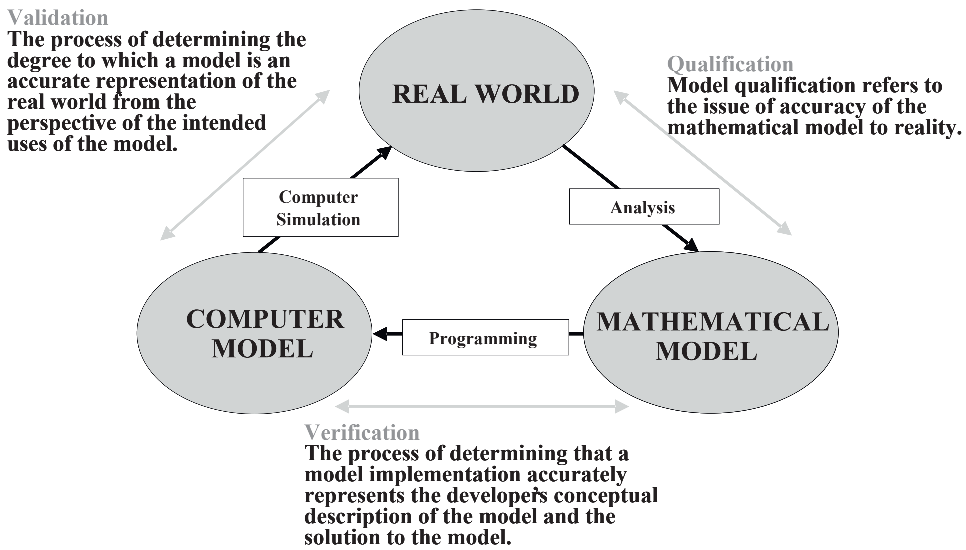

Figure 6: Inter-relationship between the real world, mathematic model and computer model.

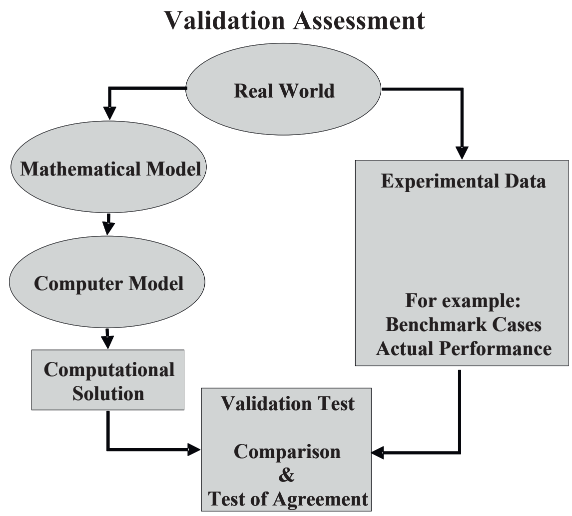

Figure 7: Proper validation process.

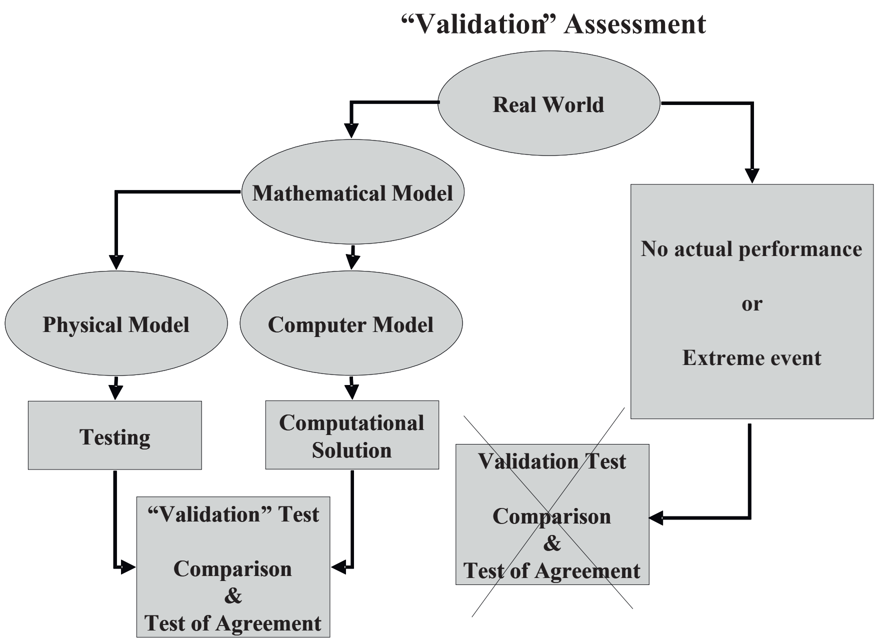

Figure 8: Pseudo-validation process. A proper validation is not possible due to lack of actual performance data.

Sometimes the velocity profile was also available for comparison. More recent studies also had photographs and videos that could be used to qualitatively examine the flow surface profiles. Table 2 shows the parameters used for validation for the spillway upgrade project models performed by the authors to date. The following sections provide additional commentary on validation of these parameters.

3.3.1 Discharge

Table 3 shows the computed flow rates compared against those obtained from either physical models or from design charts for most of the spillway upgrade projects. In general, the comparison is encouraging despite a couple of large differences, which typically occur for a low head when only a small volume

Table 2: Validation performed for the spillway upgrade projects.

Upgrade projects	Physical model scale (year of test)	USACE/ WES	Flow rate	Pressure distribution	Free surface profile	Velocity profile	Other validation sources or items validated against physical models
Warragamba Dam (drum gate bay)	1:100 (1991)	✓	✓	✓	✓	✓	An Ogee crest spillway was modelled and results validated against USACE/WES data to check the viability of the CFD technology.
Warragamba Dam (radial gate bay)	1:100 (1991)	–	✓	✓	✓	–	
Hume Dam	1:50 (1962)	–	✓	✓	✓	–	
Buffalo Dam	Not used	✓	✓	✓	✓	–	Used published data
Wivenhoe Dam (main spillway)	1:80 (1979)	–	✓	✓	✓	✓	
Goulburn Weir	Not used	–	✓	✓	–	–	Checked against theoretical flow rates
Blowering Dam	1:80 (1971)	–	✓	✓	✓	–	Shockwaves validated against theory
Tullaroop Dam	1:30 (1958)	–	✓	✓	✓	✓	
Tallowa Dam (proposed spillway)	Not used	–	✓	–	–	–	Checked against theoretical flow rate. Drowned hydraulic jump behaviour compared with published guidelines.
Copeton Dam	Not used	–	✓	–	–	–	Checked against theoretical flow rates
Pykes Creek Dam	Not used	–	✓	–	–	–	Checked against theoretical discharge rates
Lake Manchester Dam (proposed spillway)	1:40 (2007)	–	✓	✓	✓	–	Trapezoidal weir validation against published laboratory experiment.
Hinze Dam (existing and proposed spillway)	1:50 (2007)	–	✓	✓	✓	–	Vortex formation at abutment (qualitative). Energy loss validation against published experimental data.
Catagunya Dam	Not used	✓	✓	✓	–	–	Flip bucket performance validated against published data.
Wyanagala Dam	1:80 (2006)	–	✓	–	–	–	
Keepit Dam (proposed spillways)	Not used	–	✓	–	–	–	Compared discharge against theoretical

Table 3: Validation summary for discharge.

Case No.	Head above crest (m)	Flow rate (m^3/s or $m^3/s/m$ run)		Percentage difference (%)	Comments
		Reported	CFD result		
1	15.29	3350	3660	9.25	Non-standard ogee crest. Original design head. Physical model test results.
2	3.048	9.86	9.59	–2.74	Non-standard ogee crest. Physical model test results.
	6.096	29.39	30.26	2.94	
	7.544	41.56	43.26	4.09	
3	20	11750	12400	5.53	Ogee crest. Original design head. Physical model test results.
4	4.4958	793	790	–0.36	Ogee crest. Physical model test results.
5	3.805	162	167	2.99	Non-standard crest with downstream steps.
6	3.35	13.5	13.9	2.96	Ogee crest. USACE design chart.
	4.466	22	22.1	0.45	
7	1.2496	142	167	17.60	Ogee crest. Physical model test results.
	7.0106	2352	2480	5.41	
8	14.935	15280	15696	2.72	Low elliptical crest. Original design head. Physical model test results.
9	1.00	200	191	–4.5	Elliptical crest. Theoretical flow rates.
	2.00	600	584	–2.67	
	3.00	1000	990	–1.00	
	3.50	1200	1190	–0.83	
10	8.8	2350	2540	8.09	Trapezoidal crest. Physical model test results.
11	5.486	3536	3515	–0.59	Ogee crest. Discharge estimated using design chart.
	8.066	7450	7370	–1.07	
	9.426	10300	10225	–0.73	
12	Slotted ogee crests	4145	4173	0.68	Ogee crest (two levels). Physical model test results.
13	19.538	21000	20900	–0.48	Low ogee crest. Discharge extrapolated from physical model results.
14	18.295	13527	14670	8.45	Ogee crest. Estimated discharge for higher head.

of water is flowing over the spillway. The relative coarseness of the mesh at this low head level was probably the reason for this large discrepancy. The discharges for different heads as shown in table 3 were typically carried out with a constant mesh density. For lower head, the critical depth will be lower and therefore the grid spacing should ideally be reduced as well.

