5th International Symposium on Hydraulic Structures
Hydraulic Structures and Society: Engineering Challenges and Extremes
ISBN 9781742721156 - DOI: 10.14264/uql.2014.23 Brisbane, Australia, 25-27 June 2014

Rifle Creek Dam DIY Physical Modelling

F. Jacobsen[1]
[1]Engys AUST
47 Camelot Street
Brisbane Tennyson QLD 4105
AUSTRALIA
E-mail: f.jacobsen@engys.com.au

Abstract: *The application of Computational Fluid Dynamic (CFD) models can be fraught with uncertainties for the inexperienced modeller. The simulation results can vary radically depending on assumptions made regarding; boundary conditions, model domain, turbulence parameters etc. The acquisition of valuable experience and engineering judgment comes with observation of relevant experimental data, which however is often difficult to source. For Rifle creek Dam near Mount Isa, Australia, the author took the unusual step of developing a DYI (Do it Yourself) physical model of the dam was in the backyard of the CFD modeller to do experimental comparison with VOF CFD (HELYX) results. While it was well outside of the project scope it was well within the required sleep-at-night factor. The purpose of the project was to investigate methods of increasing the spillway capacity of the 1920's built dam as per ICOLD (International Committee on Large Dams) dam safely requirements for acceptable flood capacity. Many older dams are subject to similar investigations, due to revised estimated maximum flood sizes, which often also result in retrofit construction activities to increase spillway capacity. Comparison of the simulation and the physical model characteristic spillway flow of the spillway is demonstrated by numerous videos.*

Keywords: *CFD, Spillway, OpenFOAM, HELYX, arch dam*

1. INTRODUCTION

Rifle Creek Dam is an arch gravity concrete dam located on the Rifle Creek, a tributary of the Leichardt River. It has a total curve length of 125 m; maximum height of 18m and a total storage capacity of approximately 9500 ML. It was constructed in 1929 to provide a water supply to Mount Isa Mines and the township. Flood waters are currently discharged through an uncontrolled ogee overflow spillway in the centre of the dam.

AECOM Australia Pty Ltd was commissioned by the owners of Rifle Creek Dam to assess options and costs for increasing the spillway capacity of the Rifle Creek Dam spillway.

This paper comprises of a brief description of the CFD modelling followed by details the physical modelling. Many videos of the physical and CFD models were developed, some of which are rendered using third party animation software.

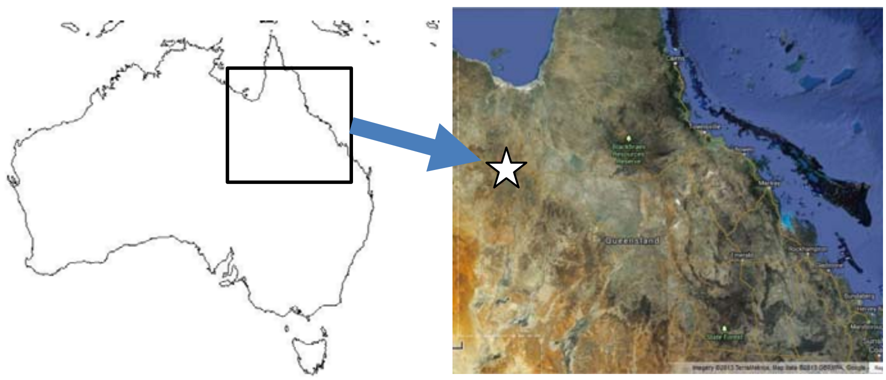

Figure 1 – Project Location

1.1. Background and Scope

The Acceptable Flood Capacity (AFC) of the dam was carried out, as per ICOLD (International Committee on Large Dams) (SMEC 2009). It was determined that the existing dam crest is overtopped by a 1 in 30 year AEP (Annual Exceedence Probability) flood. The study concluded that the AFC is a 1 in 40,000 year AEP flood which requires the dam to safely pass a discharge of 790m^3/s.

After a preliminary assessment it was concluded that the preferred option to raise the spillway capacity is to allow the concrete portion of the abutments to overflow. An assessment of the bed erosion adjacent to the downstream dissipator structure was undertaken using computational fluid dynamics (CFD) modelling techniques.

