COMMISSION INTERNATIONALE
DES GRANDS BARRAGES

VINGT-QUATRIÈME CONGRÈS
DES GRANDS BARRAGES
Kyoto, Juin 2012

COMPLEMENTARY USE OF PHYSICAL AND NUMERICAL MODELLING TECHNIQUES IN SPILLWAY DESIGN REFINEMENT *

James WILLEY
GHD Pty Ltd, Brisbane, QLD,

Thomas EWING
GHD Pty Ltd, Melbourne, VIC,

Bob WARK
GHD Pty Ltd, Perth, WA,

Eric LESLEIGHTER
Lesleighter Consulting Pty Ltd, Cooma, NSW,

AUSTRALIA

1. INTRODUCTION

The design of recent projects involving new or upgraded spillways have benefited from the complementary use of both Computational Fluid Dynamics (CFD) techniques and physical scale modelling. As well as providing significant benefits to the design process for these projects, this parallel use has provided invaluable data for the comparison of the techniques which may lead to greater confidence in the future use of standalone CFD analyses in spillway design.

The design process recently adopted for complex spillway designs involved the initial development of a concept or preliminary design using theoretical and empirical design methods, together with published data. These arrangements

* *Utilisation complémentaire des techniques de modélisation numérique et physique pour l'amélioration de la conception des déversoirs.*

were then analysed and optimised through CFD modelling and, where required, physical scale modelling was used for final verification and refinement.

A comparison of the results from CFD and physical models is presented where both have been used in the design process and conclusions are drawn on the applicability of standalone CFD analyses. Projects on which GHD was the designer and which are discussed herein include the spillway for the new Enlarged Cotter Dam and remedial works at Lake Manchester, Blue Rock and Wellington Dams.

2. BACKGROUND

Physical scale modelling has been used in the design and investigation of hydraulic structures for over 100 years. The design process has typically involved the development of a preliminary design on the basis of theoretical and empirical methods. A physical scale model of this arrangement would then be constructed in two- or three-dimensions and various scenarios run to confirm whether the hydraulic performance was acceptable and to extract data for input to the design. The methods are tried and tested and the outputs from the model testing in terms of data and observations are invaluable in the design process. However, the construction, operation, and testing of physical models is often a time-consuming and expensive exercise. Furthermore, the modification of the model to trial alternative arrangements or to optimise features can add weeks to a testing programme.

Through recent advances in computing power and modelling software capabilities, it is now feasible to undertake complex three-dimensional analyses using CFD techniques. To date, CFD modelling has generally been used as a valuable tool in the optimisation phase of the project prior to the commissioning of a physical model study. The major benefit of the CFD modelling in this capacity was that it allowed the early identification of problematic flow features and modifications to the layout could be trialled rapidly and cost-effectively. Although CFD modelling packages now have the capability to analyse complex hydraulic conditions common in spillways such as air entrainment, flow separation, turbulence and shock waves, there is however a significant lack of calibration and validation studies (between CFD and physical models and also between model and prototype) for these advanced applications and caution should be applied to their use in design.

3. CFD MODELLING CAPABILITIES AND APPLICATION TO ANALYSIS OF SPILLWAYS

The most common, and reliable CFD analysis of spillways involves modelling of flow in the spillway approach, over spillway crests, and around obstacles such as piers and flow training walls. Models of this type are used to generate rating curves, predict water surface elevations and identify zones of unfavourable flow behaviour. Examples of practical and accurate modelling of these flows are presented in this paper, and many more can be found in the published literature.

In these relatively simple cases the dynamics are dominated by what is fundamentally a transfer of potential energy to kinetic energy and flow around abrupt obstacles. Modelling of these processes to a reasonable degree of accuracy does not require accurate simulation of turbulence or boundary layers, allowing the use of coarse model grids and simple turbulence models. By contrast, the dynamics of high speed flow down steep spillways or chutes are dominated by the production and dissipation of turbulence and require careful model and mesh configuration.

Aeration of spillway flows is another process that is very dependent upon the accurate representation of turbulence and boundary layer dynamics. Several CFD codes do have the capability to model interpenetrating phases (in this instance air bubbles in water), but at present only a relatively small number of aerated flows have been modelled successfully using CFD. This was not considered in the examples presented herein as the CFD modelling in these cases was used as an initial optimisation tool using a coarser grid resolution for more rapid processing. Further work is required in this area. Further comments on the limitations of CFD are included in the conclusions.

4. COMPARISON OF CFD AND PHYSICAL MODEL DATA

4.1 ENLARGED COTTER DAM

4.1.1 Brief Description of the Project

The Enlarged Cotter Dam (ECD) is a new 87 m high roller-compacted concrete dam under construction at the time of writing in 2011. The spillway includes a central primary stepped spillway tapering from 70 m at the crest to approximately 45 m at the entrance to the stilling basin. Stepped secondary spillways were provided over each abutment and operate for floods greater than the 1:1000 annual exceedance probability (AEP) discharge. The discharge from the secondary spillways is conveyed to the river channel by stepped abutment return channels or cascades.

The primary spillway has an ogee crest designed for the maximum head, while the secondary spillway crest is horizontal with a downstream transition curve designed for half the maximum head. The primary spillway curve transitions into steps approximately 9 m horizontally from the upstream face. The design for the project utilised both two- and three-dimensional CFD analyses using in-house capabilities and a 1:45 scale three-dimensional physical model by Manly Hydraulics Laboratory (MHL).

