# Efficient and cost-effective modelling and analysis of hydraulic structures using CFD

**Thomas Ewing, Marius Jonker & James Willey**

*GHD Pty Ltd*

*The use of Computational Fluid Dynamics (CFD) modelling techniques is gaining broad acceptance in the dams industry as an important design tool for hydraulic structures. This is particularly so in the earlier stages of analysis and design where the construction of physical models would be prohibitive on the basis of cost and time. Current CFD techniques allow users to produce a rapid evaluation of the existing conditions, which when coupled with the ability to quickly test an array of potential scenarios, enables the incorporation of innovative design solutions that may otherwise not have been considered during the design selection process prior to the advent of CFD capabilities.*

*Details of a recent case study are presented to illustrate the broad capabilities and benefits of CFD modelling techniques and their application in engineering analysis and design. The case study involves modelling of the Somerset Dam, a 50 m high concrete gravity dam with a gated overflow spillway including overtopping of the spillway bridge, gates and complex flow conditions in the abutment sections, which individually and collectively could not be accurately analysed with the traditional, simplified methods. The CFD study enabled an understanding of the hydraulic behaviour including discharge efficiency, jet impact loads on the gates and gate operating equipment and bridge structure; extent of potential erosion as a result of jet impingement on the abutments; loads on sluices and behaviour of the stilling basin. In addition to being a very large and complex model, the modelling involved several novel technical aspects.*

*The case study clearly highlights the benefits of the CFD modelling in understanding the complex hydraulic conditions and delivering cost effective solutions.*

***Keywords:*** *Computational Fluid Dynamics, Somerset Dam.*

## Introduction

Computational Fluid Dynamics (CFD) modelling involves the use of a computer to solve the equations of fluid motion within a complex geometry. CFD modelling has become an established tool in dams engineering projects, where multiphase CFD modelling (incorporating air and water fluid phases) is routinely used to investigate hydraulic issues such as crest flow capacity, approach flow dynamics, estimation of hydrodynamic loads on structural and mechanical elements, and the trajectory of overflow jets.

In the past, CFD modelling of large dams has generally been undertaken as a stand-alone task within the programme of larger, multi-disciplinary dams engineering projects. The scope of the CFD investigation (including any design options to be investigated) has generally been defined during the preliminary design process and provided at the outset of the modelling exercise. This method of incorporating CFD into a large project is similar to that employed for the provision physical scale modelling, which is not surprising, considering that there is considerable overlap in the capabilities of the two modelling approaches. This approach has also reflected the amount of time required to undertake CFD modelling. While more rapid than scale modelling, CFD simulations of a large spillway structure can take in excess of 24 hours to run and 20 – 50 simulations may be required for a typical project.

Recent advances in both the software used to undertake CFD modelling and the computer hardware used to run the simulations has greatly decreased the time required to both set up and run CFD simulations. Combined, these improvements have vastly decreased the time required to run large, complex simulations of dam spillways from days to hours. This now allows CFD analysis to be undertaken as an integrated part of the multi-disciplinary concept design process, undertaken during the design process, rather than testing and verifying predefined options. This paper illustrates the benefits of this modelling approach using the example of a major dams engineering project (Somerset Dam). This project also involved several interesting technical aspects relating to CFD modelling, which are also described.

## Technological change

The amount of time and effort required to undertake CFD studies has decreased dramatically in recent times. This change has been due to both technical improvements in the CFD modelling software and improvements in computer power, especially when considered in terms of value for money. Conversely, this has also allowed the modelling of more complex structures to be undertaken within a reasonable timeframe.

Commercial CFD software is used across multiple engineering disciplines including automotive design, aerospace engineering and biomedical research. Dams engineering represents an extremely minor usage case in the broader context of CFD analysis. Despite dams engineering being only a niche application of commercial CFD modelling software, there have been significant software developments that have had a major impact upon the amount of time and effort required to undertake CFD modelling for dams engineering applications. The key aspects include:

