Article:

Chesterton, OJ, Ucuncu, M and Borman, D orcid.org/0000-0002-8421-2582 (2019) CFD modelling for dams and reservoirs – best practice workflows, specification and review. Dams and Reservoirs, 29 (4). pp. 148-157. ISSN 1368-1494

https://doi.org/10.1680/jdare.19.00032

Dams and Reservoirs Journal

Working Title: CFD modelling for Dams and Reservoirs - best practice workflows, specification and review

Authors:

Full name	Affiliation	Contact number	email
Owen John Chesterton	Mott MacDonald	+1 669 249 9598	John.chesterton@mottmac.com
Mutlu Ucuncu	Arup	+44 20 7755 4017	mutlu.ucuncu@arup.com
Duncan Borman	University of Leeds	+44 113 343 2354	d.j.borman@leeds.ac.uk

INTRODUCTION

Need for wider understanding

During the most recent British Dam Society (BDS) 2018 Conference a workshop was held to discuss Computational Fluid Dynamics modelling and its application to the Dams industry. The workshop gave an opportunity for a number of practitioners involved in both numerical and physical modelling to demonstrate the state of play and then encouraged discussion on the challenges in applying these tools to the dams industry. The majority of the attendees were not themselves practitioners but included All Reservoirs Panel Engineers (ARPEs) and others responsible for dam safety. Those in attendance highlighted the need for guidance when specifying or reviewing modelling that is not as readily accessible as a physical modelling laboratory, to help the non-CFD expert to judge the adequacy of the model to support the conclusions drawn from it.

Some of the discussions focused on the capabilities of CFD and discussed subjects such as air entrainment, Y+ and mesh sensitivity and turbulence models. More difficult to address were questions pertaining to knowing when to select CFD or Physical modelling, the accuracy of numerical models, their legal status in court, where to go for guidance and what to expect in the future of this fast-moving area.

To go some way in filling these needs, this article outlines typical Computational Fluid Dynamics best practice including workflows, specifying modelling work and expected outcomes. This includes some general discussion on the pitfalls and benefits of this type of modelling.

What is Computational Fluid Dynamics (CFD)

Computational fluid dynamics is a branch of fluid mechanics that uses numerical analysis and algorithms to obtain approximate solutions for problems related to motion of fluids. CFD modelling techniques typically involve dividing a domain into a grid or mesh and involve finding a numerical solution (e.g. for pressure, velocity, water depth) throughout the domain. For time dependent problems the simulation is progressed by solving the governing equations at incremental time steps.

CFD approaches that use a numerical grid or mesh are typically referred to as mesh based approaches (the most common in CFD is the Finite Volume Method). Differences between various commercial codes used within the Dams and Reservoir industry typically relate to the methods for discretising and then solving the governing equations, but also in the choice of sub-models that are required to capture the specific physics of a given problem. Selecting appropriate modelling assumptions for given flows is a critical part of the CFD modelling process and each assumption will have associated limitations that need to be carefully considered for any given problem.

Alternative, meshless, CFD methods include particle simulations, such as the Smoothed Particle Hydrodynamics (SPH) method which considers the trajectories and interactions of large numbers of particles and does not require a computational mesh.

The codes focused on for this article are predominantly mesh-based and those that currently have the capability to reliably predict the location of a free water surface. Codes most commonly in use at the time of writing this article included ANSYS Fluent and CFX, Siemens StarCCM++, Flow Science Flow3D, OpenFOAM and several others.

Need for industry specific guidance

CFD modelling is now being routinely used to inform decisions relating to Dam Safety. In the UK, dam owners and independent engineers appointed to make statutory recommendations in the interests of public safety are relying on CFD outputs and predictions. While many industry guidance documents have been produced for empirical physical modelling, CFD's application to Dams and Reservoirs is new and is arguably less accessible to the non-CFD expert, less understood and lacking in general guidance.

In 2002 the UKs Health and Safety Laboratory (HSE) recognised a similar gap and published a guide for HSE inspectors to guide assessments by non-CFD experts ([1] Goveau, Ledin & Lea, 2002). The goal of this article is similar (although less comprehensive) and uses this guidance as a starting point. The article is intended to set the groundwork for a similar industry specific guidance document for industry professionals and those responsible for dam safety. It is hoped that this article will begin to allow owners and engineers to assess the quality of the CFD model and help them determine when more complete expert advice is needed.

For practitioners, much practical guidance is available such as the ERCOFTAC best practice guidelines ([2] Casy, Wintergerste, & ERCOFTAC, 2000) or online wikis and guidance such as CFDonline, although many of these may appear to be unkept. For more in-depth reading there are good CFD textbooks that are freely available for reference such as 'Computational Methods for Fluid Dynamics'([3] Ferziger & Peric, 2002).