4.1.2 Spillway Discharge Rating

The initial rating curve for the project was derived from published data in [1]. Discharge coefficients were derived from the two-dimensional CFD analyses for the primary and secondary spillway crests. From the physical model study, it was only possible to calculate discharge coefficients for the primary spillway crest for discharges up to the 1:1000 AEP event or a unit discharge (q) of about 8 m³/s/m. Beyond this level, the secondary spillways began to operate and it was therefore not possible to calculate discharge coefficients for either crest without making assumptions. The discharge coefficients derived from the various methods for the primary and secondary spillway crests are presented in Fig. 1. Fig. 2 presents a comparison of the discharge rating curves derived from the discharge coefficients plotted in Fig. 1.

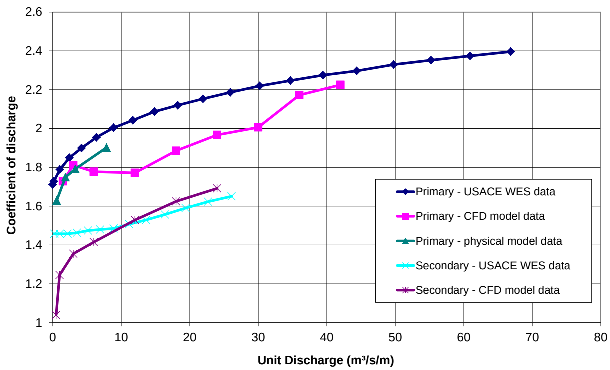


Fig. 1

Relationship between unit discharge and *discharge coefficient for Cotter Dam*

Relation entre le coefficient de débit et le débit unitaire pour le barrage de Cotter

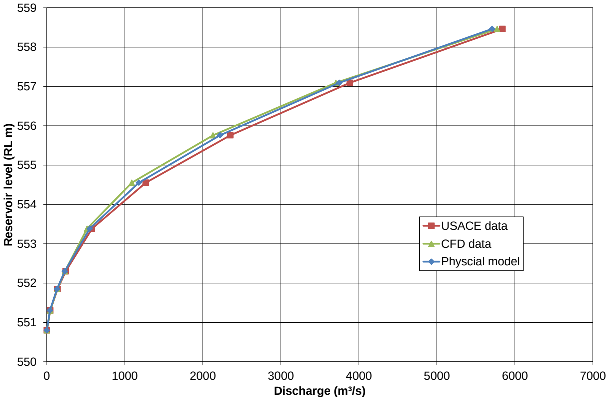


Fig. 2
Comparison of discharge rating curves for Cotter Dam
Comparaison des courbes de tarage pour le barrage de Cotter

For the primary spillway crest, the discharge coefficients calculated from the physical model are within 5% of the USACE values. This was also the case for the coefficients derived from the CFD analyses for $q < 3$ m³/s/m and $q > 36$ m³/s/m. However, over the intermediate range, the calculated coefficients were typically 8-13% less than the USACE values. For $q > 6$ m³/s/m over the secondary spillway, the CFD analyses yielded discharge coefficients within 5% of the USACE values. However, at low discharges, the CFD showed the crest was less efficient than indicated by the USACE data.

Despite the differences in the discharge coefficients noted above, the difference in the total head over the spillway for the probable maximum flood (PMF) discharge of 5,710 m³/s was about 0.5%, with a marginally higher head measured in the physical model. Similarly for the same reservoir level, the difference in discharge estimates was about 1%.

4.1.3 Pressures

This comparison is based on results from the two-dimensional CFD modelling and the three-dimensional physical model, and published data from [1] and [2]. In the physical model, pressures were measured using 145 pressure tappings connected to manometers on eight sections across the the primary and secondary spillways. Fig. 3 presents the calculated CFD and measured physical model pressures over the primary spillway crest for $q = 48$ m³/s/m together with published data from [1]. This represents the scenario with $H/H_d = 1$ (H = actual head on the crest and H_d = design head. This is plotted using the dimensionless parameters h_p/H_d and x/H_d, (h_p = pressure head and x = offset from the ogee curve origin). Reasonable correlation is evident between the three data sets.

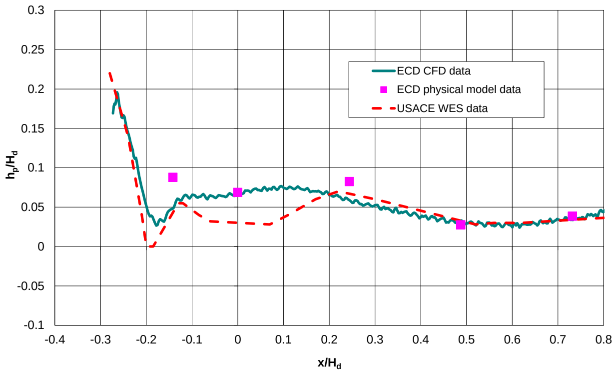
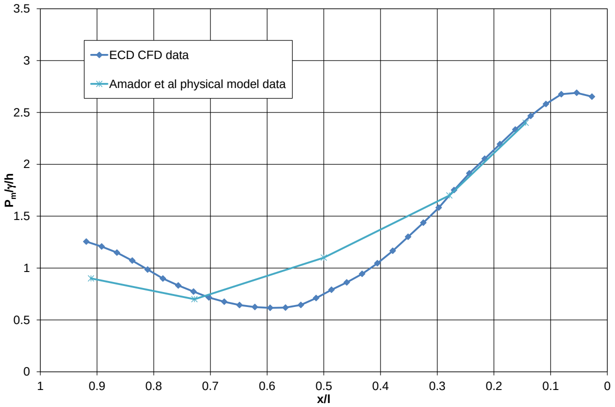