- **Improved integration with Computer Aided Design (CAD) packages including parametric design elements**. Modern CFD modelling packages allow for full integration with external CAD packages, including the transfer of geometric design parameters that are subject to optimisation. Using this type of feature, aspects of the design can be modelled and remodelled in CFD to determine the optimal design geometry. Examples of dams engineering applications where the optimisation of design parameters are useful include the radius of spillway training walls, the position of vertical and radial gates, and the height of chute walls.
- **Vast improvements in automated meshing.** Simulation of dam spillways involves transient multiphase modelling over large spatial scales. This is a computationally demanding application of CFD. In order for models to execute with reasonable amounts of time and with reasonable accuracy a hexahedral mesh with mesh refinement in key areas is generally required. Until relatively recently this type of mesh required significant manual input from an experienced meshing operator. As an example, at the time the mesh used for CFD modelling of the Enlarged Cotter Dam (Willey *et al.* 2010) took over 40 hours to manually generate (for each geometry investigated). Major advances in meshing software allow high quality meshes to be generated with no user input (once properly configured initially). In the case of the Somerset Dam (which is significantly more complex than the Enlarged Cotter Dam), the mesh took less than 15 minutes to generate.
- **Improvements in user interfaces, post processing, and reporting.** The capability of the commercial CFD packages to generate high quality figures, animations, summary result tables, and reports has improved dramatically over time. These are now generally generated as part of an automated process, allowing results to be provided to the project team as soon as the simulations have completed. The performance of the numerical models have also improved slightly, decreasing run times.

On the computational hardware side the following advances have greatly increased the speed of CFD modelling simulation:

- **Availability of multiple processor workstations with multi-core processors.** Physical aspects (die size, heat dissipation limits) have limited the advancement of processor speeds for some time. In order to improve total performance, processors have been released that incorporate multiple computational cores. It is now common for workstations to include 20 computational cores across 2 processors. This change (along with improvements to the memory bandwidth of modern processors) has resulted in an order-of-magnitude scale increase in computational power available for modelling over around 4 years.
- **Availability of high performance networks.** The availability of specialist low latency and high bandwidth networking hardware and protocols allows multiple workstations to be connected and used in parallel to process CFD simulations more quickly. Using this technology, the actual combined performance for multiple workstations is close to the theoretical sum of their individual capability, at least at the scale likely to be employed for hydraulic modelling (around ten linked workstations with around 200 combined computational cores).
- **Improved value for money.** While workstations with many cores and high performance networks have been available in some form for some time their use has been largely cost-prohibitive (for dams engineering). Improvements in the performance of these components have come simultaneously with large decreases in cost.

The combined impact of these improvements is a one to two orders of magnitude increase to the practical computational power available to hydraulic modellers over about the past 4 years. In practical terms, simulations which took 48 hours to execute in 2011 can be run in 1 - 4 hours in 2015. This improvement, along with the improvements to CAD integration and meshing mean that one (or more) large dam spillway simulations can be prepared and completed within one day (or more realistically, over-night). This change in capability allows for CFD to be integrated into the concept design phase of dams engineering projects, rather than forming a standalone study.

## Somerset Dam

Somerset Dam is a large mass concrete gravity dam located on the Stanley River, a tributary of the Brisbane River, in the upper limit of the Wivenhoe Dam storage and approximately 115 km northwest of Brisbane, Australia.

The spillway consists of a main ogee section with radial gates. A bridge deck spans the entire dam. On both sides of the ogee a gap exists between the top of the dam wall and the bridge deck, referred to as the breezeways (see Figure 1).