Kenny (2007), based on CFD research work carried out for a 2D ogee crest spillway using the software FLUENT, recommended the grid spacing in the region of interest to be 0.3 m for concept design and 0.1 m for detailed design. As discharge over a spillway crest is controlled by the critical depth, y_c, for a particular upstream head, the recommended grid spacing may be normalised as ~4.0% y_c for concept design and ~1.3% y_c for detailed design. It should be noted that Kenny's research was carried out using FLUENT, which uses a body-fitted gridding system and may apply a different wall treatment compared to that described above for FLOW-3D. Caution is advised if using Kenny's recommended grid spacing with software other than FLUENT. The use of a nested grid in combination with analysis restart would have improved the solution accuracy for the low discharge cases in table 3. At the time, however, these models were set up mainly to investigate the effect of much higher PMF discharges.

The head-discharge relationships for three selected cases are shown in figures 9 to 11. In general the discharge or flow rate, Q, as a function of the total energy head above the spillway crest, H, is governed by the following equation:

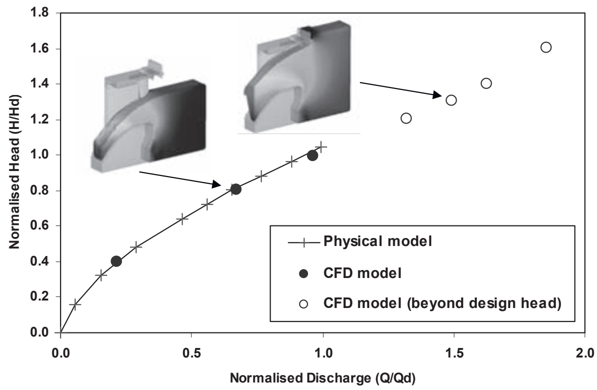

Figure 9: Head-discharge curve (Hume Dam spillway). Effects of higher heads were modelled.

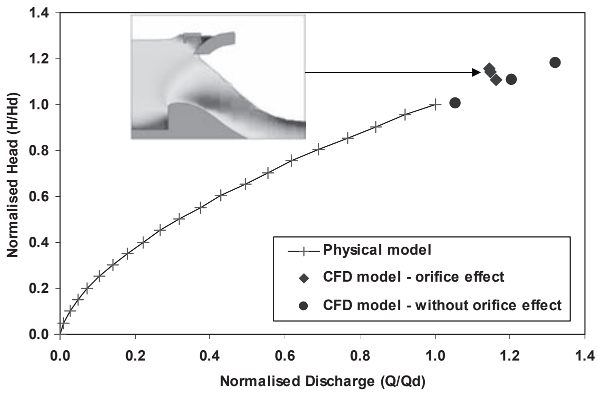

Figure 10: Head-discharge curve (Wivenhoe Dam main spillway). Effects of orifice and transitional flows were captured at higher head levels.

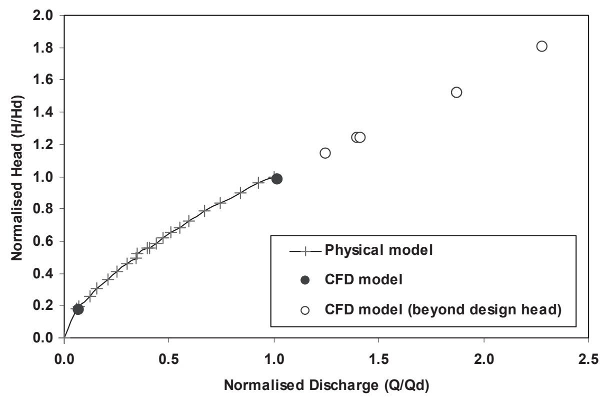

Figure 11: Head-discharge curve (Blowering Dam spillway). Effects of high and low heads were modelled.

$$Q = C_d B H^{1.5} \tag{3}$$

where B is the clear width of the spillway and C_d is the discharge coefficient. The effects due to abutments, piers, etc. are incorporated into the discharge coefficient. The plotted head and the discharge values were normalised by their original design head, H_d, and discharge, Q_d.

It can be seen that the computed results compared well with the physical model results. The discharge at head levels greater than the design could easily be computed using the CFD model. The computed flow surface and velocity contours for selected heads are also shown in figures 9 and 10. In figure 10, when the head levels were sufficiently high for the flow to impact on the crest bridge and/or the underside of the raised gates, a transition flow between the free discharge flow and a full orifice flow was captured. This result highlighted potential safety and operational implications of PMF flow. This flow behaviour will be examined further in section 3.4.

Limited validation exercises were carried out for discharges through morning glory and labyrinth spillways. The morning glory spillway features free surface weir flow and pressurised pipe flow, as well as the potential for vortex inducement and hydraulic jump. The preliminary results show very good agreement with the experimental data obtained by the USBR (1991) for the head-discharge relationship, including the transition from weir flow to pipe flow (figure 12). Repetitive symmetry boundary conditions were used for the 3D labyrinth spillway investigation. The computed head-discharge relationship (figure 13) is in reasonably good agreement with the physical model results reported by USBR (1982). There is scope to perform further validation on these types of spillways in the future. They will be the subject of future papers when more results are available.

3.3.2 Pressure distribution

The computation of spillway pressures at the crest, chute and flip bucket have been validated for various spillway upgrade projects carried out by the authors. Ho et al (2003) described an extensive crest pressure validation study that was carried out for a standard ogee spillway, with and without the pier influence. The results showed good agreement with the published data. The following paragraphs describe further studies that have been carried out to date.

Validation of the pressure distribution over a trapezoidal-profile weir was performed as part of the Lake Manchester Dam upgrade project. The computed pressure distribution highlighted the importance of adequate pressure measurements in the physical model, as shown in figure 14. The numerical model was able to compute pressure at corners where very high negative or positive pressure developed, depending whether the corner was concave or convex. It should be noted that physical models cannot capture pressure at sharp edges or corners due to instrumentation difficulties, but peak pressure can be inferred or interpolated from measurements taken in the vicinity of corners.

Besides capturing pressure distribution at the crest, the CFD model is able to compute pressures along the chute floor. Large positive or negative pressure can occur at locations where a change of grade occurs,

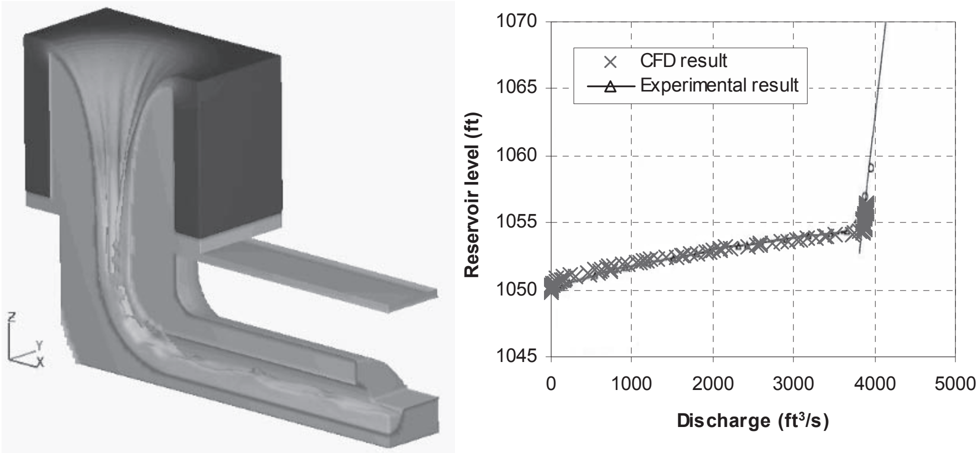

Figure 12: Head-discharge curve (right) for a morning glory spillway. Sectional view showing volume of water flowing inside the spillway (left).