Fig. 3

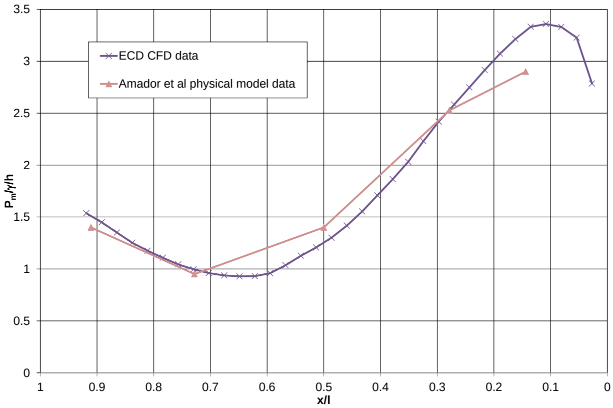
Primary spillway ogee crest pressures for Cotter Dam

Pression sur le principal seuil du déversoir en doucine du barrage de Cotter

The comparison in (b) Fig. 4 draws on results of the ECD CFD data as well as physical model study data presented in [2]. In the Amador et al data [2], pressure profiles on the horizontal and vertical faces of the steps were measured in a two-dimensional stepped channel for a range of discharges corresponding to $0.89 < y_c/h < 3.21$, where y_c is the critical depth and h is the step height. This equates to $3.5 < q < 24$ m³/s/m for the ECD arrangement.



(a)



(b)

Fig. 1
Comparison of pressures on horizontal step surface for L/k_s = 22.64 with
a) y_c/h = 1.4 and b) y_c/h = 2.7
Comparaison des pressions sur la surface horizontale au pied du déversoir pour L/K_s = 22.64 avec a) y_c/h = 1.4 et b) y_c/h = 2.7

The pressure distribution in (b) Fig. 4 is for the horizontal step surface of the step located at L/k_s = 22.64 (with L being the developed length along the chute from the crest and k_s being the roughness height of the steps perpendicular to the slope) and y_c/h values of approximately 1.4 and 2.7. Dimensionless parameters are plotted here, where P_m is the mean pressure and γ is the unit weight of the fluid. The distance, x, is measured in the upstream direction from the downstream edge of the step and is plotted as a proportion of the step length, l.

4.1.4 General Flow Conditions

The flow conditions observed in the physical model were generally well replicated in the CFD analyses. Fig. 5 shows a comparison of the CFD output and the physical model flow conditions for the 1:1 000 000 AEP event. Aside from the specific parameters discussed above, a key aspect which was modelled well was the deflection of the flow in the abutment return channels by the angled steps and the shaped downstream wall.

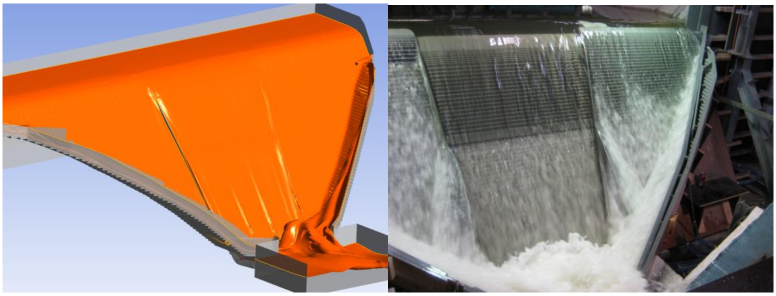


Fig. 5
Overview of Cotter Dam CFD and physical model flow conditions for
1:1 000 000 AEP event (Q = 3750 m³/s)
Aperçu des conditions d'écoulement du barrage de Cotter dans le modèle de MFN et dans le modèle physique pour l'évènement de fréquence annuelle 1 :1 000 000 (Q= 3750 m³/s)

4.2 LAKE MANCHESTER DAM

4.2.1 Brief Description of the Project

Lake Manchester Dam is a 38 m high concrete gravity dam constructed from 1912 to 1916. A dam safety upgrade was required to address identified deficiencies including inadequate spillway capacity, erosion of the spillway and unsatisfactory dam stability. The adopted upgrade solution included two rows of post-tensioned anchors, raising of the non-overflow crest of the dam, construction of a new spillway and upgrade of the outlet works. The new spillway includes widening of the approach channel, a trapezoidal flat-topped crest section to allow future placement of Fusegates and a steep, converging chute discharging into a plunge pool. Construction was completed in 2008.

CFD analyses were undertaken by WorleyParsons during the design phase on a range of geometries to review spillway approach conditions and chute hydraulics for PMF conditions. MHL was subsequently commissioned to construct and test a 1:40 scale physical model of the final arrangement for design validation and refinement over the full range of discharges. The comparison and data herein is largely based on the assessment presented in [3].

4.2.2 Spillway Discharge Rating

Reservoir levels were recorded over a range of spillway discharges in the physical model, allowing calculation of the discharge coefficient for a range of unit discharges. The CFD model was only run for PMF conditions, so only one data point is available for comparison as shown in Fig. 6 There is good agreement with the two methods. The CFD model yielded a discharge coefficient less than 3% higher than the physical model value at an equivalent discharge.