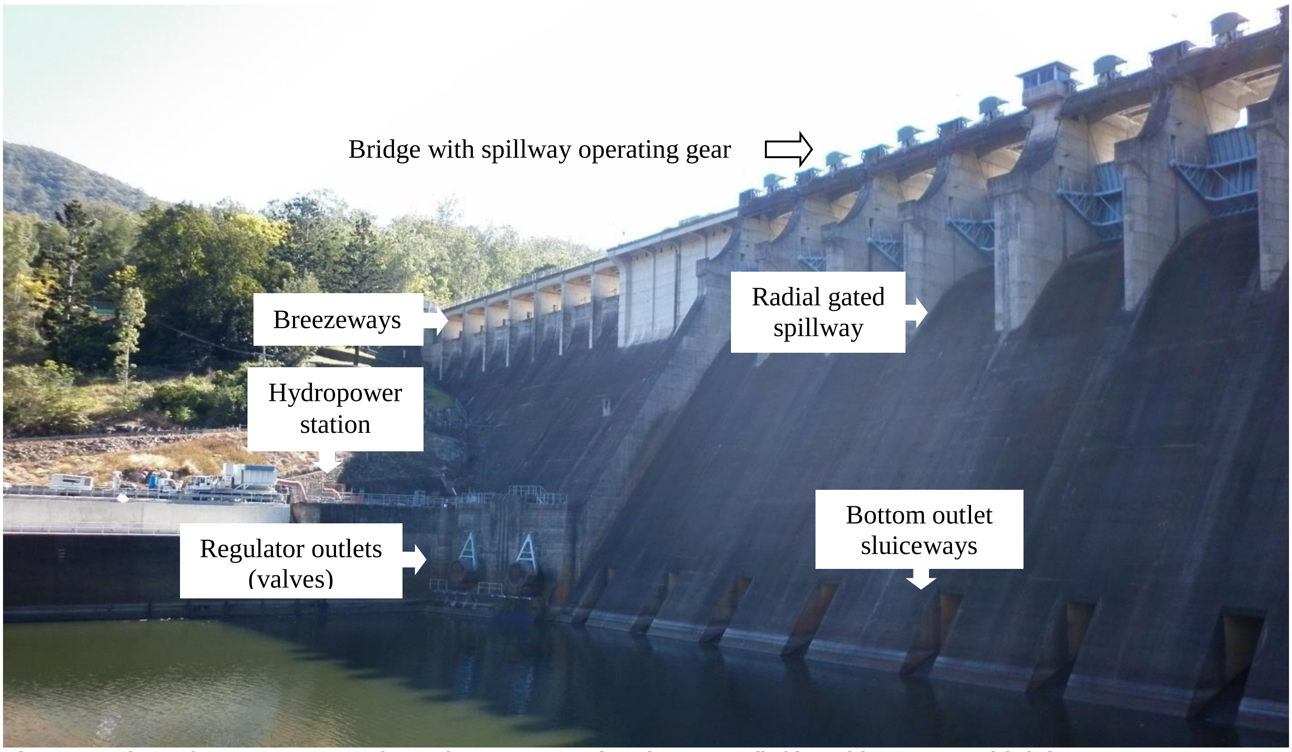

**Figure 1: View of Somerset Dam from downstream showing ogee (left) and breezeway (right).**

Seqwater, the owner of Somerset Dam, engaged GHD in 2014 to undertake a Flood Upgrade Options Feasibility Study. The key objectives of the Study was to develop high level concepts for upgrade works to Somerset Dam so it can safely withstand extreme lake levels and safely pass extreme floods up to the Probable Maximum Flood (PMF). The upgrade is necessary to address dam safety issues identified in previous studies and to satisfy the Department of Energy and Water Supply (DEWS) Guidelines on Acceptable Flood Capacity (AFC) for Water Dams (DEWS 2013).

In addition to DEWS AFC requirements, the Queensland Government is also currently considering options to improve flood mitigation at Wivenhoe Dam and other potential flood mitigation options. This may bring forward the need to upgrade Somerset Dam ahead of the minimum timeframes specified in the DEWS AFC guidelines.

The Flood Upgrade Options Feasibility Study was undertaken in view of the need for further investigations and studies as recommended in the latest safety review for Somerset Dam (URS 2014). The safety review highlighted that, as the dam is an extreme hazard dam, and using the 'fall back' approach, it is required to safely pass the PMF. The current spillway and outlet works configurations do not have sufficient capacity to pass the PMF and flows will overtop the existing breezeways (portals in non-spillway section) and bridge deck. The dam was not designed to overtop, and considering the extent of erosion, in particular on the abutments downstream of the breezeway monoliths, it was concluded that the dam cannot safely pass the design flood event. The safety review also highlighted the dam does not have adequate factors of safety under the PMF design load or the maximum design earthquake (MDE). In addition, the stilling basin slab does not satisfy current design standards for usual, unusual, and extreme flood events, and the stilling basin training walls do not meet current design standards under submerged and rapid drawdown conditions.

As part of the safety review the risk analysis was updated from the Portfolio Risk Assessment (PRA) which was completed for Seqwater's portfolio of 26 referable dams (URS 2013). The review of the risk numbers indicated that the societal risk for the Somerset Dam now plotted above the ANCOLD limit of tolerability, which requires that action should be taken to reduce the risk (except in exceptional circumstances).