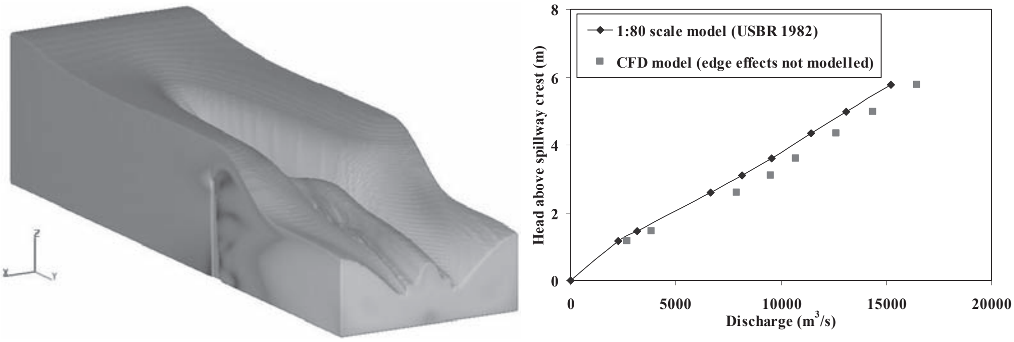

Figure 13: Head-discharge curve (right) for a labyrinth spillway. Iso-view (left) showing water discharging over the spillway in a half 3D model utilising repetitive symmetry.

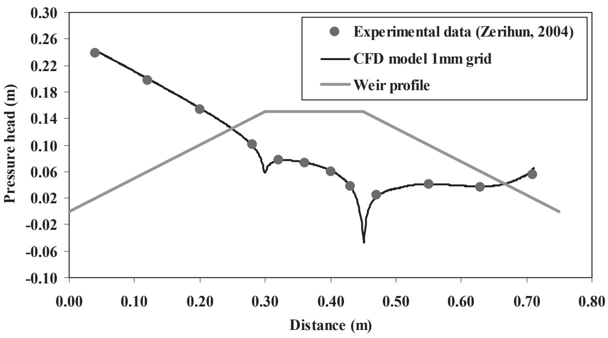

Figure 14: Comparison of crest pressure of a trapezoidal weir for CFD and experimental results.

potentially having undesirable impacts on the floor slabs. Figure 15 shows the floor pressure distribution along the centreline and near to the right training wall for a long chute with multiple grade changes over a distance of 250 m beyond the spillway crest. Although the physical model data was only obtained in a few selected locations, the CFD results compared well with the measured data. The CFD model was able to capture the pressure for the entire area along the chute that provided more useful information than the physical model. In addition, the velocity distribution along the entire spillway chute and beyond was computed for erosion assessment purpose.

Very often, a flip bucket is located at the downstream end of the spillway to dissipate the kinetic energy from the flow prior to entering the river course. The pressure distribution from the tangent point to the lip of the flip bucket is shown in figure 16 for a flip bucket validation study. Also plotted in the figure is the pressure based on the Hydraulic Design Criteria, chart number 112-7 (USACE, 1952) for comparison purposes. This theoretical pressure can be estimated from the geometry of the flip bucket, the head and the flow rate. It should be noted that zero head loss was assumed in the theoretical calculation, while energy losses due to turbulence and viscous effects have been included in the CFD model. It can be seen that the computed results are in reasonable agreement with those estimated by the design chart, including a slight drop in pressure due to losses. The occasional pressure spikes are probably due to the mesh density and the way the pressure is extracted from the model. When a curved geometry is modelled in rectangular coarse mesh, some cells may have very small VOF and hence reduced accuracy.

When excessive sub-atmospheric pressures occur along a spillway, there is an increased potential for instability of the spillway structure and damage to the concrete face of the spillway due to cavitation. This is where small bubbles of water vapour form at areas of low pressure, then subsequently collapse

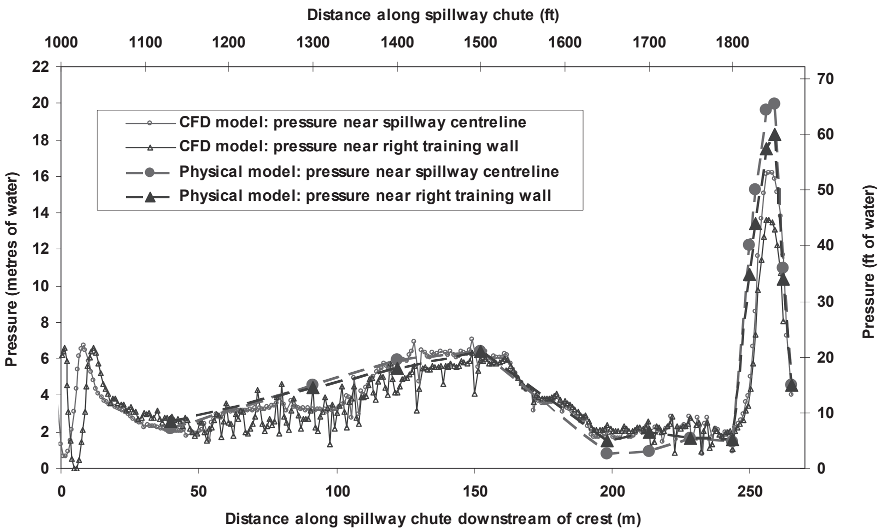

Figure 15: Pressure distribution along spillway chute.

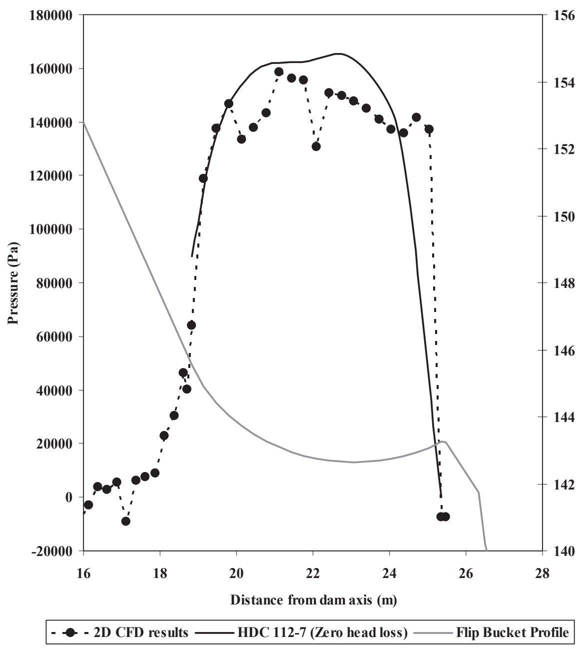

Figure 16: Pressure distribution in a flip bucket.

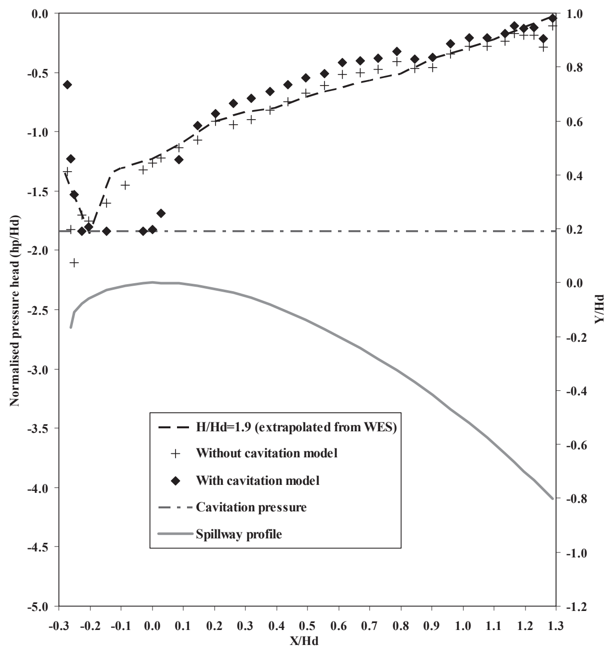

Figure 17: Normalised pressure head distribution over an ogee crest for $H/H_d = 1.9$ case with and without cavitation effect.

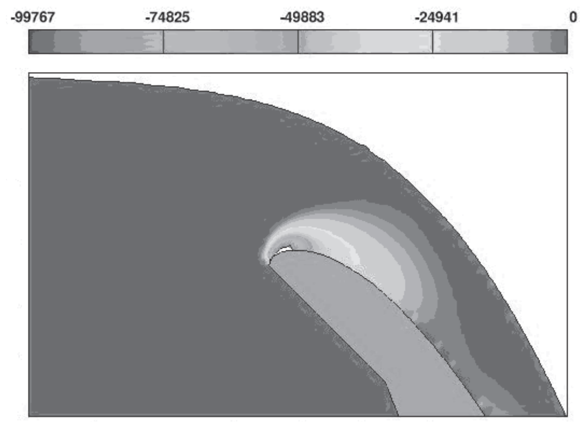

Figure 18: Negative pressure (Pa) contour over the crest with a cavitation pocket.

as they travel into a region of higher pressure. The collapse action can impart sufficiently high point loadings as to damage concrete structures.