A CFD model was developed because the arrangement of the flip bucket dissipator did not match current practice for flip buckets and roller buckets (USACE 1990). The roller buck configuration would require higher tail-water than exists in Rifle creek. The configuration of the dissipator also falls outside the guidelines as a flip bucket. The angle of the flip is vertical which is much steeper than current practice. Therefore a 2-dimensional CFD model was developed to assess flip bucket downstream erosion potential. Also, a 3-dimensional model of the crest and adjacent rock abutment was developed to assess extent of protection, but is not described further in this paper.

1.2. Rifle Creek Dam Details

A cross section and plan view of Rifle Creek Dam is shown in Figure 2. The original crest was raised by adding a rectangular section.

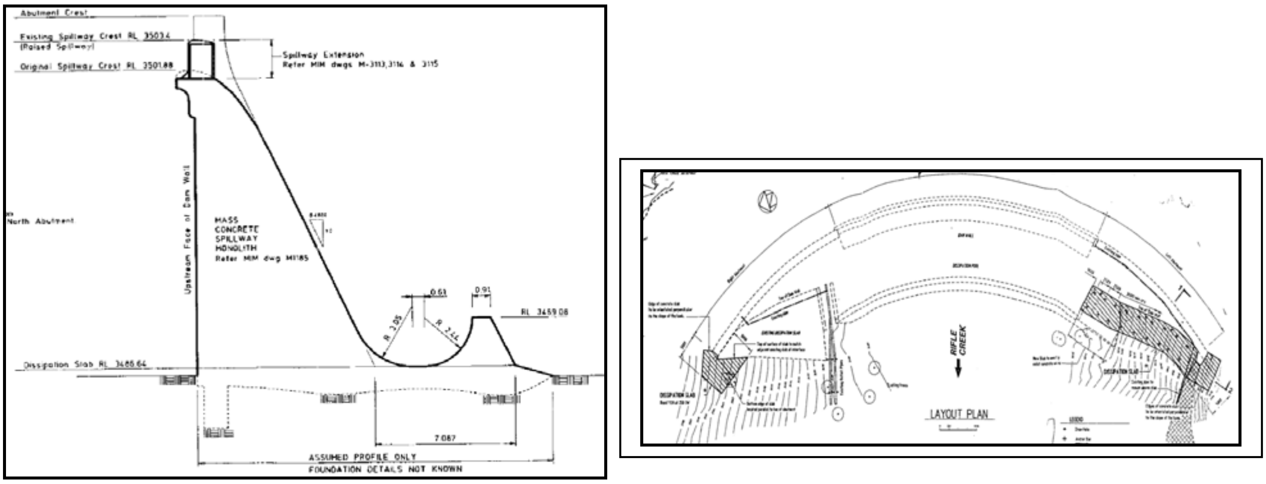

Figure 2 – Rifle Creek Dam Cross section and plan

The plan of the dam (Fig. 2) shows the central spillway section and previous abutment erosion repairs, (shown as cross hatching).

2. CFD MODEL

The software package used to undertake this study is called HELYX which comprises of a Graphical User Interface, and HELYX-Core, which is an enhanced version of the open-source CFD library OpenFOAM

HELYX can be used to simulate complex fluid flows involving chemical reactions, turbulence and heat transfer. The OpenFOAM software package has been the subject of benchmarking studies for the analysis of hydraulic structures including dams (Jacobsen 2009). A validation of HELYX was carried out by the author by replicating the same model setup as described by Savage and Johnson (2001). A

2-dimensional physical model of a spillway was used as a benchmark for comparison against various methodologies. Figure 3 shows the results of the depth of flow over the crest of the spillway for all the cases. The author has included results of the HELYX model which compares quite favourably with the other methods.