Operation of Somerset Dam during a flood event is undertaken in accordance with the Manual of Operational Procedures for Flood Mitigation at Wivenhoe Dam and Somerset Dam (Seqwater 2013). DEWS have recently led the Wivenhoe and Somerset Dams Optimisation Study (WSDOS), to review the operation of the dams and investigate possible improvements to flood operations (DEWS 2014). The outcome of the WSDOS was that both Wivenhoe Dam and Somerset Dam need future upgrades to meet the requirement to safely pass 100% of the PMF to satisfy the AFC guidelines. The WSDOS also identified appreciable potential for Somerset Dam lake level to reach the maximum safe storage level before Wivenhoe Dam lake level reaches its crest. A further limitation noted in estimating the PMF lake level for Somerset Dam is the uncertainty in the hydraulic ratings for overflow of the dam crest (flow through the breezeway, including orifice flow) and flow over the bridge deck for high to extreme lake levels.

The Flood Upgrade Options Feasibility Study therefore included CFD modelling of the entire dam structure to develop the hydraulic discharge rating, as well as to aid in the development of the concept designs to modify the dam to safely pass the PMF.

CFD modelling was used extensively in the development of a concept design for modifications to Somerset Dam to allow safe passage of extreme event flows. The following sections describe this CFD modelling, summarising the results and providing insight into the way that the modelling was incorporated into the concept design process.

### Impact of gate position of crest discharge capacity

One initial unknown identified in the development of the concept design was the effect that the radial gates would have on the capacity of the ogee when operating at high reservoir levels. Related to this issue was the potential for improvements to the ogee discharge capacity if the concept design incorporated features to rotate the radial gates further than is currently possible (over-rotation). A good understanding of the performance of the ogee section of the crest at high reservoir levels was essential to the development of design concepts.

In order to rapidly evaluate the performance of the crest, a sub-section of the crest was evaluated. This approach uses symmetry boundary conditions to simulate repeating geometry of the full ogee crest. Reducing the size of the model allowed for simulations of the existing crest arrangement to be undertaken over just a few days.

Simulations of the existing geometry indicated that the gate in the current fully open position did strongly interact with the ogee flow at high reservoir water levels. Figure 2 shows the water level results from one simulation undertaken. This simulation identified problematic issues with the existing configuration, including:

- Interaction of the radial gate with the flow over the ogee,
- Impact of flows over the bridge deck on the gate counterweight, and
- Creation of an air pocket under the bridge deck and gate which could results in surging flow behaviour as the air mass is entrained into the main flow.