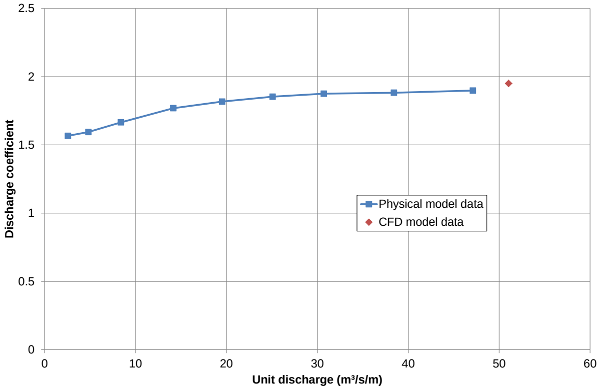


Fig. 6
Comparison of calculated discharge coefficients for Lake Manchester Dam
Comparaison des coefficients de débit calculés pour le barrage du lac Manchester

4.2.3 Pressures

Initial calibration and validation of the CFD model was undertaken using published physical model study data for trapezoidal crest shapes and good correlation was achieved. The CFD model was then applied to the actual three-dimensional geometry of the proposed arrangement. Floor pressures were extracted along the spillway centreline and are plotted in Fig.7 The pressures from the physical model are static pressures measured using pressure tappings connected to manometers. Generally, the correlation is excellent, noting that the CFD results are for a slightly higher flow of 2540 m^3/s compared with 2350 m^3/s in the physical model. This would account for some of the minor differences in the extreme pressures measured at the changes in grade.

4.2.4 Water Surface Profiles

Transverse water surface profiles at the spillway crest for the PMF were measured in the physical model and extracted from the CFD analysis (Fig. 8). Significant superelevation was evident due to approach flow conditions with a pronounced drawdown on the right side due to flow around the right side approach wall structure. In the physical model, significant temporal variability was noted due to “the major effects of flow turbulence interaction with upstream topography and the spillway structure itself” [3]. Generally, the results from the physical model and CFD analysis compare well, with the exception of adjacent to the right side wall.

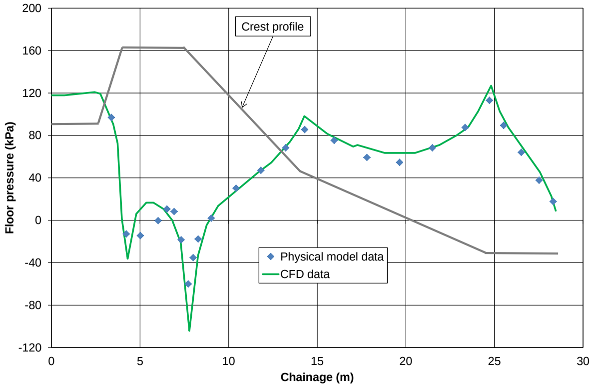


Fig. 7
Comparison of floor pressures from CFD and physical model for Lake Manchester Dam
Comparaison de la pression au sol entre le modèle physique et le modèle de MFN pour le barrage du lac Manchester

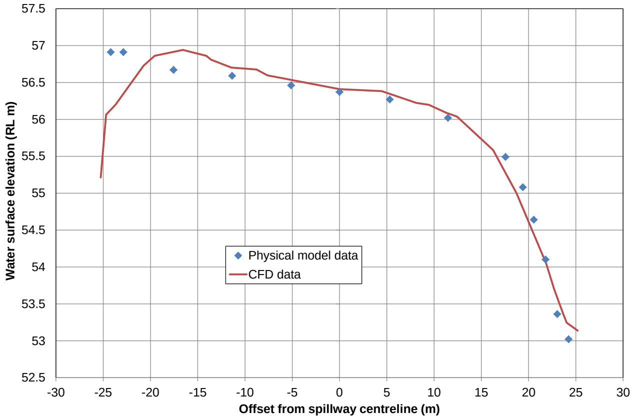


Fig. 8
Comparison of floor pressures from CFD and physical model for Lake Manchester Dam
Comparaison de la pression au sol entre le modèle physique et le modèle de MFN pour le barrage du lac Manchester

4.2.5 Chute Waves

It was found that the CFD modelling was not able to reproduce the large fluctuating waves in the converging chute which were identified in the physical model study. Data on such waves is critical for the design of chute walls in terms of height and the possible use of overhanging sections as wave deflectors.

4.3 BLUE ROCK DAM

4.3.1 Brief Description of the Project

Blue Rock Dam is located on the Tanjil River, around 30 km north of Moe, Victoria, Australia. As part of a project to upgrade the spillway and chute, a CFD study of the existing spillway and chute was undertaken. The hydraulics of the spillway had previously been investigated in a physical scale modelling study and reported in [4]. CFD model validation involved comparison of discharge rates, approach velocities, and water levels in the chute. The domain modelled in the CFD study is shown in Fig. 9.

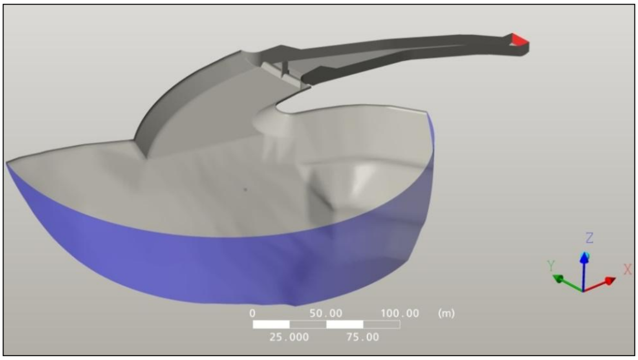


Fig. 9
Blue Rock Dam Spillway CFD Model Domain
Domaine modèle de MFN du déversoir du Barrage de Blue Rock

4.3.2 Spillway Discharge Rating

Fig. 10 compares discharge estimates from the scale model to estimates from the CFD study. The results compare well, with the CFD reservoir head consistently slightly higher than those estimated from the scale model.

The water levels from the CFD model were extracted at the model boundary where the water velocity is very low. As such, the velocity head can be assumed to be negligible and the water levels extracted from the CFD model can be assumed to represent the total head in the reservoir. The location in the scale model where the water level was estimated is not known. It was noted in the CFD study that water levels were drawn down for more than one hundred metres upstream of the spillway crest. This difference in the way that the reservoir level was measured may explain the consistent bias between the CFD and scale model results.