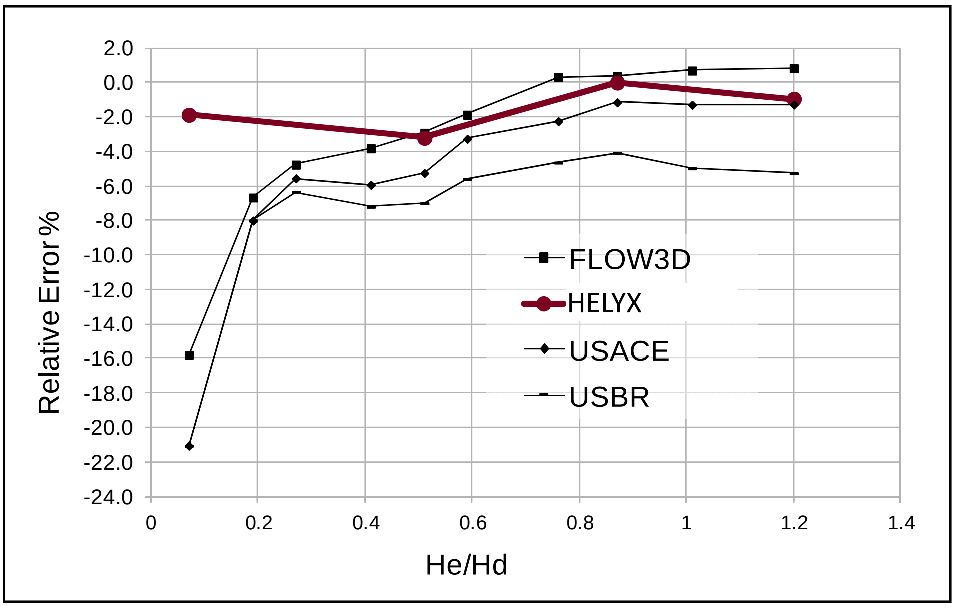

Figure 3 – Spillway Flow HELYX validation (Relative Percent Error in Depth compared to Physical Model)

1.3. Rifle Creek CFD Model setup

A cross section of the model is presented in Figure 3, which describes a typical cross section through the spillway. The model also extends downstream approximately 30 meters. This arrangement assures that flow characteristics over the spillway are realistically modelled. It should be noted that the real flow will be 3D everywhere, since the spillway is curved in plan, and that there will be discrepancies between the 2D models and the real world 3D flow, though small in the central part of the spillway.

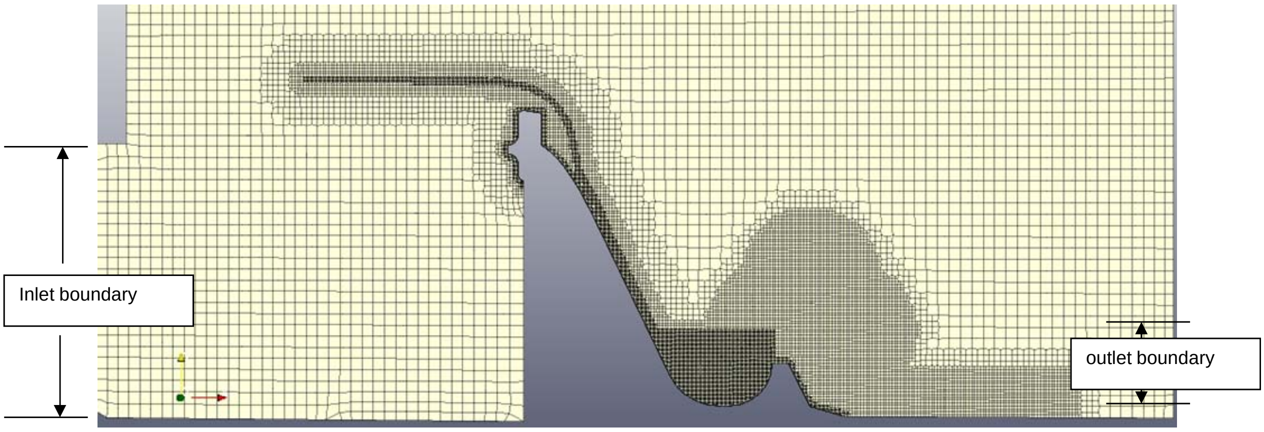

Figure 4 – HELYX CFD Mesh

The mesh consists of an unstructured grid. The mesh has been refined in the areas of interest and the concentration of grid points is reduced in areas of less interest such as above the water surface level. The mesh is comprised of 650 000 cells. The dimensionless value of *y+* indicates the required resolution of the grid spacing in the boundary laminar layer.

$$y+ = \rho * u * y / \mu \quad (1)$$

where ρ = density, u = friction velocity, y = distance from wall, μ = dynamic viscosity. The first grid point is located at approximately $y+$ = 20-80 along walls to resolve near-wall flows using the near-wall function. The mesh resolution elsewhere was determined by grid size sensitivity analysis. The time step is controlled as a function of the dimensionless Courant number value adopted (0.5). The Courant number is defined as;

$$C=u\Delta t/\Delta x \quad (2)$$

where u =magnitude of velocity Δt = the time step and Δx= grid size interval. Other details and assumption of the final model set up includes;

