Proceedings Paper:

Torres, C, Borman, D, Sleigh, A et al. (2017) Three dimensional numerical modelling of full-scale hydraulic structures. In: Proceedings of IAHR 2017. 37th IAHR World Congress, 13-18 Aug 2017, Kuala Lumpar, Malaysia. International Association for Hydro-Environment Engineering and Research (IAHR), pp. 1335-1343. ISSN: 2521-7119. EISSN: 2521-716X.

THREE DIMENSIONAL NUMERICAL MODELLING OF FULL-SCALE HYDRAULIC STRUCTURES

CATERINA TORRES[1], DUNCAN BORMAN[2], ANDY SLEIGH[3] & DAVID NEEVE[4]

[1] School of Civil Engineering, University of Leeds, Leeds UK
e-mail cnctmo@leeds.ac.uk

[2] School of Civil Engineering, University of Leeds, Leeds UK
e-mail d.j.borman@leeds.ac.uk

[3] School of Civil Engineering, University of Leeds, Leeds UK
e-mail p.a.sleigh@leeds.ac.uk

[4] Arup, Leeds, UK
e-mail david.neeve@arup.com

ABSTRACT

This study presents the three-dimensional (3D) hydraulic modelling of free surface flows over complex full-scale hydraulic structures. The work outlined therein forms part of a larger ongoing project which focuses on the assessment of the capabilities of different 3D computational fluid dynamics (CFD) techniques to reproduce hydraulic flows over real scale spillway structures and on the comparison with physical scale models. The aim of the first part of the study, presented in this work, is to evaluate a range of 3D free surface methods with a particular focus on the Eulerian mesh-based Volume of Fluid (VOF) technique. A range of 2D and 3D free surface approaches were initially investigated and validated using an experimental case with a simple geometry. The commercial solver Ansys Fluent and the CFD open source platform OpenFOAM were used to implement the VOF model and the DualSPHysics code was used to conduct simulations using the Lagrangian meshless particle-based Smoothed Particle Hydrodynamics (SPH) method. The hydraulic flow over a real hydraulic structure was subsequently modelled, applying the evaluated model implementations. The scheme consists of a newly constructed flood storage reservoir with a labyrinth weir and extended spillway. Different hydraulic conditions were modelled using a 1:25 physical scale hydraulic model of the prototype which was used to validate the numerical models. In order to remove numerical model uncertainties and provide insight into scale effects, numerical simulations were applied first to the physical scale hydraulic model and then to the full-scale prototype. Results show the model is capable of accurately predicting the interface features as well as the velocity and water depths measured in the physical model. It is observed that full-scale predictions present approximately a 17% increase in velocity and a 20% decrease in water depth compared to the equivalent scaled predictions.

Keywords: Computational Fluids Dynamics (CFD), Free-surface models, Volume-of-Fluid (VOF), Physical scale model

1 INTRODUCTION

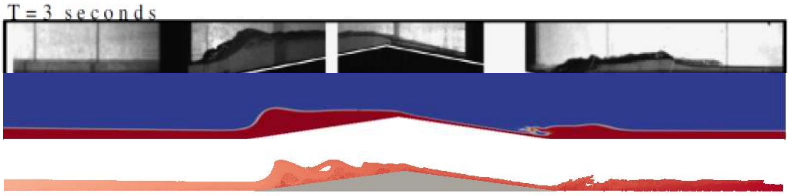
With increasing water demand and higher occurrence of extreme flooding events, the design and upgrade of hydraulic infrastructure like dams, weirs and spillways is critical for human safety and development. The current industry practice of hydraulic modelling for the design of this type of hydraulic structures is physical scale hydraulic modelling. Physical scale models are scaled representations of the prototype and are well trusted since they have for long been the conventional means to evaluate hydraulic designs. To achieve similarity between the prototype and the physical scale model, geometric, kinematic and dynamic similitudes need to be satisfied. That is, equal ratios of all length dimensions, velocities, accelerations and forces between prototype and model. However, because hydraulic models are not capable of satisfying the ratios of all forces between prototype and model simultaneously, the dominant forces affecting the flow for the type of problem (gravity, pressure, viscosity, etc.) are matched. For free surface flows the gravity effects are the most important and the Froude similarity is typically implemented. Therefore, scale effects in hydraulic models are inevitable since full dynamic similitude cannot be satisfied. When simulating free surface flows over hydraulic structures using the Froude similarity, the turbulence levels in the scale model are significantly lower, while the viscosity and surface tension effects are overestimated (Chanson, 2009). This causes the predictions of air entrainment in the physical model to be lower than in the full-scale. Scale effect challenges have been investigated in detail in some studies such as Erpicum et al. (2013) or Pfister and Chanson (2012) and limits on model upstream head, Reynolds number and Weber number have been established in order to minimise the effects of scaling.