The effect of cavitation was tested on a 2D model of an ogee crest spillway with a high head. Falvey (2007, personal communication) suggested that cavitation may be a concern when the total energy head above the spillway crest, H, is greater than $1.3H_d$. By incorporating a cavitation pressure limit of –98.97 kPa in the analysis for the $H/H_d = 1.90$ case, the negative pressure was capped as shown in the normalised pressure head plot in figure 17, and a cavitation pocket was formed when the negative pressure reached this limit as shown in figure 18. In reality, any imperfections such as small indentations on the crest may trigger localised cavitation resulting in the formation and subsequent collapse of vapour bubbles. At present the CFD model can only simulate the global cavitation behaviour. A much finer grid, perhaps of the order of millimetres, may be required to capture these cavitation bubbles. The time for a bubble to collapse is of the order of microseconds (Falvey, 1990), and therefore the analysis run time may be excessive or even prohibitive for most upgrade projects. This kind of detailed analysis may be suitable for future research study.

As an alternative to direct modelling of cavitation physics, by knowing the pressure and velocity close to the floor, the cavitation index, σ, which can be used as a measure of cavitation potential, can be computed by the following equation.

$$\sigma = \frac{p - p_v}{\rho v^2 / 2} \tag{4}$$

where p = water pressure at the floor, p_v = vapour pressure of water, ρ = density of water and v = velocity of water close to the floor.

A typical contour plot of the cavitation index is shown in figure 19 for a complex spillway arrangement. Typically physical models obtain pressure data at the centreline and one other profile. By plotting data across the entire spillway crest, CFD can pick up areas prone to cavitation damage not on the centreline. Whether cavitation can occur will depend on the critical cavitation index, which is a function of spillway geometry, aeration and flow condition. Past investigation of cavitation damage found that cavitation is a major concern when the flow velocity in a spillway reaches or exceeds about 25 m/s or the local cavitation index is less than 0.2 (Zipparo & Hasen, 1993).

3.3.3 *Flow surface profile*

The accurate computation of the free surface is essential because at high flood levels the flow may potentially impact on the crest structures causing

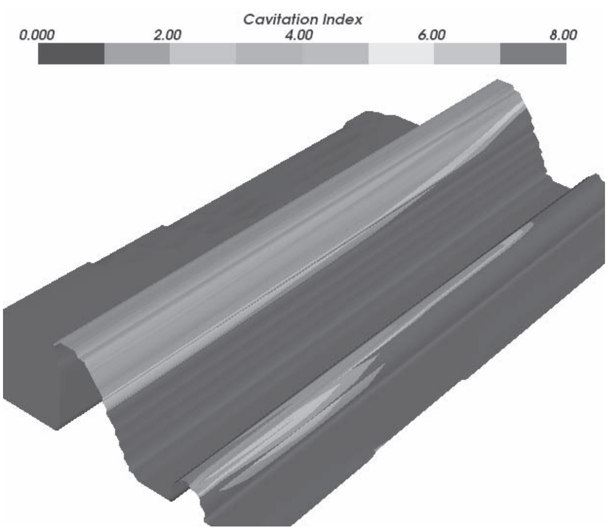

Figure 19: Cavitation index can be plotted graphically for a spillway model.

damage to the crest bridge or raised gates. Beyond the spillway crest, the discharging water may overtop chute walls if they are not high enough to contain the flow. This event may lead to erosion of the chute and/or dam foundations, which may significantly reduce the stability of the dam.

The authors have carried out validation studies for the flow surface profile over various crest shapes, including pier effects, formation of cross-wave or standing waves in contracting chutes, jet trajectory from a flip bucket, and for various other spillway geometries and features. Earlier validation of flow surface for the standard ogee spillway with and without the influence of piers was carried out using USACE data and the results can be found in Ho et al (2003). It was found that good agreement was obtained.

Figure 20 shows a comparison of flow surface between the CFD model and the physical model along the centreline and along the pier for a low elliptical crest spillway. The outlines of the spillway, raised radial gate and crest bridge are also plotted. The CFD result shows good correlation to the measurements. The influence of the pier on the flow surface was also captured correctly. This CFD model was further analysed for higher flood levels until the flow surface just touched the underside of the radial gate forming an orifice flow.

Further validation of cross-wave or shockwave formation for the supercritical flow through a contraction in a long spillway chute has been reported by Ho et al (2005). The results demonstrated that the CFD model can adequately compute the correct flow surface for practical purposes.

In the case of capturing jet trajectory, a water jet discharging from a flip bucket is shown in figure 21. The theoretical free jet trajectories (USACE, 1995) are also plotted in the figure for comparison purposes. The top curve is for a lip or exit angle of 30° and a velocity of 20 m/s. The lower curve used the averaged exit angle of 16° and the mean exit velocity of 20 m/s determined from the CFD results. The average exit velocity is slightly lower than the theoretical value deduced from the conservation of potential and kinetic energies, assuming no energy loss between the spillway crest and the flip bucket. It can be seen the computed throw distance is in good agreement with the theoretical free jet trajectory. Note that the theoretical trajectory does not take into account energy loss or air resistance. With this 2D model, further parametric studies can be performed to investigate the effects of air resistance and concrete surface roughness on the throw distance for different upstream heads.

In a 3D CFD model for the Lake Manchester Dam upgrade where a jet of water is discharging from a short, contracting chute with a horizontal flip bucket, the computed results showed that the concave wave

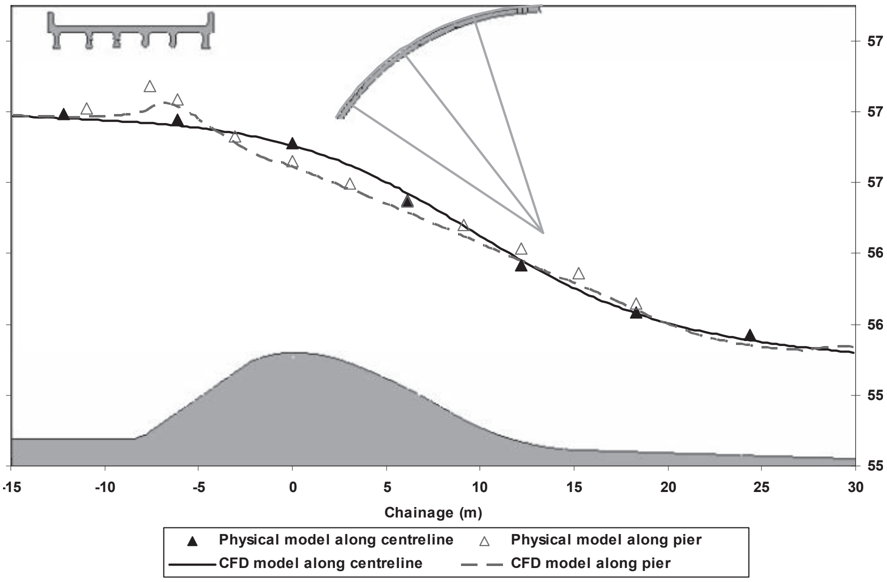

Figure 20: Flow surface comparison. Outline of piers not shown for clarity.

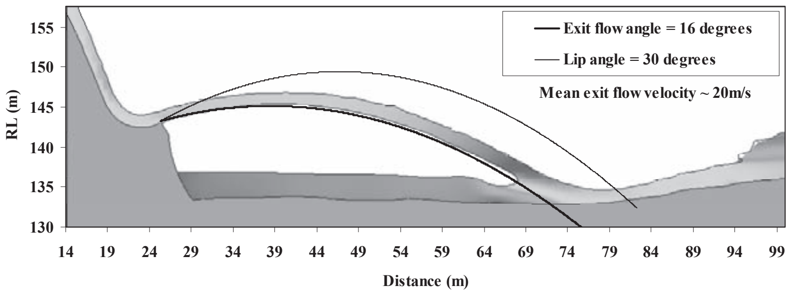

Figure 21: Flow trajectory from a flip bucket (H/H_d = 1.00).

formation in the chute was not as strong as that observed in the physical model. This may be due to the coarse grid resolution at the exit location or the choice of turbulence model in capturing flow separation. Details of the numerical and physical modelling were reported by Lesleighter et al (2008). The authors plan to carry out further numerical modelling to investigate this discrepancy.

3.4 Accurate modelling of flow behaviours

Due to the general-purpose nature of most CFD codes, the user must take special care that suitable parameters and model set up conditions are applied to ensure flow behaviours are accurately captured. Three aspects are discussed in detail.

3.4.1 Transition from free flow to obstructed flow

One of the interesting flow behaviours captured by the CFD model is the change in flow regime as the upstream head rises. If there is no interference to the flow as H increases, then the discharge will simply follow free spillway flow as described by equation (3), except the discharge coefficient may increase slightly due to increased suction effect over the spillway crest. However, the situation can be complicated by the presence of crest structures

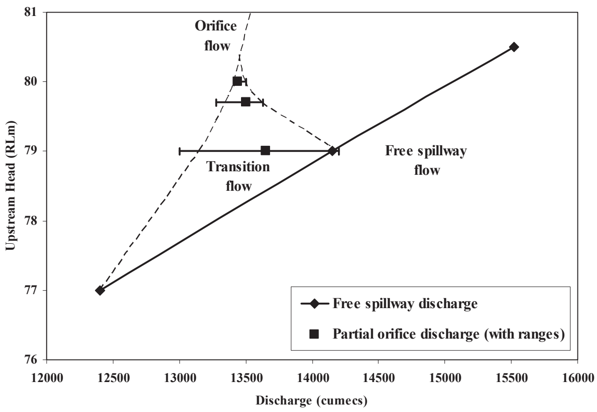

Figure 22: Head-discharge curve showing the transition flow region.

such as bridges and gates. As the upstream flood level increases, the discharging water can undergo different stages of flow behaviour, starting from a free spillway discharge, then a transition flow, followed by an orifice flow and eventually the water overtops the crest structure if the flood level is high enough. This sequence of flow behaviour was investigated for the Hume and Wivenhoe main spillways.