- Turbulence modelling: Large Eddy Simulation (LES), Smagorinsky
- Incompressible flow,
- transient
- multi- phase flow

The turbulence parameters used are included in the sub grid scale SGS kinetic energy equation, k_{sgs} (Smagorinsky 1963)

$$k_{sgs} = 2C_k/C_e \, \Delta^2 S^2 \quad (3)$$

where: C_k=0.07 C_e=1.05 are constants determined theoretically, S=rate of strain tensor, Δ=filter size.

3. CFD RESULTS

The results of the CFD analysis provide a good indication of the performance characteristics of the spillway dissipator. However the results are very different for the same case using different turbulence solvers, i.e. Large Eddy Simulation (LES) or Reynolds Averaged Naiver Stokes (RANS). It was these different results which lead to construction of the physical model which is described later. The physical modelling confirmed that in this case, the LES model closely agrees with physical modelling results.

It was found that at low flood events a hydraulic roller forms at the toe of the dissipator, as shown in Figure 5. Erosion occurs adjacent to the flip bucket at lower flows. This result is validated by the existing protection works at the toe of the dissipator Figure 5 (left). It is proposed to extend the existing rock protection to a total of 10m from the toe of the dissipator.

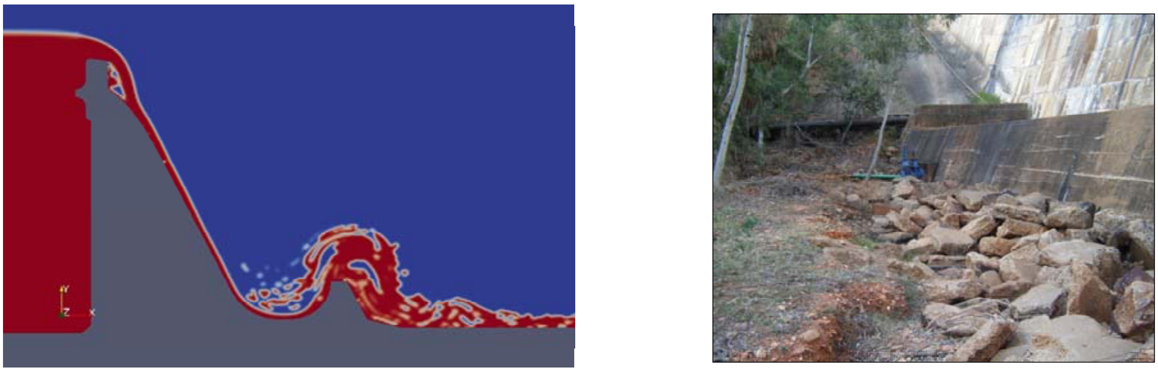

Figure 5 – 25% AFC (329 m^3/s), (left). Existing erosion protection at the toe of the dam, (right)

During high flow events flow is directed away from the dam in a trajectory which is characteristic of a flip bucket Figure 6. Therefore existing dumped rock protection work is sufficient to maintain erosion protection under a range of flood events from rare to extreme.

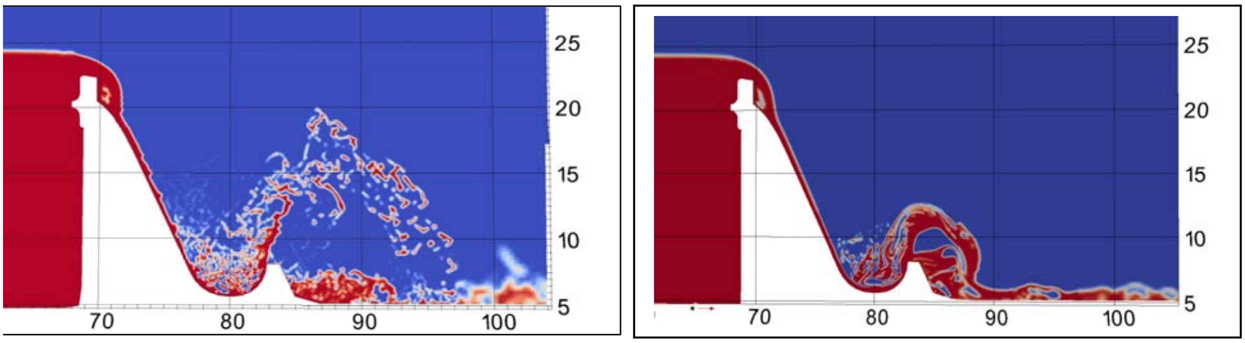

Figure 6 – 50% AFC (447 m^3/s), LES (left) RANS (right)

The disparaging results for the two turbulence models are also shown, i.e. LES Figure 6 (left) and RANS Figure 6 (right).