Advances in computer processing power enabled dramatic improvements over the recent decades and a range of numerical approaches have been proposed to model free surface flows over complex structures. Such numerical models can provide detailed continuous data predictions of the relevant field quantities across the entire domain, which from a design perspective is very attractive. The additional information provided by CFD models provides richer understanding of the problem as well as the ability to simulate the full-scale prototype flow conditions which cannot be achieved by the physical scale models. However, these models have been validated only in a limited number of cases and flow conditions to model free surface flows over full-scale complex structures like weirs and spillways. 3D CFD models have a strong potential to provide accurate and flexible solutions with high prospects of time and cost savings in the design process. Further evidence of accurate numerical modelling of free surface flows over weirs and spillways is needed to demonstrate the reliability of such numerical approaches for specific applications. Developing a range of exemplar validated cases along with guidance on numerical aspects will provide a resource for those working in this area. One of the most well-known CFD models to simulate hydraulic free surface flows is the Eulerian grid-based Volume of Fluid (VOF) model by Hirt and Nichols (1981). The VOF employs the volume fraction function with values between zero and one to distinguish between the two fluids. The exact position of the interface is determined by solving an advection equation for the volume fraction function. This equation is solved using interface capturing schemes. Numerous formulations of the VOF model along with a range of discretisation schemes have been proposed. The VOF method has been successfully applied to model free surface flows in many problems, some examples are Oertel and Bung (2012) where the VOF model captured different types of flows generated in breaking waves; Biscarini et al. (2010) where the VOF model was used to reproduce several dam break flows, Hieu and Tanimoto (2006) who applied the VOF model to model wave-structure interactions and Borman et al. (2014) who found that VOF model could reliably predict shape and position of complex hydraulic jumps using validation data from a novel measurement approach. Flow over simple hydraulic structures has also been accurately simulated and validated with experimental data in studies like Sarker and Rhodes (2004)

In the present work, the capabilities of the VOF method to reproduce a real hydraulic structure are evaluated and utilised to provide insight into scale effects. In order to be able to validate a numerical technique, full dynamic similarity between the numerical model and the physical process modelled should be present. Therefore the physical model of a hydraulic structure is first simulated and the models are validated. Subsequently, modelling of the full-scale prototype is undertaken. Section 2 describes the validation study conducted to test the 2D and 3D VOF and SPH models for a simplified experimental geometry. Section 3 shows the application of the VOF model to simulate the flow over a large labyrinth weir and an extended spillway.

2 CFD VALIDATION STUDY

Initially the VOF and the SPH methods were investigated using a relatively simple dam break geometry, using data available in Biscarini et al. (2010). This is based on an experimental dam break case over a triangular obstacle set up in the laboratory. Figure 1 shows the experimental and numerical results at 3 seconds after the dam break.



Experimental results

VOF volume fraction contour plots

SPH free surface

Figure 1: Experimental measurements from Biscarini et al. (2010), 2D VOF water volume fraction and 2D SPH predictions 3 seconds after the dam break

The experiment involves the release of water which is able to flow over the top of the triangular obstacle. Part of the water reflects back upstream and the other part flows over the top of the obstacle and discharges into the second pool. 2D and 3D VOF simulations were conducted of the model set up using 6 meshes with different cell sizes ranging from 2×10^{-2} m to 1.5×10^{-3} m on the horizontal directions (x,y) and from 5×10^{-3} m to 4×10^{-4} m in the vertical direction (z). In the VOF simulations the Reynolds-Averaged Navier Stokes (RANS) equations were adopted to solve the effect of turbulence using the standard k-ε model with a standard wall function. The water free surface was resolved using an explicit VOF multiphase model available in Fluent and OpenFOAM to solve the transient free surface flow. The interface capturing scheme implemented was geometric reconstruction using a PLIC (Piecewise Linear Interface Construction) approach in Fluent and the multi-dimensional limiter for explicit solution (MULES) algorithm in OpenFOAM (which incorporates a compressive term in the transport equation of the volume fraction similarly to the Compressive Interface Capturing Scheme for Arbitrary Meshes (CICSAM) method). The pressure-velocity algorithm used was PISO. 2D and 3D SPH simulations were performed with particle spacing ranging from 2.5×10^{-3} m to 5×10^{-4} m using the Symplectic time step algorithm, laminar + SPS viscosity treatment and Cubic Splines. Sensitivity analyses in respect to different Fluent model implementations were performed in order to decide optimal model settings. These included sensitivity to turbulence model, different multiphase model and interface tracking scheme in the

VOF model. Sensitivity analyses were also conducted for the SPH simulations in respect to the time step algorithm, the viscosity treatment and the Kernel definition. The main conclusions of this CFD validation study can be summarised as follows:

- The 2D and 3D VOF predictions using Fluent and OpenFOAM (with fixed time stepping) accurately reproduce the flow features and the free surface depths measured in the experiment. The use of adaptive time stepping in the 2D and 3D VOF models provides accurate results in OpenFOAM;
- A mesh with cell size $1x10^{-2}$ m (x, y) by $2.5x10^{-3}$ m (z) with a fixed time step size of $1x10^{-3}$ s is considered to be appropriate for the dam break case modelled and the dimensions of the domain (5.6 x 0.5 x 0.1 m);
- The sensitivity analyses show no significant changes in the free surface predictions when using the SST k-ω and the standard k-ε turbulence models;
- The model shows comparable results with the implementation of two different interface capturing schemes (PLIC and CICSAM);
- The use of the Fluent Eulerian-Eulerian multiphase model significantly improved the slight flow delay observed in Fluent when using adaptive time stepping and also presents an accurate capture of the flow behavior;
- 2D SPH model using a particle spacing value (dp) of $1x10^{-3}$ m provides an acceptable estimation of the flow characteristics and free surface. Numerical predictions were not found to be sensitive to viscosity treatment or kernel definition but they were strongly dependent on the time step algorithm. The Symplectic algorithm is recommended for this type of problem. 3D results present satisfactory representation of the interface and flow features for a particle spacing value of $5x10^{-3}$ m.