A computed rating curve is shown in figure 22 for the Wivenhoe Dam main spillway using a detailed 3D model that consists of three bays. It was found that a range of discharge is possible for a given head within the transition zone. This behaviour is best understood by observation of the transient flow of water in the 3D model. The upper nappe coming through a bay is influenced by the piers on either side of the bay and the upstream approach condition into the bay. Although the bow waves generated by the piers are expected to be symmetrical across the bay, this is only true if the approach flow is also symmetrical and normal to the spillway crest. The fact the incoming flow from the reservoir has to go through an approach channel that has non-symmetrical cross-section means the approach flow is no longer symmetrical by the time it reaches the bays. As the water flows over the spillway crest, the highest upper nappe will first impact on the underside of the skinplate of the raised radial gate (this is for the situation where the upstream head is high enough to impact on the raised gates). The impact zone will spread sideways, but the remaining flow area passes below the skinplate, thus creating a partial orifice flow condition. Interestingly, the extent of this impact zone across the skinplate changes with time and this behaviour also varies from bay to bay because of the transient non-symmetrical nature of the approach flow. At the impact zone, water becomes trapped between the impacting flow and the skin plate. The trapped water level builds up until its head becomes sufficient to cause a "flushing" regime where the trapped water escapes and the process begins again. This resulted in periodic surges of water flowing through a bay. This is captured in the analysis by observing the fluctuating flow rate through a bay with time resulting in a range of discharge. This range becomes narrower or smaller as the upstream head is higher as orifice flow begins to dominate. Eventually, when the head level is high enough, a complete orifice flow will occur when the entire skin plate is impacted upon across all the bays and the discharge converges to a single curve. Depending on the raised gate and crest bridge configuration, overtopping may eventually occur when the upstream head level is sufficiently high. In this event, further deviation of discharge curve or even another transition zone can occur.

The above transition flow regime can only be captured correctly using a 3D model of the entire spillway. A 2D or a half 3D symmetric model will not model the orifice flow spreading across the bay and the different flow regimes occurring at different bays. The modelling strategy may need to be considered carefully because multiple discharge solutions can exist for a given head. Numerically, an implicit scheme developed to only capture a steady-state flow condition may find only one discharge solution, but not be able to find the range of discharge in the transition zone. In the worst case, the solution may fail to converge. Therefore an explicit transient scheme is more suitable. The discharge history will be computed and the maximum and minimum discharge can be extracted.

3.4.2 Boundary condition

The choice of upstream boundary condition in a spillway model can affect the accuracy to which the user is able to determine the head-discharge relationship for the system. Suppose the upstream boundary is sufficiently far away from the first control section in the model, if a head (pressure) control is used at the boundary, the discharge history can be easily computed at any spillway cross-section and the transient flow behaviour can be observed. However, if a fixed flow rate is introduced at the boundary, the velocity head and flow depth above the spillway crest has to be monitored with time, and a suitable averaging of various upstream locations conditions carried out to determine the appropriate head level. An inappropriate selection of head measurement locations may reduce the accuracy of the computed head-discharge relationship.

3.4.3 Initial condition

Besides the boundary condition, the initial model condition may affect the behaviour of the flow. Ideally, the upstream boundary head would be set to rise gradually over time simulating the full flood hydrograph and ensuring the flow conditions are accurately built up over time. But this will take a much longer runtime especially if it is a large 3D model. In present practice, the time frame is generally prohibitively long. More commonly, an initial block of water corresponding to the upstream head is placed in the upstream side of the spillway with a vertical wall of water standing at the crest. The analysis will then commence with this wall of water rushing down the crest towards the downstream outflow boundary, like a dam-break type of simulation. In most cases where the crest geometry is smooth without sharp change in geometry, the correct flow regime is simulated. In the case of sharp crested weir or labyrinth spillway, the water may cling to the downstream face of the weir generating negative pressure, which may not be realistic for a high upstream head. This situation can be alleviated by either introducing air or a fictitious cavitation pressure so that the lower nappe can separate from the wall surface.

However, if there is insufficient aeration behind the nappe, the stream of water may fluctuate as can be observed in reality. A combination of techniques, such as using a gradual rise in the reservoir level, controlling the air supply and using a two-fluid (air and water) model may be required to correctly simulate this phenomenon.

Clearly, good engineering judgement and an understanding of the expected flow patterns is essential for critical review of the analysis results.

4 WATER-STRUCTURE INTERACTION

An understanding of the interaction between hydrodynamic flows and submerged structures is of importance when determining suitable structural design loadings and examining the potential for structural dynamic response. In most CFD analyses at the present time, non-flow regions are assumed to be rigid and cannot deform as water flows pass them. However, if a submerged structure is flexible and can deform significantly under hydrodynamic pressure, then a proper water-structure interaction analysis is required because the flow behaviour will be influenced by the deforming structure. Depending on the degree of structural flexibility, various approaches to water-structure interaction are presently available:

1. Rigid and non-deformable structure. The hydrodynamic load can be extracted directly from the CFD analysis. For example, forces and pressures on concrete piers, bridge beams and spillway surface. Validation studies by the authors have shown that the drag force acting on a slender member submerged in a steady current computed by the CFD model is in reasonable agreement with theoretical values.
2. Submerged structure that may be considered to be deformable, but the deformation under hydrodynamic condition does not significantly affect the water flow. This can be assessed by considering the structure to be rigid initially in the CFD analysis and the computed pressure on the structure will then be used in a structural analysis (eg. FEA) to determine the amount of deformation the structure may experience. Some engineering judgement will be needed to determine if the deformation is enough to alter the flow behaviour.
3. Submerged structure that will deform considerably under hydrodynamic pressure. A fully-coupled fluid-structural analysis or an iterative sequentially coupled fluid-structural analysis will be required.

In recent years, a technology convergence has been occurring between CFD and FEA software. General-purpose CFD packages may now include movable structures, flexible membranes and computation of simple internal structural stresses. FEA packages have incorporated deformable fluid elements. Between these lie interfaces that allow CFD and FEA software to interact in an iterative manner with respect to analysis time. Each alterative has its strengths and weaknesses, but none yet provide a full solution to fluid-structure interaction.

A few FEA software packages claim full fluid-structure interaction can be done, but at the time of writing the fluid flow is typically limited to laminar flow and for enclosed flow only. In the situation where a gate structure is impacted upon by a free surface flow during a PMF event, the flow will be highly turbulent. Irrespective of which approach was selected, assessment should also be made to determine the potential for a dynamic structural response due to a cyclical hydraulic load. For example, the vortex-induced vibration that can occur in a structural member that is submerged in a water flow. A simple assessment can be made by calculating the vortex shedding frequency using the computed flow velocity in the empirical formula:

$$f_l = S_t \frac{U}{D} \tag{5}$$

where f_l = vortex shedding frequency; S_t = Strouhal number, which is a function of cross-sectional shape and Reynolds number; U = flow velocity computed in the vicinity of the member; and D = cross-sectional dimension.

The vibration characteristic of the member, such as its natural frequencies and mode shapes, will also be required. The assessment methodology can be found in most structural dynamics textbooks or by modal analysis using FEA. This type of assessment was carried out in a number of upgrade projects for radial gates' members that will be submerged under the revised PMF event.

Additionally, if a transient analysis is performed, the "dynamic" steady-state load history can be examined to identify whether the hydrodynamic load experienced by the member has a "forcing" frequency that is close to any of the natural frequencies of the member to ensure the member is not undergoing resonance that may be detrimental to the stability or long-term performance of the structural system.

5 EROSION ASSESSMENT

In the absence of a reliable scour and sediment transport modelling capability in present-day CFD software packages, the erosion assessment can be performed in a number of ways, depending on the type of flow and the materials involved. Typically for spillway applications, the erosion potential due to flow discharge may be required for:

- an unlined approach channel or unlined spillway chute
- abutments adjacent to the spillway
- a water jet entering a plunge pool from a flip bucket
- an over fall cascade entering a plunge pool
- an overtopping flow impacting on downstream face of a concrete dam and foundation.