4. PHYSICAL MODELLING

While the author is confident of the CFD results which were reported to the client, the basis of the confidence stems from previous validation studies and projects. These previous studies undertaken by the author do not include flip bucket energy dissipators. Therefore the grand decision was made to construct a scale physical model at the home of the author using laminate timber material left over from a recent re-flooring of a family room. The principle model objectives include measurement of the height and extent of the flip buck plume and observable flow characteristics.

The construction objectives include accurate geometry to within 1mm. The geometry of the 1920's spillway consists of mostly straight lines and circle arcs with easily identifiable center points. This is in contrast to modern ogee crest spillways with crest geometry identified by equations and compound arcs. The main area where the model setup differed from a hydraulic laboratory was in the accuracy of the instruments for measuring flow. It was envisaged that the discharge capacity would be determined by filling a container within a time frame. Also, a model developed in a laboratory would be wider at around 600 mm compared to the current study which is 200 mm.

1.4. Physical Model Scale Similitude

To meet the objectives and for the exercise to be valid, a reasonable geometric scale had to be chosen and also determination of which flow similarity ratio for kinematic similarity to use. The relevant parameters requiring consideration for this dimensional analysis include; geometry, flow properties and fluid properties .i.e. density of water (kgm^{-3}) dynamic viscosity (Pa s),etc. This leads to five dimensional independent parameters which consist of; Froude number (ratio of inertia force to gravity force), Euler number (ratio of pressure force to gravity force) Reynolds number (representing turbulence) Weber number (surface tension) and Sarrau-Mach number (inertial force to elasticity force). This raises the question of which of these dimensional parameters to use in order to achieve practical dynamic similarity?

Compressibility is negligible for free water surface cases, such as ours, and the Sarrau-Mach number is very small in both model and prototype. The Weber number becomes prominent when modelling air entrainment or dispersed bubbles. The Euler number is normally adopted for modelling in wind tunnels. With free-surface flows, the characteristics are primarily dependent on; incompressible, gravitational and inertial forces. Therefore Froude is adopted and achieved by setting the model's Froude number equal to the prototype's Froude number. Froude number is used generally for scaling free surface flows for hydraulic structures.

Finally how to choose the geometric scale? The main consideration for choosing the right geometric scale becomes clearer when we consider that the spillway energy dissipator is very turbulent. By adopting Froude scale similitude it follows that Reynolds number similitude is not concurrently met. However it is considered important that the model Reynolds number should be kept as large as possible so that model turbulence is fully developed (Chanson 1999).

By trial and error, calculation of Reynolds number in the model, within the bucket, for different geometric scales results in a Reynolds number >$5e^5$, which is a turbulent flow regime, for a geometric scale ratio of 1:50. This geometric scale falls within the range typically used for free surface flow models, and fortunately provides a practical size for scale model construction.

In summary, Froude number was adopted for flow scale similarity and Reynolds number used to confirm geometric scale. Both the physical model and the 3D CFD model (described later) developed for comparison with the physical model have the same geometric scale therefore full dynamic similitude was achieved. However the final full scale CFD model was revised based on the lessons learnt from the physical model. The following scaling relations can be deduced:

- Length ratio: Lr = Lm/Lp = 1/50
- Velocity ratio: Vr = $Lr^{1/2} = (1/50)^{1/2}$ = 1/7.07
- Discharge ratio: Qr = $Lr^{5/2} = (1/50)^{5/2}$ = 1/17,680

The model layout is shown in Figure 7. The model spillway crest is approximately 360 mm high and 200 mm wide. The list of construction materials used is listed below;

- Left over laminate Flooring material
- Liquid nails
- Silica (Heaps)
- Plastic 1L milk bottle
- Clear Poly-plastic
- White color spray cans
- 2'' pump capacity approximately 5l/s