3 CFD MODELLING OF A REAL HYDRAULIC STRUCTURE

3.1 Methodology

Using the numerical implementations from the initial validation study, VOF simulations of a real hydraulic structure were undertaken. The scheme is composed of an embankment dam with a labyrinth weir and extended spillway to allow controlled discharge. The spillway channel has a length of approximately 150 m. The width of the labyrinth weir is 32 m which is the widest part of the spillway. 75 m downstream the weir the spillway channel narrows down to 20 m and increases in gradient. There are approximately 9 m of steep channel and then there is a further change in gradient to become gentler and constant until the end sill. Thus, the spillway has three different gradients along the channel which together with the complexity of the labyrinth weir makes it a challenging geometry to mesh and conduct CFD simulations on. In addition, the existence of a road embankment immediately downstream of the spillway creates an impoundment for the discharged water which will reflect in different levels of tail water created behind the road embankment. A hydraulic jump is expected at each of the different water levels. In order to confirm the structure design and inspect the hydraulic characteristics, hydraulic modelling is undertaken. A 1:25 physical scale hydraulic model was built based on Froude similarity. The experiments included 8 different flow rates with three different levels of tail water each. The layout of the scheme and a photograph of the physical scale hydraulic model of the spillway and surrounding terrain are shown on Figure 2.

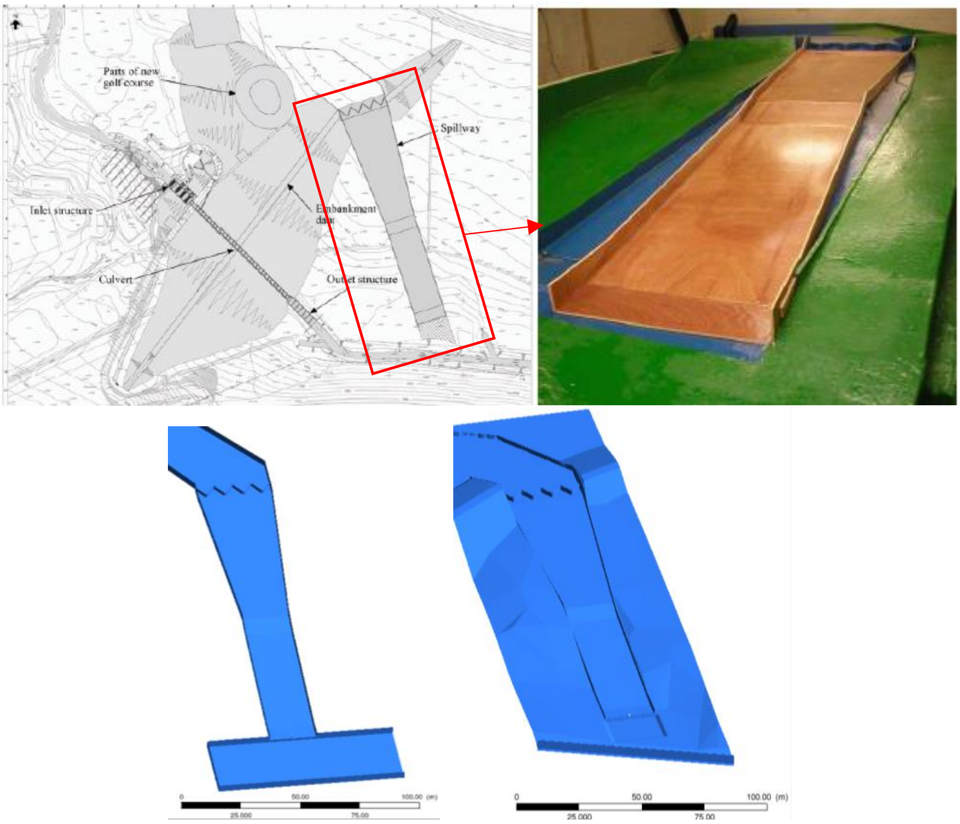


Figure 2: Layout of the flood storage reservoir with labyrinth weir and spillway (top left) from Brinded (2014) and model of the weir and spillway including surrounding terrain (top right). Intermediate and comprehensive geometries of spillway (bottom left and right)

The creation of the modelling geometry was possible given the availability of the full CAD drawings which include the detailed topography of the surrounding terrain. The approach taken to model this complex case is to first extract an intermediate geometry comprising the spillway and labyrinth weir only. The model is implemented and refined using the intermediate geometry with measurements of water depths and velocities in the spillway from the physical scale model. Decisions are then made on design cell size which will inform the meshing of the more comprehensive geometry. Modelling the comprehensive geometry allows the simulation of different tail water level conditions and an assessment of the capability of the model to predict the characteristics of the hydraulic jump. The creation of the modelling geometries is done using the Civil 3D toolbox in AutoCAD and the hexahedral meshing of the geometry is performed with the Ansys Workbench meshing application. Figure 2 shows the intermediate and the comprehensive geometries created for the CFD study. In order to capture impacts of scaling and appropriately validate the model, numerical modelling is first applied to the laboratory scale model and then to the full-scale prototype. CFD simulations are conducted in both Ansys Fluent and OpenFOAM to allow for solver comparison. Numerical results presented are primarily from the intermediate geometry and therefore the validation of the model is based on the flow behaviour within the spillway.