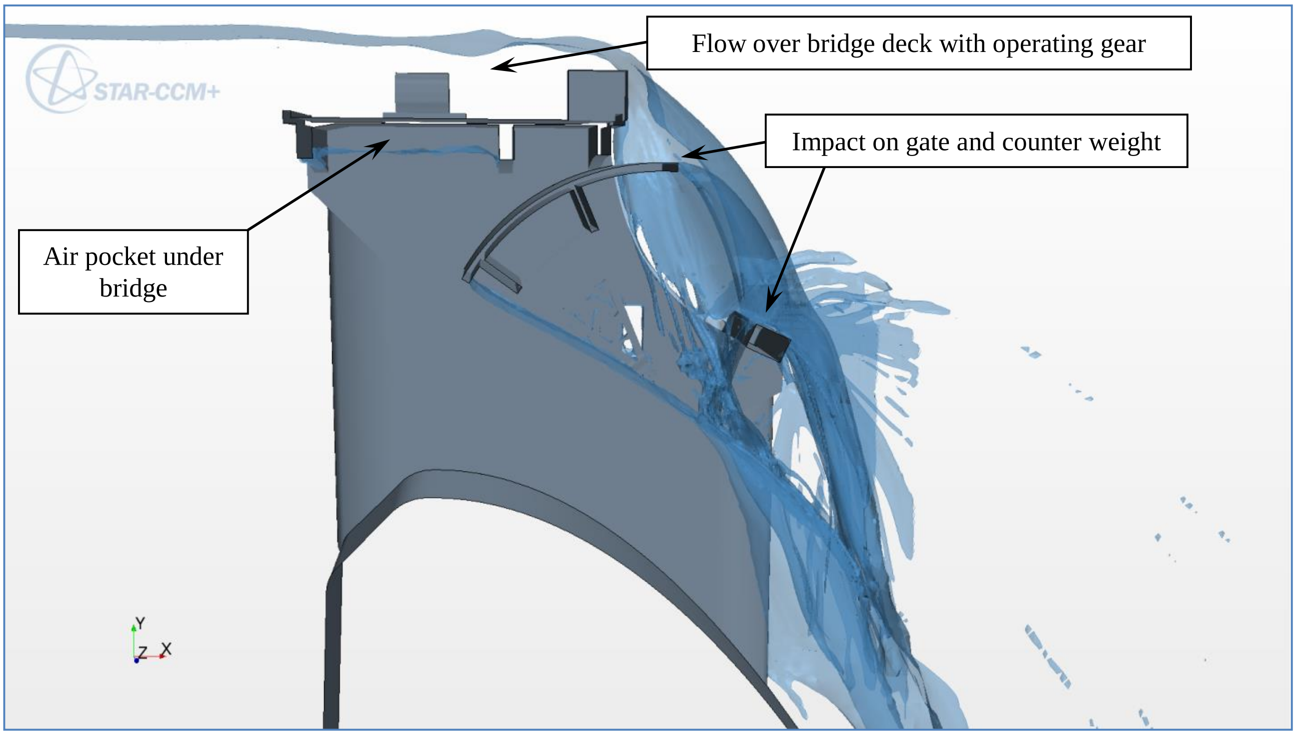


**Figure 2: CFD water level results for the existing crest arrangement (115.0 m AHD reservoir water level).**

**These results led to a phase of concept development relating to modifications to the crest ( Figure 3). Key modifications include over-rotation of the radial gate, addition of a 'knife edge' wall on the underside of the upstream face of the bridge deck, and the addition of a flow deflector to provide ventilation to the back of the crest peirs (and to prevent the formation of an air pocket under the deck).**

Figure 3 shows the flow trajectory for the modified arrangement. In addition to the elimination of the air pocket under the bridge deck, the flows are deflected over the gate counterweight. Flows are also increased by 23% at this water level under the modified arrangement. It was identified as part of these simulations that at high reservoir levels, crest pressures drop to levels where cavitation will occur. Modelling of the existing crest requires that this issue be considered when configuring the CFD model. These crest simulations included a cavitation sub model.

Figure 3 shows a zone at the crest where cavitation results in the formation of water vapor. Early identification and analysis of the low crest pressure issue allowed the incorporation of appropriate modifications (a modified crest profile) into the concept design.