Figure 7 – Physical Model Arrangement

1.5. Physical Modelling Results

The CFD model was modified for final comparison with the physical model. It was extruded to a 3-dimensional model with the same width as the physical model. Also the CFD model was rescaled to the physical model scale. The inlet boundary flow was adjusted to match the pump capacity which did not have variable speed adjustment. The results graphically illustrated as a series of videos. Many photos and videos were captured of the physical model. Both the CFD and physical model displayed the same average plume height (10.0m) and length (19m) as well as the same characteristic flow patterns at the flip bucket, as illustrated in Figure 8, with no discernable difference. The flow clearly separates from the downstream edge of the flip bucket as shown in Figure 9. From observation, it was found that the CFD model did not capture flow separation at the crest and the average tail water level in the bucket at 3.1m was less than that for the physical model, measured 4.9m. This was determined to be due to lack of mesh refinement at these locations. No more work was carried out on the 3D CFD model. Further mesh refinement, based on lessons learnt from the physical model, was applied to the 2D CFD mesh which consequently captured these flow characteristics, as shown in Figure 10.

Although the configuration of the spillway dissipator is outside the current guidelines for operation as either a flip bucket or roller bucket, it is evident that a physical model was constructed during the original design of the dam.

Both the physical and CFD model confirm that the spillway dissipator operates like a flip bucket. However, the original physical model report is no longer available.