3.2 Scaled Intermediate Geometry

The physical scale hydraulic model was scaled using Froude number similarity, where the Froude number of the physical scale model and that of the prototype are the same. The model scale is 1:25 and by geometric similitude, the length ratio is equal to the model scale as shown in Eq. [1], where L_m is the characteristic lenght in the model and L_p the characteristic length in the prototype. Eq. [2] shows the velocity equivalence, where v_m is the water velocity in the model and v_p is the flow velocity in the prototype. Eq. [3] shows the flow rate correlation, where Q_m is the flow rate in the model and Q_p is the flow rate in the prototype. The time equivalence is shown in Eq. [4] where T_m is the time in the model and T_p is the real time.

$$S = \frac{L_p}{L_m} \qquad [1]$$

$$v_p = v_m\sqrt{S} \qquad [2]$$

$$Q_p = Q_m S^{5/2} \qquad [3]$$

$$T_p = T_m \sqrt{S} \qquad [4]$$

The intermediate geometry (originally constructed to be full-scale) was scaled down to the physical model dimensions (i.e. 25 times smaller), to run scaled CFD simulations. Numerical simulations for the scaled model have complete dynamic similitude and hence all the force ratios in the numerical model coincide with those in the hydraulic physical model. Mesh independence was investigated with the creation of three hexahedral meshes of average cell size 0.02 m, 0.008 m and 0.004 m with base inflation. These meshes had 645,397, 3.3 million and 6.7 million elements respectively. The mesh with cell size 0.008 m with base inflation (0.2 m with inflation in full-scale) was found to be of appropriate resolution for this case and numerical predictions did not present substantial changes with further refinement. VOF simulations were conducted in both Ansys Fluent version 14.5.7 and in OpenFOAM version 3.0.0. The RANS equations were solved using the k-ε turbulence model using a standard wall function and the PISO algorithm was employed to solve velocity-pressure coupling in both solvers. The volume fraction function was solved using the MULES scheme in OpenFOAM and the PLIC approach in Fluent. Time step was fixed in Fluent and equal to 0.001 s and adaptive in OpenFOAM with a restriction of the CFL number to 0.1.

The flow rate of 40 m^3/s was scaled accordingly and simulations were run using the design mesh (0.008 m with base inflation) in both solvers. A constant flow rate was implemented and run until an effective steady state was achieved. The same approach was implemented in the physical scale model experiments.

Figure 3 shows the experimental free surface and the numerically predicted free surface once the models reached steady state. Numerical results present accurate capturing of the complex configuration of cross waves generated by the labyrinth weir in both solvers.