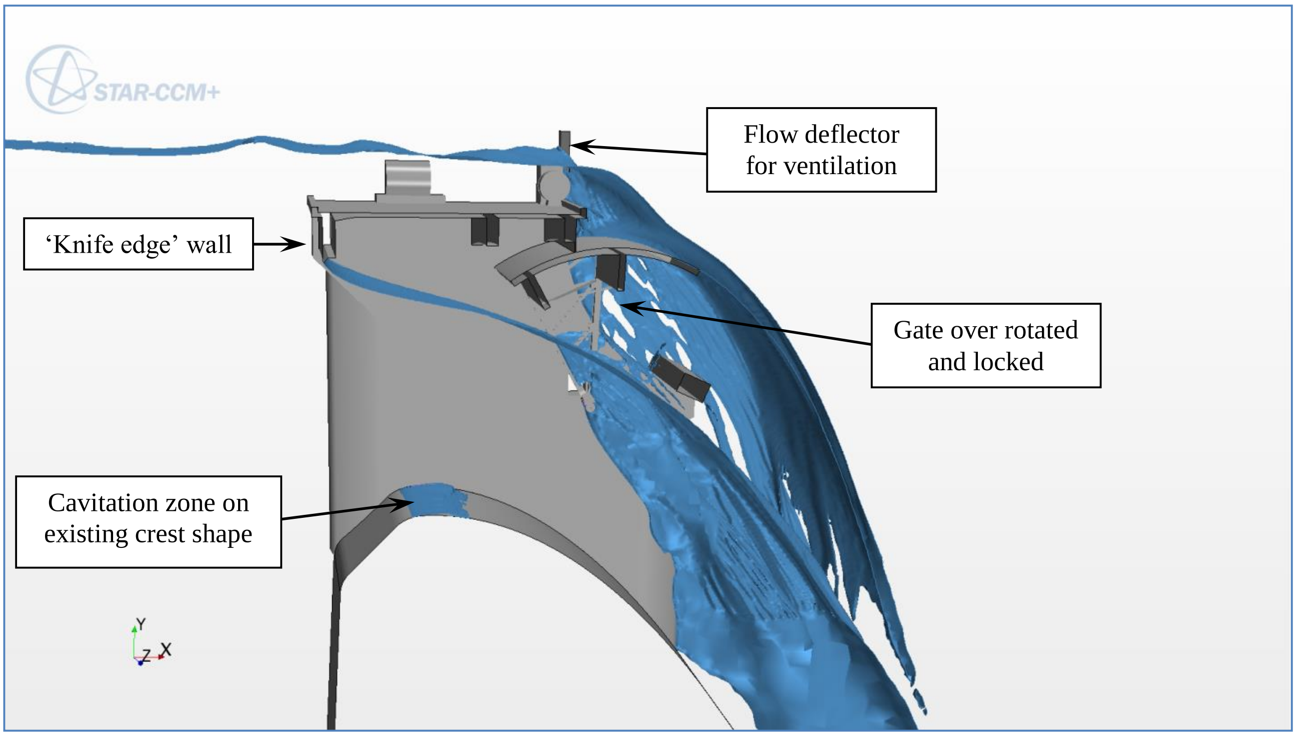


**Figure 3: CFD water level results for the modified crest arrangement (115.0 m AHD reservoir water level).**

### Capacity of the crest and breezeway options

One of the options considered during the development of the final concept design involved only minor modifications to the crest (over-rotation of gates, addition of knife edge wall, bridge deck flow deflectors), and flow was allowed to pass over the existing breezeway structure. The concept involved the creation of new stepped abutment channels to convey the breezeway and bridge deck overflows on the abutments into the main channel at the toe of the dam. In order to quantify the discharge capacity of this option, a large CFD model of the entire proposed structure was constructed. Despite the geometric complexity of this large model (incorporating gates, flow deflectors, the bridge deck, stepped chutes, and the natural channel downstream), modelling of this options was undertaken over the course of a few days during the concept design phase. An example of model output is shown in Figure 4.