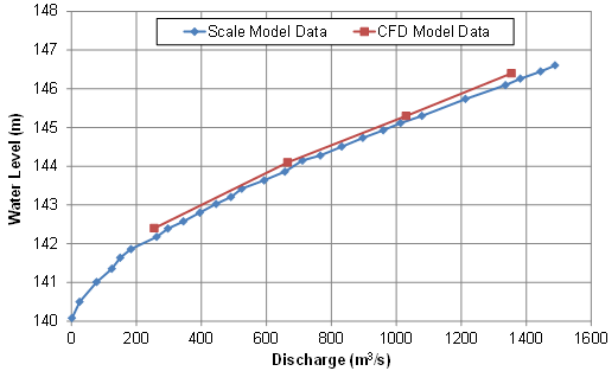
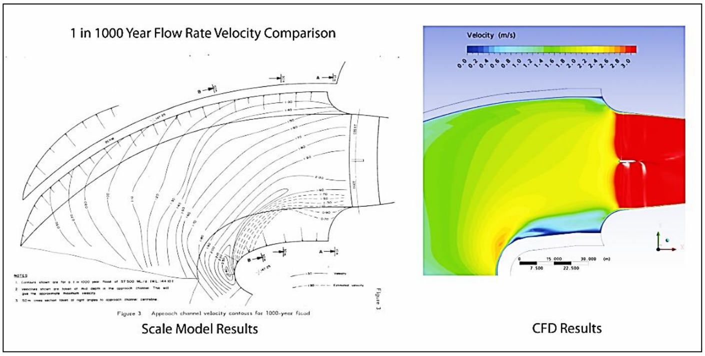


Fig. 10
Blue Rock Dam Spillway Model Discharge Comparison
Comparaison du débit du déversoir du Barrage de Blue Rock

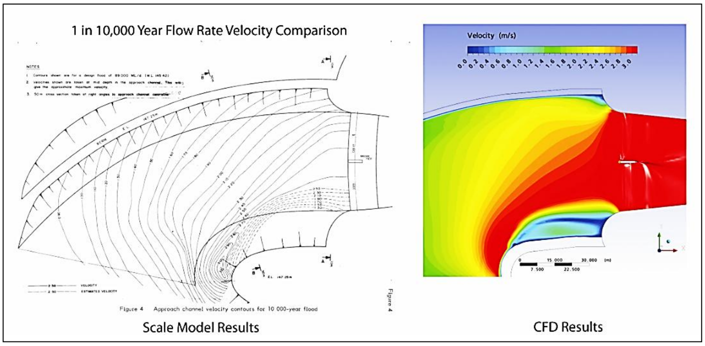
4.3.3 Spillway Approach Flow

The scale model study investigated variability in water velocities in the approach channel in considerable detail. The passage of water around the embankment results in the formation of a recirculating zone near the spillway. Two contour drawings of the velocity distribution were included in the scale modelling report, and these were compared with the output of the CFD model.

In the scale model the velocity data were measured using an impeller based current meter at a variable, intermediate depth in an attempt to provide an estimate of the maximum velocity. The velocity data from the CFD model was extracted on a plane at an intermediate (and constant) depth, and as such the results are not directly comparable. A qualitative comparison is possible however and at both flow rates the CFD model successfully reproduced zones of flow separation, as well as the general trends of velocity in the approach.



(a)



(b)

Fig. 11
Comparison of CFD and Scale Model Approach Velocity, a) 1:1000 AEP discharge and b) 1:10 000 AEP discharge
Comparaison de la vélocité entre le modèle de MFN et le modèle physique pour un débit de fréquence annuelle de a) 1:1000 et b) 1 :10 000

4.3.4 Spillway Chute Water Levels

Cross-sectional water surface profiles were measured at several cross sections (control points) along the length of the chute. Fig. 12 shows that at a low flow rate there is a good agreement between the CFD and scale model results, fluctuations of water level and the formation of standing waves at this flow rate are limited.

At high flow rates, the formation of standing waves is more pronounced, and differences between the CFD and scale model results are evident (Fig. 13). It is likely that these results diverge because of insufficient CFD model resolution in the near wall area along the length of the chute. This coarse resolution does not allow the velocity profile in the chute to be modelled accurately, and this has an impact on the velocity predictions, the effects of roughness, and ultimately the location of standing waves. In the context of this study, this limitation of the constructed CFD model was noted and the results interpreted in the context of this limitation.