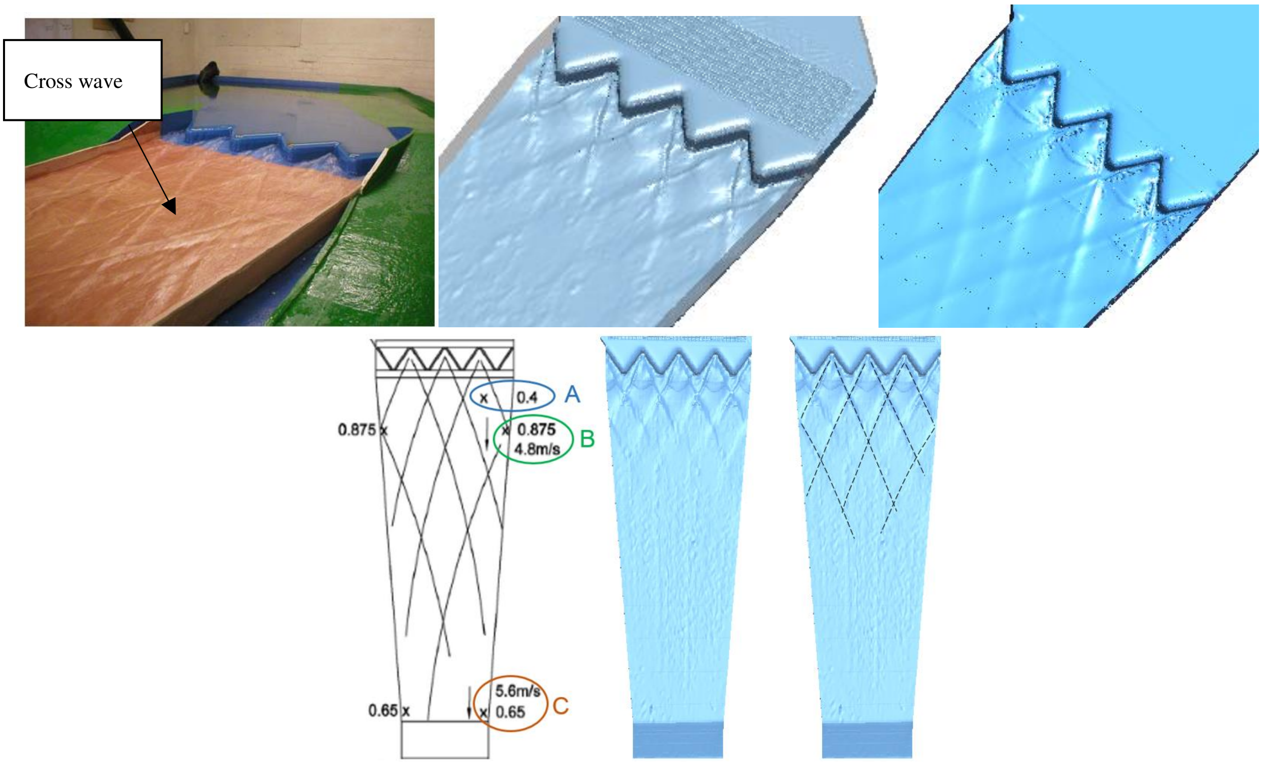


Figure 3: Free surface cross waves pattern in the experiment (top left), numerically predicted with OpenFOAM (top centre) and Fluent (top right). Measurement locations (bottom left) and free surface predictions with the drawing of the cross waves observed in the experiment and predicted by the model for a scaled flow rate of 40 m^3/s

Point measurements of velocity and depth are available at several locations outlined in Figure 3. Experimental measurements presented are the highest values of depth and velocity measured in the experiment and have an error of ±0.025 m and ±0.05 m/s in the prototype respectively.

A full time history of the simulation is presented in Figure 4 which shows that the steady state is reached after around 90 s. Figure 4 shows the plot of experimental values of water depth along with OpenFOAM and Fluent predictions at location A and velocity magnitude predictions at locations B and C. The contours of volume fraction of the water phase are plotted on a plane perpendicular to the flow at the same coordinate point as where the numerical results are plotted. These are presented at the three locations with a line indicating the point where the numerical predictions were extracted and plotted on the time series graphs. Point A is located in between cross wave crests (as shown in Figure 3) immediately downstream of the labyrinth weir. There is good agreement between the numerical results predicted using OpenFOAM and the experimental measurements. Fluent results present slightly lower values of water depth to those from OpenFOAM and are also in reasonable agreement with values recorded in the experiment. It should also be noted that the maximum water depth was recorded in the experiments.

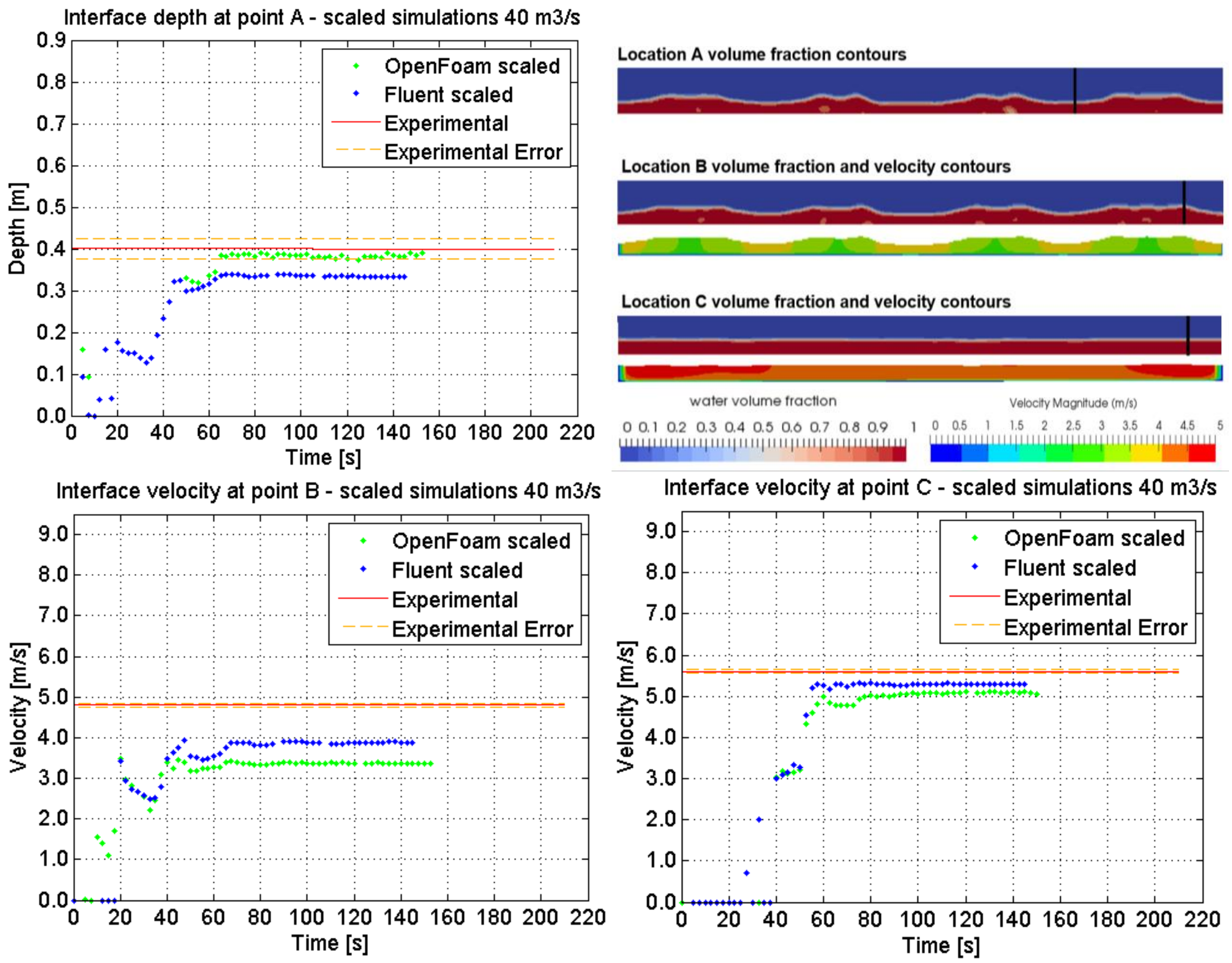


Figure 4: Experimental measurements and numerical predictions of water depth at location A (top left) and velocity magnitude at locations B and C (bottom) against time for a flow rate of $40 m^3/s$. Cross sectional profile of water volume fraction and velocity contours at a plane passing through the three locations across the width of the channel (top right)