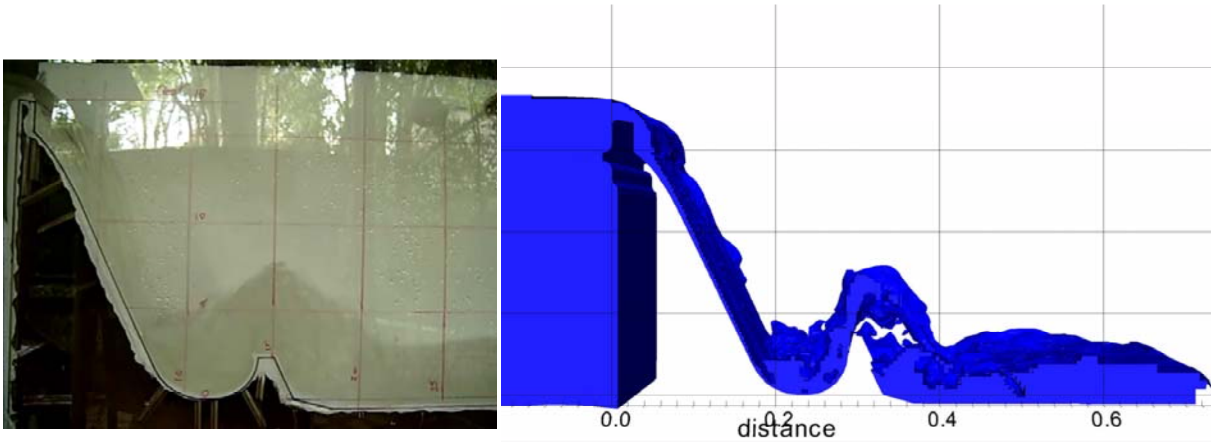

Figure 8 – CFD &Physical model comparison: side view

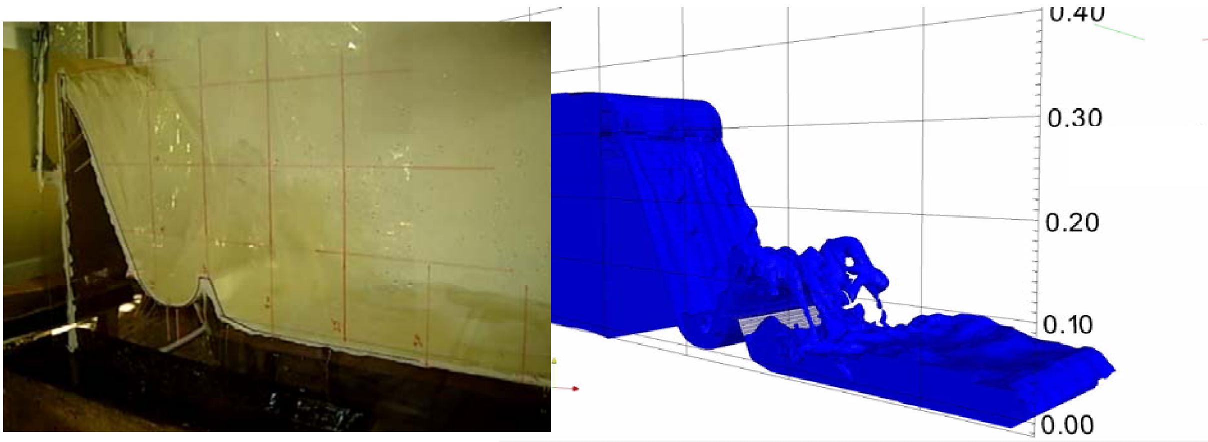

Figure 9 - CFD &Physical model comparison: three quarter view

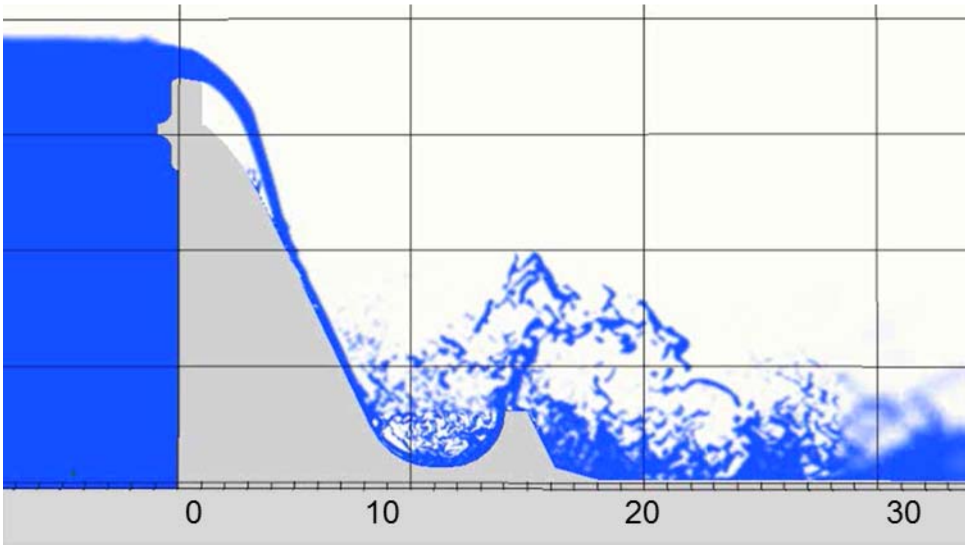

Figure 10 – Final CFD 2-dimensional model

5. SUMMARY

A comparison of the HELYX CFD package was made against a physical model built in the backyard of the author, which was outside the project scope, for a dam in North Queensland. This arose due to disparity of results when using two common turbulence equations. It was found that there was excellent agreement between the CFD LES model and physical model. The physical model was constructed privately at the home of the author in relatively short time and was of immense practical value. The Rifle Creek dam spillway has a simple geometry which lends itself well to this type of very rare endeavour. While limitations are acknowledged, such as accurate flow measurement, it was extremely useful for the project and a great experience that is highly recommended.

6. ACKNOWLEDGMENTS

I wish to thank AECOM for permission to present this paper and my ex-colleagues at AECOM including Ralf Sieberer for their kind help regarding all aspects of the Rifle Creek project.

7. REFERENCES

1. Savage, B., and Johnson, M.C. (2001), *Flow Over an Ogee Spillway: Physical and Numerical Model Case Study*. Journal of Hydraulic Engineering, 127 (8), 640-649.

2. F. Jacobsen, F. (2009), *Application of OpenFOAM for designing Hydraulic water Structures*, Paper presented for Open Source CFD International Conference 2009, Barcelona 12th/13th November 2009.

3. SMEC (2009), *Rifle Creek Dam Failure Impact Assessment*, SMEC

4. USACE (1990), *Hydraulic design of spillways*, USACE Engineering Manual EM 1110-2-1603

5. Chanson, H (1999), *The Hydraulics of Open Channel Flow*, Edward Arnold, 338 Euston Road, London NW1 3BH, UK.

6. Smagorinsky, J (1963), *General circulation experiments with the primitive equations, i. the basic experiment*., Monthly Weather Review, 91: pp 99-164, 1963.