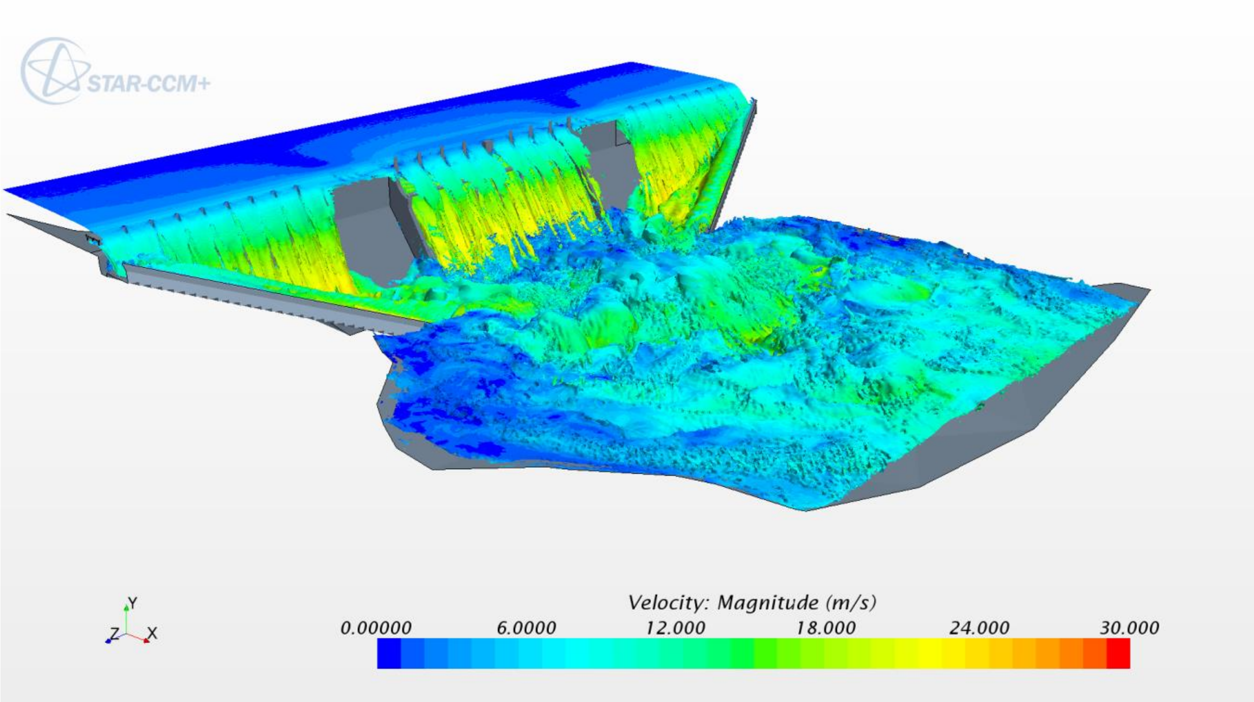


**Figure 4: Surface velocity results for a proposed crest arrangement.**

Modelling the spillway efficiently at this scale presented several technical challenges. Modelling at this scale requires a coarser numerical mesh than that used for the previous crest modelling. Due to the coarser mesh, accurate simulation of the complex air flows at the back of the bridge deck flow deflectors and pylons was not feasible. Despite this, the correct pressure in the air mass under the bridge deck is crucial for the accurate simulation of flow over the ogee and breezeway sections of the crest. In order to counteract this issue, air vents were added to the large scale CFD model under the bridge deck. These vents ensure that the air mass under the deck does not deviate significantly from atmospheric pressure and replicate the effective operation of the bridge flow deflectors, demonstrated in the smaller scale simulations.

Simulation of cavitation on the ogee crest was also more challenging at this scale. The scale of the full model, as well as the larger cell size and longer model time-step adopted precluded the use of an explicit cavitation sub-model (as was used in the crest simulations). As an alternative, pressures within the simulation were simply limited to that where cavitation occurs. This approach prevents simulation of very low pressures at the crest which could unrealistically improve the crest efficiency.

The results of simulations of this configuration were then compared to corresponding CFD results for a competing configuration where the ogee crest profile was modified, the breezeways closed to flow, and a wall added to the bridge deck to prevent overtopping flow. With these flow results available during the concept design stage the options could be compared with a high level of confidence early in the project.

CFD modelling was also used in several additional investigations which informed the final concept design. These included an analysis of hydrodynamic forces and flow capacity of the sluice gates in the bottom outlet sluiceways, an estimation of the loads on the bridge deck for the case where it is allowed to overtop, as well as a trajectory analysis for flows over the breezeway and bridge deck used to inform an erosion analysis of the abutments.

## Summary and conclusions

Significant improvements in the capabilities of CFD modelling software and hardware now mean that hydraulic simulations can be undertaken very rapidly and can be fully incorporated into the dams engineering design process. The benefits of such an approach were demonstrated in a recent concept design project involving Somerset Dam. In this instance, the use of the CFD models allowed the project team to identify hydraulic issues early and to investigate a broader range of engineering options, including more speculative options. Early analysis of options allowed for effort to be focussed on evaluating and optimising only the most promising options.

Use of CFD during the design process presents additional technical challenges to modelling engineers as the modelling undertaken must meet both quality standards and timeliness requirements. As such it is critical that engineers have a clear understanding of the capabilities and limits of the CFD modelling approach as applied to dams engineering. Continuous review of the modelling methodology and results is essential as the designs develop.

The capabilities of CFD modelling software and hardware will continue to develop in coming years. This will, no doubt, continue to change the way that CFD modelling is used in dams engineering projects. In the near term it is likely that accurate real-time analysis of complex dam structures will become a practical, as well as automated spillway design optimisation. These advancements, when available, will further add to the tools available to designers.

### Acknowledgements

The authors would like to thank our client, Seqwater, for permission to publish this paper, in particular Barton Maher and Michel Raymond. The authors would also like to thank GHD for their support in the preparation of this paper.

### References

DEWS 2013. Guidelines on Acceptable Flood Capacity for Dams. Queensland Department of Energy and Water Supply.

GHD 2015. Somerset Dam Flood Upgrade Options Feasibility Study. Draft Report, GHD.

Seqwater 2013. Manual of Operational Procedures for Flood Mitigation at Wivenhoe Dam and Somerset Dam.

URS 2014. Somerset Dam Safety Review. Final Report, URS.

Willey, J.; Ewing, T.C.; Lesleighter, E.; Dymke, J. 2010. Numerical and physical modelling for a complex stepped spillway. *Journal on Hydropower and Dams*, Volume 17, Issue 3.