Location of experimental points B and C is shown in Figure 3. Point B coincides with the crest of one of the cross waves and point C is located just before the first change in gradient. Overall numerical predictions at the interface appear acceptably close to the maximum values of velocity measured in the experiment. In addition, the water volume fraction contours show a very accurate capturing of the flow features. The water depth and velocity predictions from Fluent and OpenFOAM are overall comparable.

Point C is located next to the spillway wall at a point where there is a change in gradient. At this location the cross waves have settled and the water free surface profile shows a constant depth across the spillway. This situation is consistent in the numerical and the physical models as shown in the velocity graphs and in the volume fraction contour plots

Numerical results of depth and velocity at location C calculated with both solvers are slightly lower than the maximum values of experimental measurements. The presence of flat free surface allows a simple calculation of a mass balance at location C to check the reliability of the numerical predictions. Eq. 5 can be employed to determine the flow rate at location C:

$$Q = A \cdot v \qquad [5]$$

Where Q is the flow rate, A is the flow area and v is the average velocity. Using the OpenFOAM numerical predictions at location C, for a width of the channel of 20 m at point C, the flow rate is calculated as per Eq. 6:

$$Q = 0.4m \cdot 20m \ \cdot 5\frac{m}{s} = 40\frac{m^3}{s} \qquad [6]$$

Which is equal to the flow rate established in the inlet and hence conservation of mass is obeyed. Therefore, this calculation highlights the value of conducting numerical simulations to aid structure design and the need to inspect scale and measurement effects of the physical scale hydraulic models.

3.3 Full-Scale Intermediate Geometry

In the full-scale analysis, the exact same three meshes previously used in the scaled modelling were employed to model the real size structure. These had 0.5 m, 0.2 m and 0.1 m average cell size with a base inflation layer, which generated cell sizes of 0.1 m, 0.06 m and 0.04m on the z direction for each mesh respectively. Simulations using mesh with cell size 0.1 m indicate that there are only very minor differences to those using mesh with cell size 0.2 m which was therefore chosen as design mesh. The same simulations conducted in the scaled geometry using a flow rate of 40 m^3/s were conducted in full-scale using Fluent and OpenFOAM. It should be noted that the mesh used, for each solver, at the different scales has the same total number of cells when run at the two scales (it is a scaled version of the same mesh). Figure 5 shows the OpenFOAM and Fluent scaled and full-scale predictions of interface features and interface velocity contours once the model reached steady state using the design mesh. The free surface patterns appear to be different for the scaled and the full-scale simulations, presenting differences in shape and length. The crossing point of the cross waves is located at approximately 1 m further downstream in the full-scale cases compared to the scaled ones. This is shown in both solvers and indicated with an arrow on the free surface plots in Figure 5. Also, the full-scale waves structures are wider than the scaled ones. Figure 5 also shows the velocities are higher in the full-scale than in the scaled simulations, and this situation occurs in the results computed using the two CFD packages although is more pronounced in OpenFOAM.

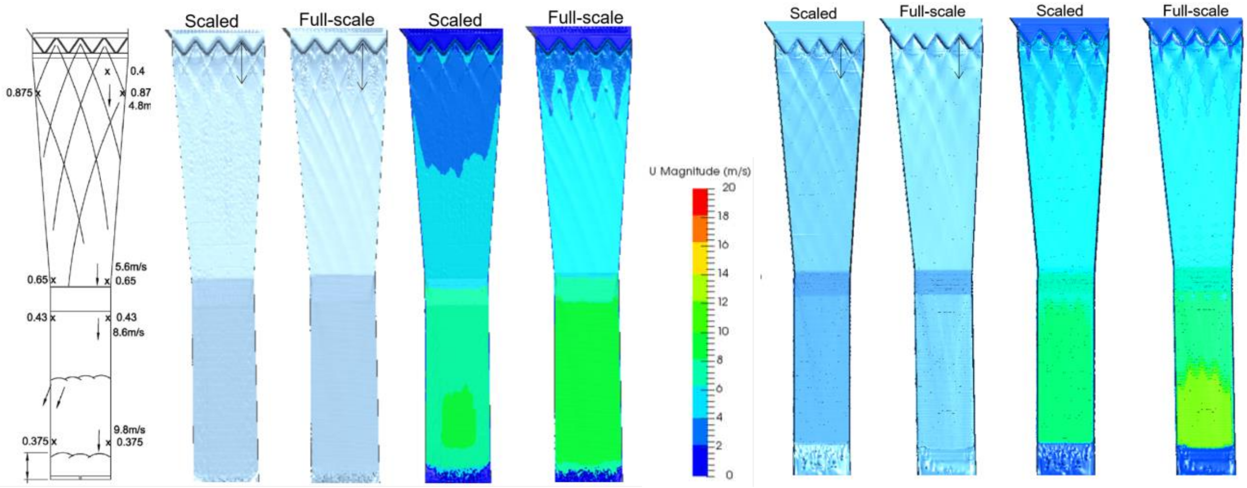


Figure 5*:* Physical scale model results (left), OpenFOAM (left of the legend) and Fluent (right of legend) predictions of the wave profiles and velocity contours at the water surface for a scaled and full-scale flow rate of 40 m^3/s