It should be noted that most scour models in CFD analysis are based on empirical models and can only deal with sediments consisting of purely non-cohesive deposit or purely cohesive deposit, but they cannot be applied to a deposit that has both friction and cohesion. Furthermore, they cannot be used on rock that is normally encountered in dam sites.

However, the CFD model can provide detailed data for the velocity field, whether it is a recirculation flow inside a plunge pool (see figure 23) or the secondary swirl in a channel flow.

For non-jet flow, a simple preliminary scour assessment may involve evaluating the peak velocity computed by the CFD model close to the floor in question and comparing it with the maximum permissible velocity for floor material available in a number of publications, for example, USACE (1994). The reliability of the computed velocity will depend on the floor geometry, roughness and mesh resolution.

A more rigorous approach is to compute the shear stress and assess against the critical bed shear stress that initiates sediment motion, which is a function of non-cohesive grain size (Chen & Liew, 2003). However, this approach is only suitable for non-cohesive sediments.

In the case of rock subjected to water forces, whether due to jet impact or high velocity flow, the rock must be evaluated for erosion assessment. For non-jet flow, the computed shear stress can be used to determine erosive capacity (stream power) and the material erosive resistance (Shields and Erodibility Index methods) as given by Annandale (2006). This type of assessment was performed for the unlined stilling basin walls, channelised flow at the toe of dam and overland flow along the abutment for the Hinze Dam Stage 3 upgrade project (Phillips & Riddette, 2007). For overland flow, it was found that it was difficult to accurately model velocity due to a combination of high floor roughness and shallow flow depth.

For the case where jet scour assessment at the downstream dam wall is required for flood waters overtopping a dam crest, an extensive investigation of the jet stream power was performed by the authors. A typical free-falling jet or cascade will accelerate under gravity, which by continuity results in a contraction of the jet thickness. The jet will also spread laterally due to turbulence and aeration at the exposed edges. The stream power per unit area or stream power density is given by:

$$p_{jet} = \frac{\gamma q z}{B} \qquad (6)$$

where p_{jet} = streampower on a horizontal plane at elevation z; γ = unit weight of water; q = unit discharge per metre width; z = vertical distance travelled below reservoir level; and B = horizontal jet width at elevation z.

Ervine & Falvey (1987) proposed that these turbulence effects also travel into the core of the jet, eventually leading to break-up of the core after some distance of fall. The single-fluid CFD model computes the gravitation effects on the free-falling jet, typical of the jet core, which takes into account the initial velocity distribution at the top of the fall, however, it does not currently allow for turbulent jet spreading or break-up of the core. An example of this is shown in figure 24. Despite this, the extent of jet spreading and core break-up can be assessed by applying the computed flow values to empirical formulas such as those presented in Frizell (2006a; 2006b).

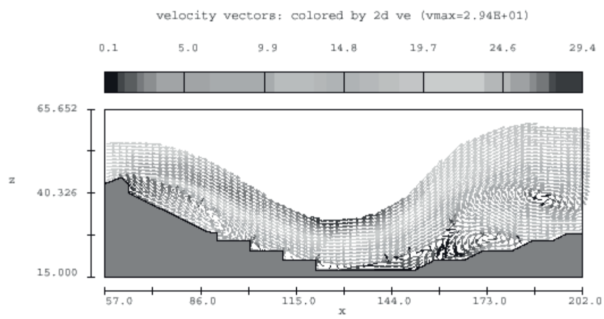

Figure 23: Velocity vectors in a plunge pool showing flow recirculation.

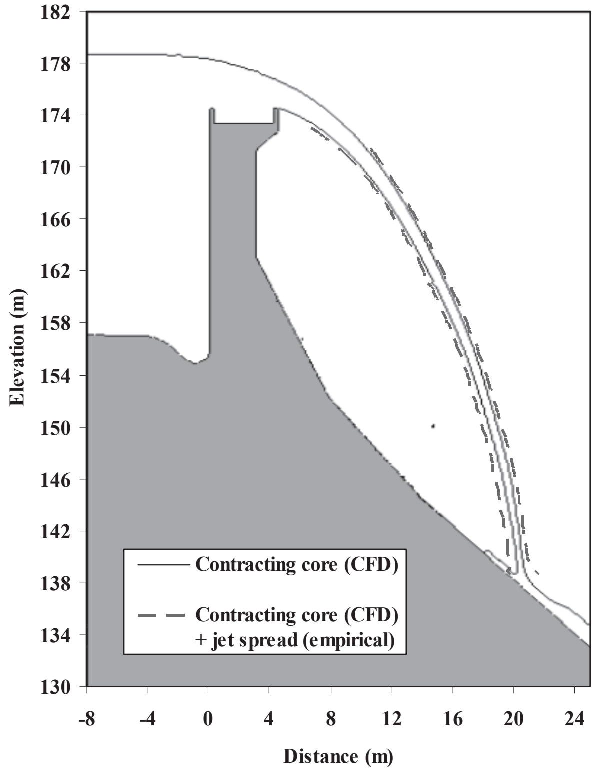

Figure 24: Computed jet width of a free-falling cascade from an overtopped dam crest.

Further research should focus on correctly capturing the spreading and break-up of a free-falling jet. The inclusion of air in the model allowing for aeration, selecting the appropriate turbulence model and mesh resolution are a few suggested aspects to address.

6 DISCUSSIONS AND FUTURE CHALLENGES

6.1 Modelling reliability

From the experience gained from the 16 spillway upgrade projects and other validation exercises, it will be useful to begin the discussion by examining the reliability indices for hydraulic modelling from the "Computational Procedures For Dam Engineering Reliability and Applicability" bulletin published by the International Committee On Large Dams (ICOLD, 2001). The recommendations are reproduced in table 4. The reliability indices (RIs) in relation to using numerical models according to ICOLD are:

- RI #1 – the phenomena related to dam safety can be confidently analysed by means of numerical models.
- RI #2 – the phenomena related to dam safety can be analysed by means of numerical models but with some limitations and/or difficulties (simplifications in the computational hypotheses, lack or difficulty to get fully reliable experimental data, cost of the analyses, etc.).
- RI #3 – the phenomena related to dam safety can be analysed by means of numerical models whose results can give only qualitative or comparative indications, eg. because of strong simplifications needed, etc.
- RI #4 – the phenomena related to dam safety cannot at present be analysed by means of numerical models.

The ICOLD bulletin proposes an RI number for the six dam safety hydraulic behaviours that are discussed below.

6.1.1 Cavitation (current RI number = 2)

The potential for cavitation to occur over a spillway can certainly be computed using CFD. The pressure and velocity in the area in question can be accurately predicted allowing the cavitation potential to be determined. For design purposes, cavitation should be avoided. Currently, computational methods may not be adequate to study the cavitation behaviour in detail, for example, the formation of vapour bubbles and their collapse. It should be noted that there are only a few documented cases where cavitation actually occurred (Falvey, 1990). To further study this phenomenon in a laboratory will involve running a physical model inside a specialised vacuum chamber, which can be quite complicated and costly. Until there is enough good quality measurement, any attempt to simulate cavitation computationally will purely be an academic exercise. Assuming the design philosophy is to prevent cavitation from happening under a certain flood event, there may be no need to consider what may happen once cavitation occurs. Under this condition, the RI may be upgraded to 1.

6.1.2 Erosion

6.1.2.1 Flow velocities on dam face and floating debris impact (current RI number = 3)

One aspect of the erosion assessment depends on the accurate prediction of flow velocity. CFD modelling can predict reliable velocities for uniform flow provided the model is properly prepared and analysed. For non-uniform flow, the computed shear stress at the floor can be used for erosion assessments. As such, from the velocity point of view in a non-jet erosion assessment, the RI number may be upgraded to 2.

In the case of floating debris, although floating bodies can be simulated in the analysis, the actual impact between the debris and spillway crest structure such as gates can be much more complex. The shape and the mass of the debris have to be predetermined, and the coefficient of restitution can lie anywhere between perfectly elastic and perfectly plastic. For preliminary assessment, the computed velocities in the vicinity of the structure in question can be used to compute the kinetic energy and momentum prior to the impact. The appropriate assumptions can be made for the other parameters to evaluate the potential damage to the impacted structure.

6.1.2.2 Downstream heel erosion (current RI number = 4)

For jet erosion there is still uncertainty in regard to modelling the jet width for a free fall. For overland flow, where shallow depths and high roughness are encountered, the computed velocities may be less reliable. For these cases, the RI number may still be 4.

In regard to scour modelling, some studies are showing promise with mixed successes. The authors

Table 4: Summary of reliability of numerical models for hydraulic modelling (ICOLD, 2001).

Phenomena	Construction	First filling	Operation	Reliability Index
Cavitation (pressure)	✓	✓	✓	2
Erosion (velocities & solid material content)	✓	✓	✓	3
Dislocation of paving slabs of spillways (due to oscillation under pressure)	NA	✓	✓	4
Extreme flood (discharge)	✓	✓	✓	1
Downstream heel erosion (discharge & kinetic energy)	NA	✓	✓	4
Siltation (solid transport)	NA	NA	✓	2 to 3

believe further research and validation is required for it to be reliable.