CFD MODEL WORKFLOW

Prior to starting the CFD modelling process, it is important that the modeller has a good understanding of the problem to be solved and the objectives of the modelling to be undertaken. These two dictate the nature of the further steps of the CFD analysis process flow.

Typical steps required for the CFD modelling process are described

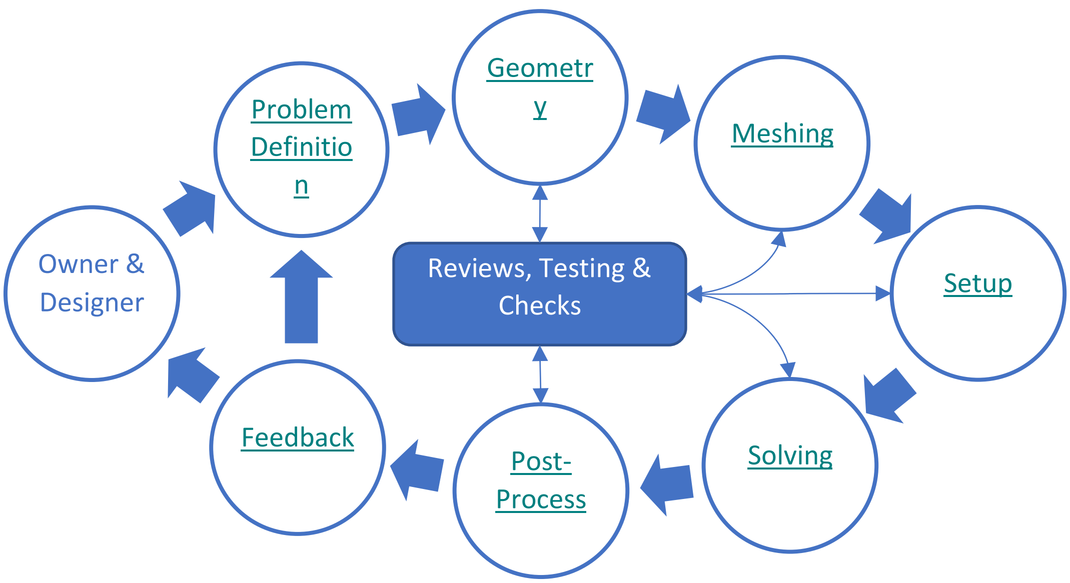
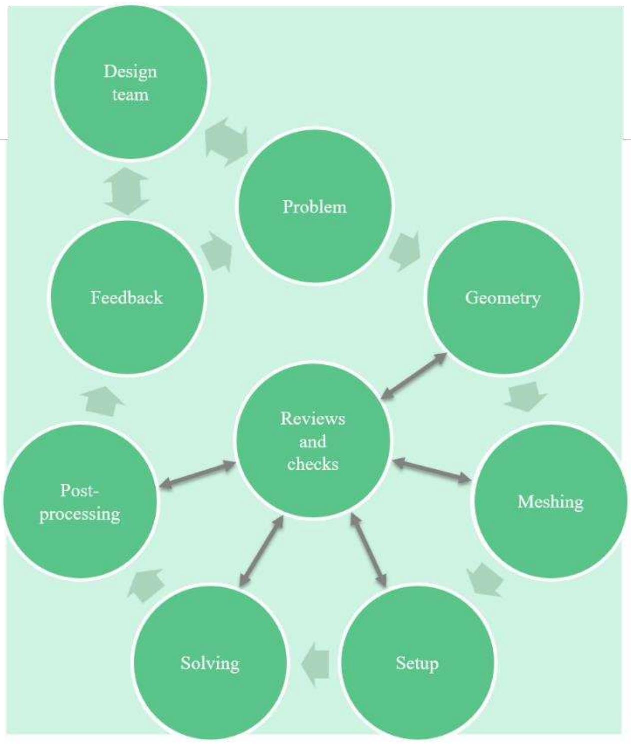


Figure 1 and in the sections below.