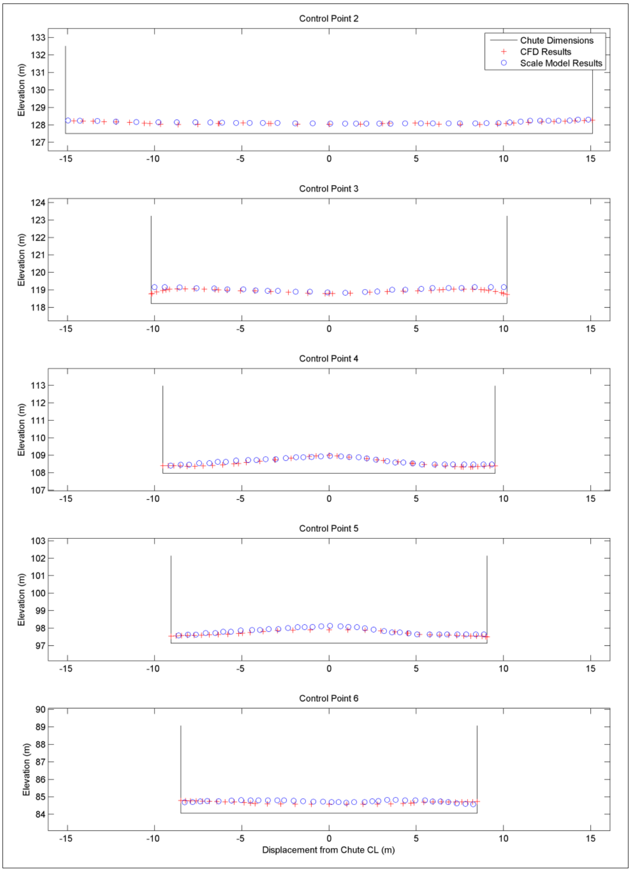


Fig. 12
Comparison of CFD and scale model chute water surface profiles – low flow rate
Comparaison entre le modèle de MFN et le modèle réduit, du profil de la chute d'eau, à petit débit

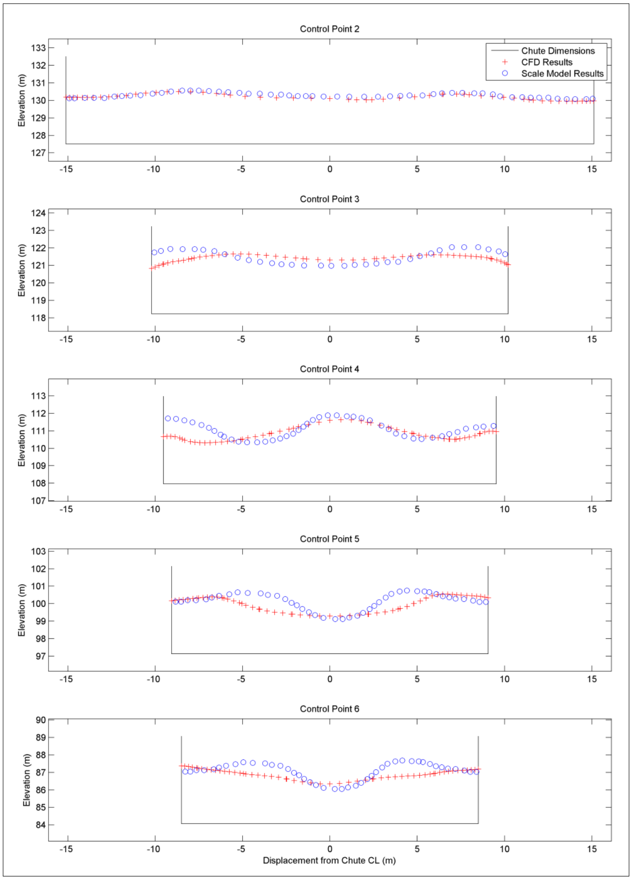


Fig. 13
Comparison of CFD and scale model chute water surface profiles – high flow rate
Comparaison entre le modèle de MFN et le modèle réduit, du profil de la chute d'eau, à large débit

4.4 WELLINGTON DAM

4.4.1 Brief Description of the Project

Wellington Dam in Western Australia was constructed in 1933. The CFD study was undertaken using in-house capabilities as part of a project which aimed to stabilise the spillway by adding piers, a new bridge structure, and post-tensioned anchors. The objectives of the CFD study were to investigate the uplift generated on the spillway crest under high operating levels, and to investigate the impact of the piers upon the discharge capacity of the spillway. The CFD study was undertaken in parallel with a scale modelling study, and the results were compared after both investigations were complete.

4.4.2 Scale Modelling Study

A scale model investigation was undertaken by the Manly Hydraulics Laboratory. The model constructed was at 1:24 scale of a two dimensional section of the spillway, incorporating three piers (Fig. 14). The scale model was evaluated at water levels from 1 m to 6 m (relative to the crest), in 1 m increments.

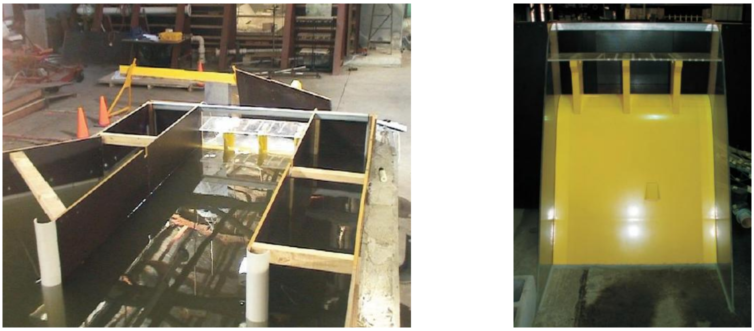


Fig. 14
Photographs of the Wellington Dam Spillway Scale Model
Photographie du modèle réduit du déversoir du barrage de Wellington

4.4.3 CFD Modelling Study

A three dimensional, multiple phase model of a two-dimensional section of the spillway and pier was undertaken. The domain was designed to mimic the design evaluated in the physical modelling study, in order to allow comparisons between the two sets of data. Conveniently, flow between the pylons has two planes of symmetry. The first plane intersects the middle of each pylon while the second intersects at the midpoint between two pylons. Identifying these planes of symmetry allowed the CFD model to be simplified to the zone shown in Fig. 15. The spillway is a repeating set of these units, and the use of symmetry boundary conditions on each side of this domain reduced the size of the domain to a minimum. As with the scale model, six water levels were evaluated.