Data from the full-scale simulations were plotted at the same locations as previously outlined with the scaled simulations and also at locations D and E, as shown in Figure 6. A graph showing the steady-state values of interface depth and velocity of the scaled and full-scale OpenFOAM simulations at the different measurement locations is shown in Figure 6. It is observed that scaled results present higher water depth and lower velocities than their full-scale equivalent. This implies the full-scale flow expected in the prototype is shallower and faster than that predicted in the physical scale model in all locations.

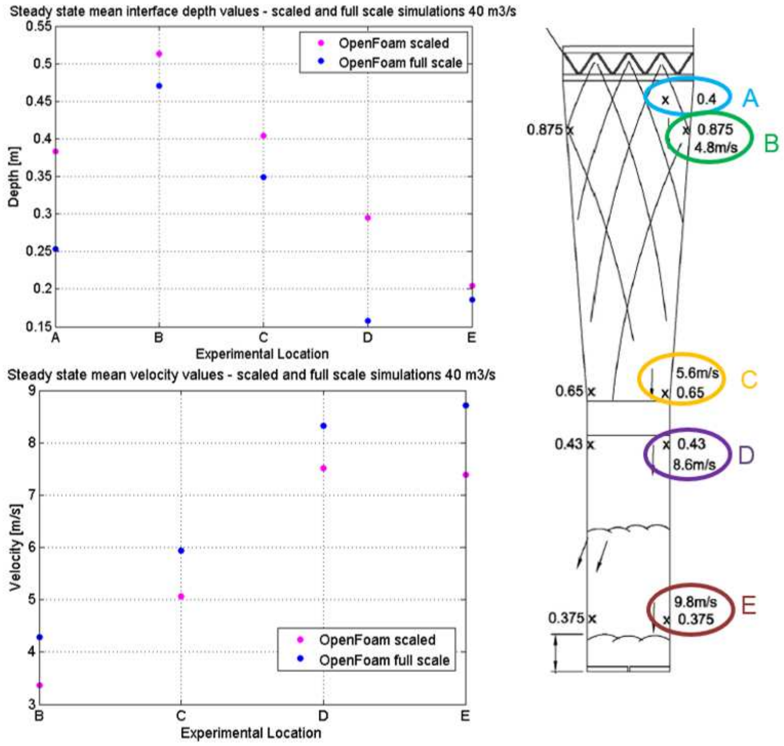


Figure 6: Scaled and full-scale OpenFOAM predictions of water depth (top) and velocity (bottom) and experimental loacations on the spillway for a flow rate of 40m^3/s

The scaled-down results present higher water depths and lower velocities than their equivalent full-scale results. It is calculated that full-scale results present an increase in velocity of approximately 17% and the decrease in water depth is estimated to be around 20% compared to the scaled results. This provides an indication as to the likely inaccuracies that could arise when using the physical model to estimate the full-scale flow behaviour.

3.4 Full-Scale Comprehensive Geometry

Preliminary VOF simulations with a relatively coarse mesh were conducted on the full-scale comprehensive geometry of the domain in Fluent and OpenFOAM. The main purpose of these initial simulations was to confirm the extracted geometry could be successfully meshed with acceptable quality for the VOF model to run in the two CFD codes (given the irregularity of the surrounding terrain and associate complexity of the geometry). A second purpose in conducting the initial VOF simulations using a preliminary coarse mesh was to be able to extract depth measurements to inform the process of establishing realistic downstream boundary conditions such that tail water levels could be maintained as in the experiments. Initial results (using a mesh with no inflation of 0.4 m cell size) show very realistic estimates of flow characteristics as well as reasonable agreement between Fluent and OpenFOAM predictions. Figure 7 shows the physical scale model in operation for a flow of 159.5 m^3/s with low tail water level and the equivalent simulation results of free surface characteristics and velocity contours predicted using OpenFOAM. Results with higher resolution of the comprehensive domain will be discussed in detail in the IAHR World Congress as will results from a full range of flow rates.