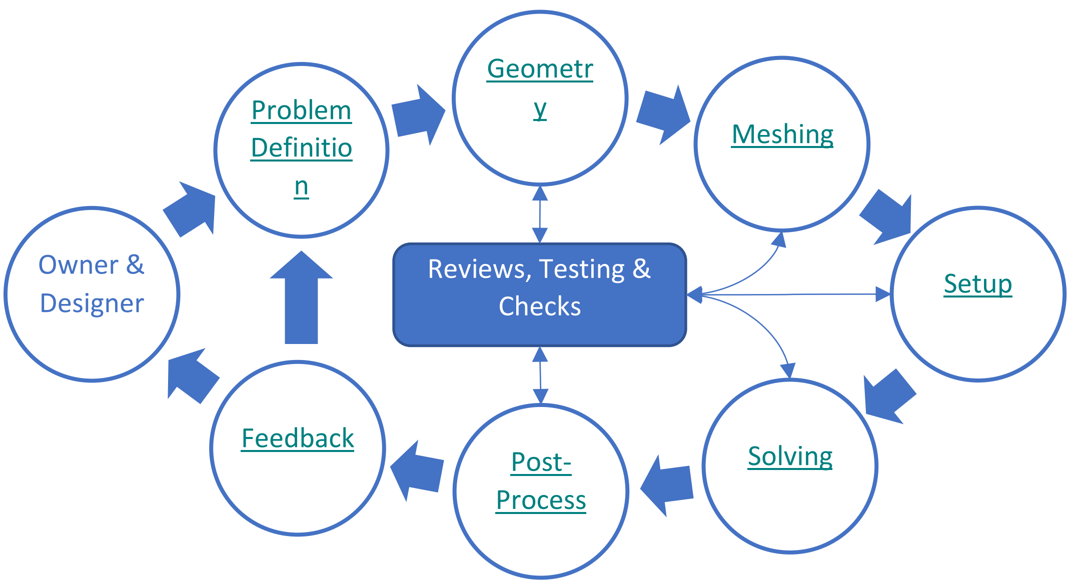


Figure 1: CFD Workflow

Problem definition

Modelling projects will often start with a clear statement of purpose outlining the key objectives of the project and the model runs that are intended to provide the solution. This may be partially covered by the specification, but any additional inputs would typically be provided and vetted in a kick-off meeting with the owner, panel engineer, external reviewer and designers.

An early check should always be carried out using basic hydraulic equations and formula to determine whether modelling is necessary and what the general flow characteristics may be. A sense check on the modelling method, whether physical or numerical, should identify a change in approach if this was appropriate.

Model geometry

The first step to build the CFD model is constructing the 3D representation of the structure or components. This is typically done in a 3D modelling package such as Autodesk's Civil3D, Microstation, or other 3D design software.

Geometries developed for most dam and reservoir hydraulics will necessarily be 3D. Reliance on 2D simulations may remove 3D flow patterns that may affect the outcome (in some situations, 2D studies may be useful to provide insight prior to a full 3D simulation). The level of detail provided within the 3D model is often high, as minor changes in geometry such sharp corners instead of curved, oversimplification of bridge piers or weir steps can lead to quite different results.

Development of the model geometry may be from a design model or may be acquired through traditional survey or laser scans. Most models will include data from a range of sources all with different levels of accuracy. The extent of the simulation domain should be chosen in a way that the domain boundaries are sufficiently away from the areas of interest, so that the boundary effects do not influence the flow properties in these areas. For example, for a spillway modelling CFD study, the inlet boundary in the upstream reservoir should be sufficiently away from the spillway weir so that the simulation predicts natural flow over the weir, rather than the inlet boundary imposing flow velocities in this region which could result in incorrect weir coefficients. Poor downstream boundary conditions in an open channel may dictate the flow regime and profiles, introducing features such as hydraulic jumps or incorrectly predicting their location and strength.

In many cases, it may be required to do geometrical simplifications and approximations to manage the size of the model and the associated computational cost. For example, geometrical details of the surface roughness features are not practical to include in the CFD model explicitly. Walls with such features are typically included in the model as a network of simplified flat surfaces, and the roughness effect of these walls are represented implicitly by means of numerical approximations (i.e. rough wall boundary condition with specified roughness height).

Figure 2: Spillway geometry showing high level of detail and a simplified but adequate geometry

One of the main advantages of CFD models when compared with physical models is that CFD models can easily be constructed in 1:1 scale of the real structure, so that the dynamic similarity is achieved, and problems associated with scaling are avoided. CFD model geometries at 1:1 scale should be used unless the objective of the analysis is to compare CFD modelling results directly with the model scale physical test results.

Computational mesh

Once the geometry has been defined the fluid domain is then subdivided into cells or control volumes to create a 3D computational mesh using a specialised software programme or one of the larger commercial CFD software packages that have this capability inbuilt.

In order to capture all the physical phenomena in question, a sufficiently high resolution in the areas where large gradients in the flow variables are expected is required. For example, in open channel and pipe flow simulations, the computational mesh next to the wall should be more refined enabling greater resolution to capture the rapidly changing flow variables near to the wall, such as velocity.