4.4.4 Spillway Discharge Rating Curve Comparison

The discharge results from the two models were converted into flow per 7.62 m section and compared (Fig. 16). This corresponds to the flow between one set of pier structures (one 'bay'). There is a very close agreement between the CFD and scale model, the difference being a maximum of 3.0% (at 1 m surcharge level), and a minimum of 0.1% (at a surcharge level of 4 m).

4.4.5 Crest Pressure Comparison

Pressure was measured on the scale model using piezometers installed in lines parallel to the flow. Line J was located close to a pier (480 mm prototype offset), Line K was further from the pier (960 mm prototype offset), while Line G was located at the midpoint between the piers. Pressure data were extracted from the CFD model along the same lines and compared for the 6 m surcharge level (Fig. 17). The results of the CFD and scale modelling compare well and show extensive regions of sub-atmospheric pressure at the crest. Similar agreement between the models was evident for the other water surcharge levels. As well as comparing well to each other, the results compared well with previous experimental results [5]. The CFD model results allowed for the total uplift on the crest element to be calculated, a result that was used in the design of the post-tensioned anchors.

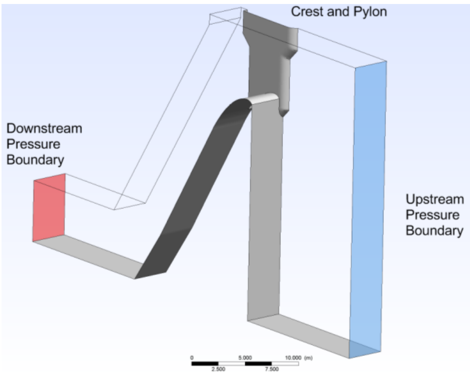


Fig. 15
Diagram of the Wellington Dam Spillway CFD Model
Diagramme du modèle de MFN du déversoir du barrage de Wellington

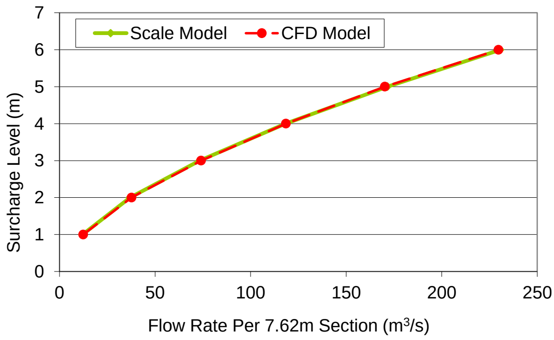


Fig. 16
Wellington Dam Spillway Discharge Comparison
Comparaison du débit du déversoir du barrage de Wellington

4.4.6 Water Level Comparison

As with the pressure, the water level was measured in the scale model along lines parallel to the flow. The CFD and scale model water levels were compared along Lines K and G. Fig. 18 shows a comparison undertaken for the 6 m surcharge level. There is a close agreement between the two models and this was also observed at the other water levels.

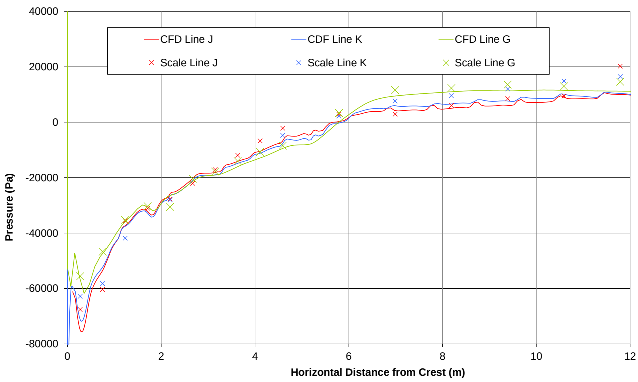


Fig. 17
Wellington Dam Spillway Crest Pressure Comparison
Comparaison de la pression au seuil du déversoir du barrage de Wellington

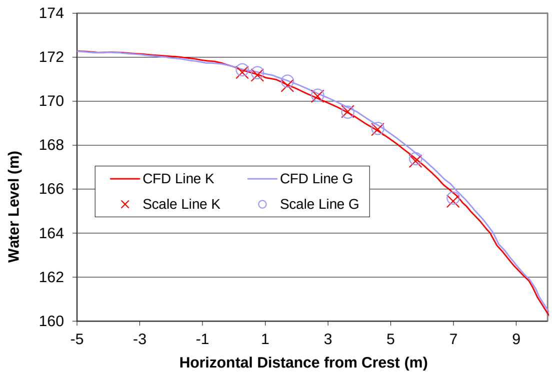


Fig. 18
Wellington Dam Spillway Crest Water Level Comparison
Comparaison du profil de la surface de l'eau pour le déversoir du barrage de Wellington

5. CONCLUSIONS

CFD modelling has been shown in the applications summarised herein to be an effective modelling tool for spillway design applications. This assessment was however limited to consideration of static conditions. Comparisons with scale modelling data show that excellent results can be produced, especially in areas where the velocity is relatively low, for example the spillway approach and crest region. For studies of these regions, CFD models using relatively coarse grids can produce accurate results very rapidly and allow for design alternatives and multiple upstream water levels or discharges to be evaluated and optimised rapidly.

It is critical to note that further work is required to improve CFD modelling techniques in relation to transient pressures, turbulence, cavitation, effects of air, two-phase flows, structure vibration and pressure pulsations in hydraulic jumps and on structure boundaries.

Use of CFD in optimisation encourages experimentation and innovation, and may allow higher performance or lower costs solutions to be found. Once a preferred design is determined, a physical model can be constructed and tested in order to increase confidence in the design, as well as investigate phenomena that are difficult to test in a CFD model such as the points noted above.