6.1.3 Dislocation of paving slabs of spillways due to oscillating pressure (current RI number = 4)

For steady-state condition, the velocity and pressure distributions over slabs can be predicted with confidence from a CFD model. The transient behaviour may be modelled using an explicit solver. The chaotic nature of a flow discharge cannot be modelled directly, but flow fluctuations can be simulated through the inherent non-linearity in the Navier-Stoke equation together with modelled geometric asymmetries and imperfections, and selection of a suitable turbulence model. This may bring the RI number to 3, but the real challenge will be to simulate the dynamic pressure inside joints between slabs and below the slabs, which involves capturing the hydraulic behaviour in a very minute region several orders of magnitude smaller than the overall spillway dimensions. This requires further research and validation.

6.1.4 Extreme flood (current RI number = 1)

The ICOLD bulletin refers this to the hydrological evaluation of the extreme floods. CFD modelling would rarely be the most appropriate tool for hydrological problems, with the exception of determining the discharge coefficient of the spillway under an extreme flood discharge (including the interaction of two or more adjacent spillways), or the rating curve for a range of flood levels.

Under high discharge cases, the CFD model has been found to predict slightly higher discharges than those obtained from physical models, but in general it is sufficiently accurate for practical purposes. The model can correctly capture the different flow regimes as the upstream head rises. The over-prediction may be due to the presence of negative pressure occurring numerically over the spillway crest or on the downstream face of a weir, which can be unrealistic. For a smooth crest (eg. ogee), this negative pressure can be generated by the discretisation process that creates piece-wise representation of a smooth crest, especially when a coarse mesh is used. A convex edge is formed between adjacent grids and thus a negative (or a lower positive) pressure will be computed. A finer mesh will probably alleviate this discretisation problem. In the case for a non-smooth crest (eg. a sharp crested weir or abutment overtopping), the introduction of air or using a negative pressure cap in the model will allow the lower nappe to flow free from the weir downstream face. However, for small discharge events the lower nappe can stick to the downstream face of the weir as observed in reality.

Therefore, in the context of evaluating the discharge coefficient of spillway for extreme floods, the RI number of 1 is appropriate. However, the authors suggest a RI number of 2 for sharp crested weirs for low to moderate floods.

6.1.5 Siltation – solid transport (current RI number = 2 to 3)

The authors have not carried out any sediment transport modelling from the catchment to the reservoir, nor in the river downstream of spillway, using CFD because it is not usually considered for spillway hydraulics. Therefore, it is outside the topic of this paper.

6.2 Limitations

Despite the benefits offered by the CFD technology, there are currently a number of limitations that it cannot reliably perform for spillway applications. Table 5 is a list of limitations based on the experience obtained from performing the spillway upgrade projects. Some possible aspects for future research to resolve these limitations are also suggested.

6.3 Design tool

The validation exercises and the successful application to the numerous spillway upgrade projects demonstrated that this new modelling technology is a viable computer-aided engineering tool. By understanding its limitations and strengths, engineers can use it with the appropriate level of confidence for spillway design.

Phillips & Riddette (2007) described the use of CFD modelling in conjunction with a physical model during various stages of the design process for the Hinze Dam Stage 3 upgrade works. CFD modelling played a key role in the rapid assessment of initial spillway concepts, was used to validate the approach flow velocities in the physical model, and provided detailed design data for input to structural and erosion calculations.

The authors have also performed concept development of a seawater return energy dissipation outfall structure for a proposed upgrade of a petrochemical processing facility and an outlet pipe for a desalination plant using this modelling tool. The high-speed computation allowed different outfall structures to be analysed, and the hydraulic performance checked against the design criteria quite rapidly. The preferred concept was then further analysed and the geometry/configuration was fine-tuned for different discharge and tailwater conditions in the detailed design stage. It is interesting to note that physical model was not required by the client or the consultants for this project.

7 CONCLUSIONS

This paper gives an introductory insight into the current state of the practice in CFD modelling of

Table 5: A list of current limitations and suggestions for future research.

Item	Limitation	Progress to date[a]	Suggested further work
1	Cavitation	Successful computation of pressure for a range of crest shapes. Cavitation potential determined from empirical equations.	Direct modelling of cavitation for investigative purposes. Forensic CFD investigation of cavitation damage events to better understand the conditions leading to damages.
2	Air-entrainment effect	Successful validation at aeration inception point for limited number of flow conditions.	Validation of air quantity entrained and downstream bulking effects for surface entrainment, and side/bottom devices for entraining air into chute flows.
3	Scour modelling	Current published findings report some limited success using discretised empirical sediment transport equations.	Development in software algorithm by vendors
4	Air demand	Not attempted by authors	Validation of air demand along chutes and behind free falling jet.
5	Overland flow – shallow depth and high roughness	Qualitative agreement for velocity computation. Difficult to extract shear stress due to conflict between high roughness and small mesh size required for shallow flows (different length scales issue)	Parametric study to identify limits of reliability for a combination of floor roughness and flow depth.
6	Thin jets and break up of jets	Limited progress with single-fluid model at jet core only. Requires a very fine grid resolution	Extend to two-fluid model to include air entrainment and jet break-up. Validate impact pressures against experimental data.
7	Fluctuating pressures at spillway/apron floors	Observed results from RANS turbulence model with relatively coarse grid. No validation to date.	Obtain reliable experimental data for validation. Parametric study with grid size and turbulence models to identify level of reliability of computed pressure.
8	Dynamic interaction	Vortex formation has been qualitatively captured. No validation of vortex strength to date.	Obtain reliable experimental/prototype data for validation. Parametric study with mesh size and flow models
9	Long runtimes for complex models	Parallelised software used to run analysis on a multi-CPU computer. Hardware and software cost implications for cluster-type analysis.	Software vendors to develop more efficient software, and hardware suppliers to develop faster computers

[a] From literature review and authors' experience.

spillways in Australia. This pioneering technology has successfully been applied to a number of spillway upgrade projects. It has become a viable design tool for the understanding of hydraulic behaviour in spillway performance.

In view of the overwhelming positive evidence from the validation exercises and real-world projects utilising CFD models, some of the reliability issues regarding the use of numerical hydraulic modelling for dam engineering suggested by ICOLD may need to be reassessed and revised. A number of limitations have also been identified for future research and validation focus.

It is inevitable that computer technology will continue to grow and improve in the future. The role of conducting scaled physical model testing will need to be reappraised. Although physical models can still provide valuable information, it is anticipated that CFD models will be routinely used for concept study. When the preferred option is selected, the physical model may serve to confirm design expectation. This computer-aided rapid prototyping approach is already a common practice in the aerospace and automotive industries.

Just as finite element analyses are now routinely used to compute stresses and deformations of structures under static and seismic loads instead of performing photoelastic analysis on gelatine models and using shaker tables, CFD models undertaken with prudent engineering experience and judgement may in the future replace the need for physical model testing.

ACKNOWLEDGEMENT

The authors would like to thank WorleyParsons for permission to publish some of the analysis results in this paper.

REFERENCES

Abbott, M. B. & Basco, D. R. 1989, *Computational fluid dynamics: An introduction for engineers*, John Wiley and Sons, New York.

American Institute of Aeronautics and Astronautics (AIAA), 1998, *Guide for the verification and validation of computational fluid dynamics simulations*, AIAA G-077-1998.

American Society of Mechanical Engineers (ASME), 2006, *Guide for verification and validation in computational solid mechanics*.

Annandale, G. W. 2006, *Scour technology*, 1st edition, McGraw-Hill, New York.

Barton, A. F. 2003, "A numerical model for the hydraulics of vertical slot fishways", MEngSc (research) thesis, Department of Civil Engineering, Monash University.

Brady, P. 2003a, "3-d unsteady modelling of sewer overflows using computational fluid dynamics", BEng (honours) thesis, University of Technology, Sydney.

Brady, P. 2003b, "An investigation of free surface hydraulic structures using computational fluid dynamics", Doctoral Assessment Report, University of Technology, Sydney.

Cederstrom, M., Hammar, L., Johansson, N. & Yang, J. 2000, "Modelling of spillway discharge capacity with computational fluid dynamics (CFD)", *Proceedings of the 20th International Congress*, International Committee On Large Dams, Beijing.

Chen, W. F. & Liew, J. Y. R. 2003, *Civil engineering handbook*, 2nd edition, CRC Press.

Cole, B. (editor), 2000, *Dam Technology in Australia 1850 – 1999*, Australian National Committee On Large Dams, Australia.

Edwards, J. L. 2006, "The analysis of spillways using computational fluid dynamics", BEng thesis, University of Sydney.