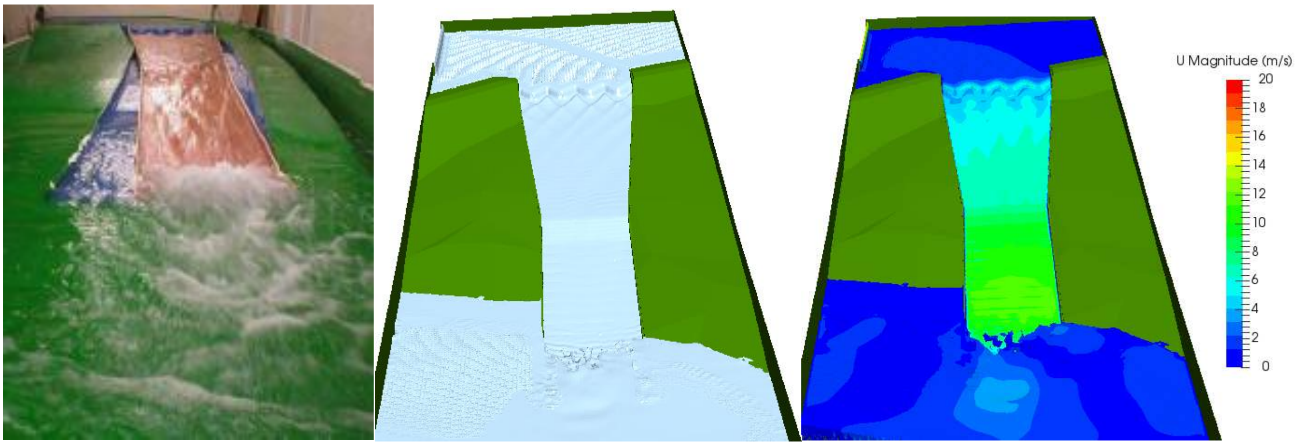


Figure 7: Photograph of the physical scale hydraulic model for a probable maximum flow of 159.5 m^3/s with a low tail water level and equivalent preliminary simulation results of free surface characteristics and velocity contours

4 INITIAL CONCLUSIONS

This study forms part of a larger on-going work which is being conducted at the present time on evaluating the ability of VOF-CFD approaches to be used effectively for design of large-scale hydraulic infrastructure to identify best-practice and to better understand the limitations. A summary of key findings with the work conducted to-date can be outlined as follows:

- OpenFOAM and Fluent VOF-CFD simulations applied to a scaled spillway geometry using a mesh of cell size 0.008 m show overall accurate predictions for water velocity, depth and wave patterns observed in experiments. The complex configuration of the cross waves generated by the labyrinth weir is well reproduced even in the mesh of the lowest resolution both in scaled and full-scale simulations. The interface features become very well defined when using the meshes of intermediate resolution (0.008 m scaled and 0.2 m full-scale) and are in strong agreement with the experiments;

- In a comparison between full-scale and scaled simulations it is observed that for the same mesh the scaled simulations predict higher depths and lower velocities than the equivalent full-scale simulation. The increase in velocity when modelling full-scale is estimated to be around 17% and the decrease in water depth is approximately 20% for a flow rate of 40m^3/s. In addition, the free surface profile between these present significant differences in shape and size (when comparing the full-scale simulations and scaled ones). This study highlights the importance and value in conducting CFD simulations to help understand any scaling differences that may be observed in a full-scale design compared with that observed in the physical scale hydraulic model. A detailed analysis investigating the scale effects of the hydraulic model is currently being performed and will be presented in the near future. However, results to date demonstrate that unlike the physical scale models, full-scale CFD simulations allow the prediction of many full-scale flow conditions which will be occurring at the prototype scale;

- A successful workflow and methodology has been developed to obtain the complex comprehensive modelling domain. An appropriate meshing strategy for the geometry has also been determined and successfully evaluated. Simulations using a reduced resolution mesh on a geometry that include the entire modelling domain have been undertaken in Fluent and OpenFOAM with the currently available results appearing promising. Results from a fully refined comprehensive mesh will be presented at the conference.

REFERENCES

Biscarini, C., Di Francesco, S. & Manciola, P. 2010. CFD modelling approach for dam break flow studies. Hydrology and Earth System Sciences, 14, 705-718.

Borman, D., Sleigh, A., Coughtrie, A. & Horton, L. 2014. Hydraulic free surface modelling with a novel validation approach. 9th South African Conference on Computational and Applied Mechanics. Somerset West.

Brinded, P., Gilbert, R., Kelham, P. & Peters, A. Eller Beck Flood Storage Reservoir – the challenges of low impact flood storage design. In: Pepper, A., ed. 18th Biennial Conference of the British Dam Society at Queen's University, 2014 Belfast. ICE Publishing.

Chanson, H. 2009. Current knowledge in hydraulic jumps and related phenomena. A survey of experimental results. European Journal of Mechanics - B/Fluids, 28, 191-210.

Erpicum, S., Silvestri, A., Dewals, B., Archambeau, P. & Pirotton, M. Escoloubre Piano Key weir: Prototype versus scale models. In: AL., E. E., ed. Second international workshop on labyrinth and piano key weirs, 2013 Chatou, Paris, France.

Hieu, P. D. & Tanimoto, K. 2006. Verification of a VOF-based two-phase flow model for wave breaking and wave–structure interactions. Ocean Engineering, 33, 1565-1588.

Hirt, C. W. & Nichols, B. D. 1981. VOF method for the dynamics of free boundaries. Journal of Computational Physics, 39, 201-225.

Oertel, M. & Bung, D. B. 2012. Initial stage of two-dimensional dam-break waves: laboratory versus VOF. Journal of Hydraulic Research, 50, 89-97.

Pfister, M. & Chanson, H. 2012. Scale effects in physical hydraulic engineering models. Journal of Hydraulic Research, 50, 244-246.

Sarker, M. A. & Rhodes, D. G. 2004. Calculation of free-surface profile over a rectangular broad-crested weir. Flow Measurement and Instrumentation, 15, 215-219.