While it may be tempting from a cost and project delivery point of view to abandon scale model testing entirely, there are clearly circumstances where practical CFD modelling fails to deliver accurate results. These include high Reynolds number flows down spillways and chutes, flow over aerator structures, aerated flows, and hydraulic jumps. In some cases, these flow conditions may also be challenging for practical scale models. An engineer, with access to both sets of results (CFD and scale modelling), can make informed judgements, based on the advantages and disadvantages of each technique.

Complementary use of CFD and scale models allows for a degree of cross checking between the two sets of model results, allows for innovation and refinement of designs, provides greater confidence in the final design and will inevitably lead to improvements in both CFD modelling methodologies and scale modelling practices.

ACKNOWLEDGEMENTS

The authors would like to thank GHD for the support provided in the preparation of this paper. They would also like to thank the following organisations for their permission to use data from their projects in the assessment presented herein: Bulk Water Alliance and ACTEW Corporation for the Enlarged Cotter Dam, Seqwater for Lake Manchester Dam, Southern Rural Water for Blue Rock Dam and Water Corporation Western Australia for Wellington Dam.

REFERENCES

[1] UNITED STATES ARMY CORPS OF ENGINEERS. *Hydraulic Design Criteria*. 1987.

[2] AMADOR A., SANCHEZ-JUNY M., DOLZ J. Developing flow region and pressure fluctuations on steeply sloping stepped spillways. *ASCE J. Hyd. Eng.*, Vol 135, No 12, pp 1092-1100. 2009.

[3] LESLEIGHTER E., McPHERSON B., RIDETTE K., WILLIAMS J. Modelling procedures used for the spillway upgrade for Lake Manchester Dam. *ANCOLD Conference: Dams and Water for the Future*. 2008.

[4] STATE RIVERS AND WATER SUPPLY COMMISSION MAJOR PROJECTS DESIGN DIVISION. *Blue Rock Dam – Concrete Lined Chute Spillway Hydraulic Model Studies*. 1983.

[5] HURST P.C., EWING T.C., FOX S., WARK, R. Calculating spillway crest pressures for high-head operation. *NZCOLD/ANCOLD 2007 Conference*. 2007.

SUMMARY

The design of recent projects involving new or upgraded spillways have benefited from the complementary use of both Computational Fluid Dynamics (CFD) techniques and physical scale modelling. The design process recently adopted for complex spillway designs or upgrades involved the initial development of a preliminary design using theoretical and empirical design methods together with published data. These arrangements were then analysed and optimised through CFD modelling and, where required, physical scale modelling was used for final verification and refinement.

A comparison of the results from CFD and physical models is presented where both have been used in the design process and conclusions are drawn on the applicability of standalone CFD analyses.

CFD modelling has been shown in the summarised applications to be an effective modelling tool for spillway design applications; however this assessment was limited to consideration of static conditions. Comparisons with scale modelling data show that excellent results can be produced, especially in areas where the velocity is relatively low, for example the spillway approach and crest region. For studies of these regions, CFD models using relatively coarse grids can produce accurate results very rapidly and allow for design alternatives and multiple upstream water levels or discharges to be evaluated and optimised rapidly.

It is critical to note that further work is required to improve CFD modelling techniques in relation to transient pressures, turbulence, cavitation, effects of air, two-phase flows, structure vibration and pressure pulsations in hydraulic jumps and on structure boundaries.

Complementary use of CFD and scale models allows for a degree of cross checking between the two sets of model results, allows for innovation and refinement of designs, provides greater confidence in the final design and will inevitably lead to improvements in both CFD modelling methodologies and scale modelling practices.

RÉSUMÉ

La conception de nouveaux déversoirs ainsi que leur amélioration a bénéficié récemment de l'utilisation des techniques complémentaires de la Mécanique des Fluides Numériques (MFN) et des techniques de modélisation physique. Le récent processus adopté lors de la conception ou de l'amélioration de déversoirs à écoulement complexe comprend un développement préliminaire qui utilise des méthodes théoriques et empiriques ainsi que des données

publiées. Ces développements préliminaires sont ensuite analysés et optimisés par la MFN et si nécessaire un modèle physique réduit est utilisé pour des optimisations et vérifications finales.

Une comparaison des résultats de la MFN et des modèles physiques est présentée. Les deux méthodes ont été utilisées dans le processus de la conception de déversoir et des conclusions en ont été tirées sur la possibilité de développer une analyse uniquement en MFN.

La MFN a montré dans les exemples présentés qu'elle était un outil de modélisation efficace pour la conception de déversoir, cependant cette étude était limitée à des considérations de conditions statiques.

Une comparaison avec des modèles physiques montre que de très bons résultats peuvent être produits ; en particulier lorsque la vélocité est relativement petite, à l'approche et au seuil du déversoir par exemple. Pour l'étude de ces régions, les modèles de MFN peuvent produire des résultats corrects très rapidement tout en utilisant un maillage relativement espacé. Ils peuvent aussi permettre l'évaluation et l'optimisation rapide de différentes idées de conception pour des débits et hauteurs d'eau multiples.

Il est essentiel de noter qu'il faut plus d'études pour améliorer les techniques de MFN en relation avec les effets de cavitation ; turbulence ; effet de l'air ; vibration de structure et les pulsations de pression dans les ressauts hydrauliques ainsi qu'à la limite des structures.

L'utilisation complémentaire de la MFN et des modèles physiques permet un élément de vérification de l'ensemble des deux résultats ; elle permet aussi une plus grande innovation et optimisation dans l'étude de conception ; donne une plus grande confiance dans le projet adopté et apportera inévitablement des améliorations dans les techniques de MFN et de modèles physiques.