Ervine, D. A. & Falvey, H. T. 1987, "Behaviour of turbulent water jets in the atmosphere and in plunge pools", *Proceedings Institution of Civil Engineers*, Pt. 2, Vol. 83, pp. 295-314.

Falvey, H. T. 1990, *Cavitation in chutes and spillways*, Engineering Monograph No. 42, US Department of the Interior, Bureau of Reclamation, Denver, Colorado.

Flow Science, Inc., 2008, *FLOW-3D User Manual Version 9.3*.

Frizell, K. 2006a, "Hydraulic investigations of the erosion potential of flows overtopping Gibson Dam Sun River Project, Montana Great Plains Region", USBR Water Resources Research Laboratory, Denver Colorado.

Frizell, K. 2006b, "Hydraulic investigations of the erosion potential of flows overtopping Owyhee Dam – Owyhee Project, Oregon-Idaho Pacific Northwest Region", USBR Water Resources Research Laboratory, Denver Colorado.

Gessler, D. 2005, "CFD Modeling of Spillway Performance", *Proceedings of the World Water and Environmental Resources Congress 2005*, American Society of Civil Engineers, Anchorage, Alaska.

Green, J. H. & Meighen, J. 2006, "PMP estimates – are we kidding ourselves?", *Proceedings of the 46th ANCOLD Conference*, Australian National Committee On Large Dams, Manly.

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.

Hirt, C. W. 2003, "Modelling turbulent entrainment of air at a free surface", Technical Note 61, Flow Science, Inc.

Hirt, C. W. & Nichols, B. D. 1981, "Volume of fluid (VOF) method for the dynamics of free boundaries", *Journal of Computational Physics*, Vol. 39, pp. 201-225.

Hirt, C. W. & Sicilian, J. M. 1985, "A porosity technique for the definition of obstacles in rectangular cell meshes", *Proceedings of the 4th International Conference of Ship Hydrodynamics*, National Academy of Science, Washington, DC.

Ho, D. K. H., Boyes, K. M., Donohoo, S. M. & Cooper, B. W. 2003, "Numerical flow analysis for spillways", *Proceedings of the 43rd ANCOLD Conference*, Australian National Committee On Large Dams, Hobart.

Ho, D. K. H., Riddette, K. M. & Donohoo, S. M. 2004, "Analysis of spillway flow by computational fluid dynamics technique", *Proceedings of the 8th National Conference on Hydraulics in Water Engineering*, The Institution of Engineers, Australia, Gold Coast.

Ho, D. K. H., Riddette, K. M., Hogg, M. C., Sinha, J. & Roberts, J. 2005, "Blowering Dam – spillway hydraulic modelling", *Proceedings in the 45th ANCOLD Conference*, Australian National Committee On Large Dams, Fremantle.

Hurst, P., Ewing, T., Fox, S. & Wark, B. 2007, "Calculating spillway crest pressures for high-head operation", *Proceedings of the 2007 NZSOLD/ANCOLD Conference*, Queenstown, New Zealand.

International Committee On Large Dams (ICOLD), 2001, "Computational procedures for dam engineering reliability and applicability", Bulletin 122.

Kenny, R. J. 2007, "Computational fluid dynamics modelling of an uncontrolled ogee spillway crest using Fluent software", MEng Thesis, Queensland University of Technology.

Kjellesvig, H. M. 1996, "Numerical modelling of flow over a spillway", Hydroinformatics'96, Balkema, Rotterdam.

Lesleighter, E., McPherson, B., Riddette, K. & Williams, J. 2008, "Modelling procedures used for the spillway upgrade for Lake Manchester Dam", *Proceedings of the 2008 ANCOLD Conference*, Australian National Committee On Large Dams, Gold Coast.

Maynord, S. T. 1985, "General spillway investigation hydraulic model investigation", Technical report HL-85-1, US Army Engineer Waterways Experiment Station, Vicksburg, Miss.

Phillips, M. A. & Riddette, K. M. 2007, "Computational fluid dynamics and physical hydraulic modelling – do we need both? The design of the Hinze Dam Stage 3", *Proceedings of the 2007 NZSOLD/ANCOLD Conference*, Queenstown, New Zealand.

Riddette, K. M., Ho, D. K. H. & Edwards, J. L. 2006, "Hydraulic flow simulation – five years on: lessons learned and future challenges", *Proceedings of the 46th ANCOLD Conference*, Australian National Committee On Large Dams, Manly.

Riddette, K. M., Ho, D. K. H. & Phillips, M. A. 2008, "Modelling extreme flows over a stepped spillway", *Proceedings of the 2008 ANCOLD Conference*, Australian National Committee On Large Dams, Gold Coast.

Rodi, W. 1980, *Turbulence models and their application in hydraulics – A state of the art review*, 2nd edition, International Association of Hydraulic Research.

Savage, B. M. & Johnson, M. C. 2001, "Flow over ogee spillway: physical and numerical model case study", *Journal of Hydraulic Engineering*, ASCE, Vol. 127, No. 8, pp. 640-649.

Souders, D. T. & Hirt, C. W. 2003, "Modeling roughness effects on open channel flows", Technical Note 60, Flow Science, Inc.

Stanford University, 2008, *National Performance of Dams Program*, Department of Civil and Environmental Engineering, http://npdp.stanford.edu, viewed December 2008.

Teklemariam, E., Korbaylo, B. W., Groeneveld, J. L. & Fuchs, D. M. 2002, "Computational fluid dynamics: diverse applications in hydropower project's design and analysis", *Proceedings of the 55th Annual Canadian Water Resources Association Conference*, Winnipeg.

US Army Corps of Engineers (USACE), 1952, "Corps of Engineers hydraulic design criteria", Waterways Experiment Station, revised in subsequent years.

US Army Corps of Engineers (USACE), 1994, "Hydraulic design of flood control channels", EM 1110-2-1601.

US Army Corps of Engineers (USACE), 1995, "Hydraulic design of spillways", *Technical Engineering and Design Guides as adapted from the US Army Corps of Engineers*, No. 12, ASCE.

US Bureau of Reclamation (USBR), 1982, "Hydraulic model study of Ute Dam labyrinth spillway", Report GR-8-75, US Department of the Interior, Bureau of Reclamation, Denver, Colorado.

US Bureau of Reclamation (USBR), 1991, "Morning-glory spillway model Beaver Run Dam Pennsylvania", Report R-91-03, US Department of the Interior, Bureau of Reclamation, Denver, Colorado.

Versteeg, H. K. & Malalasekera, W. 1995, *An introduction to computational fluid dynamics the finite volume method*, Prentice Hall.

Wilcox, D. C. 1993, *Turbulence modeling for CFD*, DCW Industries, Inc.

Yang, J. & Johansson, N. 1998, "Determination of spillway discharge capacity – CFD modelling and experiment verification", *Proceedings of the 3rd International Conference on Advances in Hydro-science and Engineering*, Cottbus, Germany.

Zerihun, Y. T. 2004, "A one-dimensional Boussinesq-type momentum model for steady rapidly varied open channel flows", PhD thesis, University of Melbourne.

Zipparo, V. J. & Hasen, H. 1993, *Davis' handbook of applied hydraulics*, 4th edition, McGraw-Hill, New York.

DAVID HO

David Ho was awarded a Bachelor of Science (1st class honours) in Civil Engineering from the University of Manchester, UK, in 1985. At the same university, he carried out research work developing a non-linear three-dimensional elasto-plastic finite element program to study the simulation of braced excavations and construction of reinforced earth walls, and received his Doctor of Philosophy Degree in 1998. He has worked in consulting engineering companies Sir Alexander Gibbs & Partners, UK, and LongMac Associates, Australia, specialising in numerical analysis for civil and geotechnical engineering projects. Subsequently, he worked for Compumod in Sydney performing consulting projects using computer-aided-engineering tools for a diverse range of engineering applications. During this time, he was a part-time lecturer teaching finite element analysis at the University of Technology Sydney. Currently, he is the Technical Manager of the Advanced Analysis group in WorleyParsons, managing a range of numerical simulations including hydraulic structures, ground-structure and fluid-structure interactions. He is a chartered professional engineer in Australia, and a registered (advanced) analyst of the National Agency For Finite Element Methods and Standards (NAFEMS), UK.

KAREN RIDDETTE

Karen Riddette graduated from the University of Technology Sydney in 1999 with a Bachelor of Engineering (1st class honours) in Civil and Environmental Engineering. Since then she has worked for consulting engineering company WorleyParsons in the Advanced Analysis group. During this time Karen has pioneered the use of computational fluid dynamics for dam spillway modelling in Australia, carrying out numerous numerical analyses and validation studies for a range of hydraulic structures and physical phenomena. In her present role as Senior Engineer/Analyst, Karen's recent projects have focused on modelling the effects of extreme flood discharge on the existing and proposed spillway facilities of large dams throughout Australia, and hydraulic performance of fishways, seawater intake pumping stations, cooling water outfall and large water tanks. She is a member of Engineers Australia.