The accuracy of the solution and the stability of the simulation are also highly dependent on the quality of the mesh - it is not only a measure of mesh resolution. Mesh quality measures depend on how the equations of fluid motion are approximated (discretised) and applied in the CFD software. Typical mesh quality metrics to watch are cell aspect ratio, non-orthogonality and skewness.

Building a high-quality mesh requires the consideration of:

- mesh resolution requirements to capture regions of rapid variation in the solution, such as near wall regions, hydraulic jumps, shear layers, etc.
- mesh resolution requirements of physical sub-models used, such as turbulence, air entrainment, sediment transport, etc.
- mesh quality metrics appropriate for the numerical evaluation method used.

Ultimately the CFD modeller should aim to achieve a mesh-independent solution, in other words the accuracy of the solution should no longer be influenced by further mesh refinements. Typically, the first set of simulations are carried out using a coarser mesh and subsequent simulations are carried out on finer mesh, until the CFD predictions do not differ significantly with further mesh refinement.

For most dams and reservoir simulations the scales are large and the purpose of simulation and turn-over time need to be considered. Efficiency is gained by concentrating the efforts on mesh quality and resolution in the area of interest, while lower mesh resolution is applied in the far field regions.

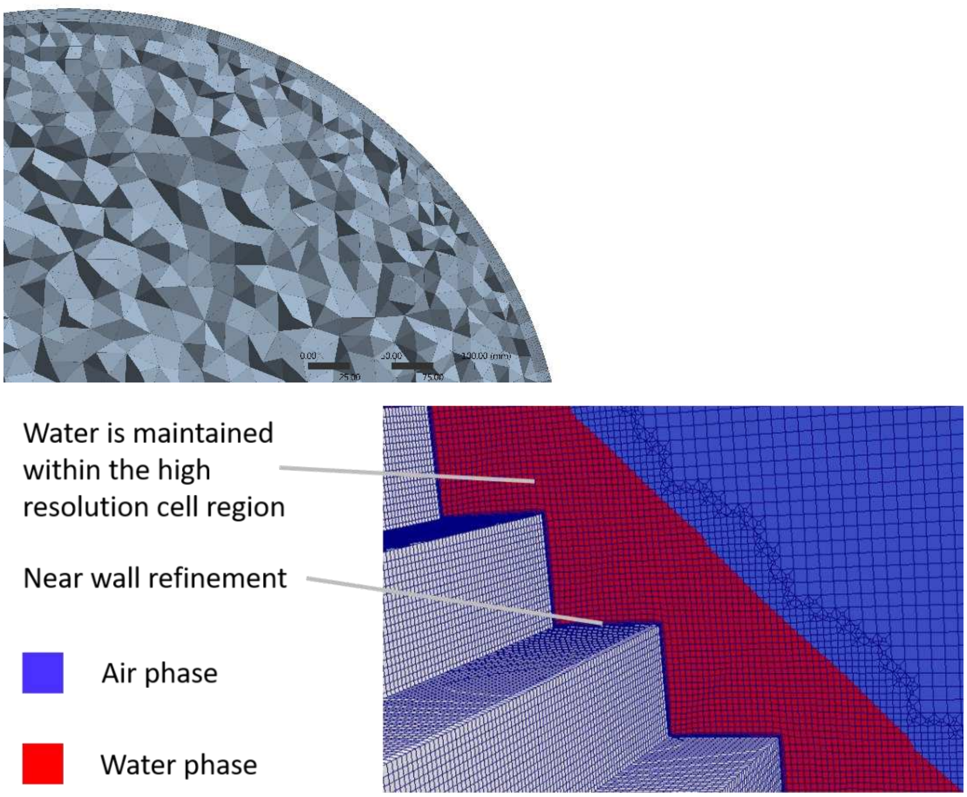


Figure 3 Surface mesh on the spillway steps and the computational mesh over the slice through the flow domain

Model setup

This is the stage where the physics of the problem is defined. Depending on the problem being tackled, and the software chosen, the CFD modeller needs to choose a set of appropriate mathematical models and simplifications. Specification of the following features of the simulations is crucial at the outset of any CFD modelling study related to hydraulic structures:

- **Steady-state vs. transient flow:** Steady-state models may be applicable if the flow characteristics do not change in time (i.e. assessment of water residence time in a service reservoir with fixed inlet and outlet positions and flow rates). Transient models should be used for unsteady or highly turbulent flow problems. Running a naturally transient problem in steady-state mode will lead to inaccurate solutions.
- **Single vs. multiphase flow:** Decision on single vs multiphase flow depends on the type of problem we are dealing with. Single phase model (water only) is appropriate for pipe flow type problems while open channel flows are likely to require a multiphase model with an interface tracking model used to define the free surface.
- **Laminar vs. turbulent flow:** For the flows under consideration in dams and reservoirs industry, we expect significant turbulent content to the flow. Turbulence exists at a wide range of scales, with large vortices decaying progressively into smaller ones, until the turbulent energy is dissipated due to the viscosity of the fluid. Modelling all these scales explicitly requires Direct Numerical Simulation (DNS) which is a highly computationally expensive method and not routinely used. The two main alternative methods for modelling of turbulence are Reynolds-Averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) methods.
  - Typical RANS simulations solve for the turbulent content of the flow by introducing new flow variables to represent turbulent kinetic energy and turbulence dissipation. This provides a statistically averaged representation of the turbulent flow.
  - LES modelling approach resolves the transient, three-dimensional flow features larger than the characteristic computational grid scale, with robust models used for turbulent features smaller than this scale. It has great potential as it enables larger scale turbulent structures to be explicitly resolved, rather than modelled as in RANS. However, even with its significant potential, LES has not yet replaced RANS, mainly due to the following factors:
    - Even with the significant increases in computing power that have come about, it is still computationally very expensive to perform LES on a routine basis.
    - LES is yet to reach a level of maturity that users without substantial experience and expertise can obtain results with the necessary level of solution fidelity and reliability.
- **Isothermal vs non-isothermal flow:** The temperature variations within the water typically have negligible impact on water and properties in flows under consideration in dams and reservoirs industry.
- **Physical properties:** The physical properties such as density and viscosity of the fluids under consideration and environmental factors such as gravitational constant are defined at this stage.

- **Boundary conditions:** Boundary conditions are used to constrain the simulation domain by a known set of conditions and flow properties.

  Wall boundaries are used to represent the solid surfaces that are in contact with the fluid (water and air). No-slip boundary condition which assumes that the velocities reach zero at the walls is applied by default to the wall boundaries. Surface roughness properties can be defined for turbulent flows to represent the effect of roughness features on the flow properties. Due to the large gradient of flow variables near the wall, it is computationally very expensive to explicitly resolve all the flow features in the near wall regions in most of the engineering applications. A common approach is to employ wall functions, which are empirical equations used to model the near wall regions without resolving the boundary layer. The CFD modeller should ensure that the near wall region mesh satisfies the requirements of the wall function, i.e. the dimensionless distance (y+) of the first cell from the wall is within the appropriate range for the wall function used.

  The boundaries that permit the flow to enter or exit the simulation domain (inlet, outlet and pressure type boundaries) should be located sufficiently far away from the regions of interest so that the boundary effects do not have an influence on the flow behaviour in these areas. For example, for a reservoir spillway model, the boundary defining the incoming flow into the reservoir should be far from the approach to the spillway so that the flow velocities in this region are naturally developed, rather than being enforced by the inlet boundary.

- **Initial conditions:** For a transient simulation, defining an appropriate and approximate starting solution of the whole simulation domain obtained by inputs from traditional first principle calculations can significantly reduce the required simulation time to reach the final solution. An example of this can be given as initialising the upstream reservoir level of a spillway modelling simulation by the predictions from an earlier flood study. In some cases, the end result of a transient simulation is influenced by the initial conditions.

- **Discretisation schemes:** The CFD modeller should choose higher order discretisation schemes where possible in order to minimise the numerical errors (as appropriate, a sensitivity study can be undertaken to understand the impact of using alternatives schemes). Appropriate Courant number considerations should be taken for time step size selection in transient simulations. The Courant number describes the number of cells through which the flow information can be transmitted within a time step.

Obtaining reliable solutions

Once the model is set up, initial runs and testing can be carried out. This includes testing the model sensitivity to mesh resolution as discussed earlier, or checking for errors as explained below.

The accuracy of a CFD simulation is influenced by several sources of error:

- **Modelling errors:** These errors are the differences between the real physics of the flow and the mathematical representation used in describing the flow. For example, turbulence models do not solve the full spectrum of the flow physics, but approximate the effect of turbulence on the flow properties. The difference between the real flow and the model predictions is referred as modelling errors.

- **Numerical errors:** The differences between the CFD predictions and the exact solution of the governing differential equations of the fluid motion are called numerical errors. One of the most commonly seen numerical errors is numerical diffusion, which is the over-prediction of diffusion of mass, momentum or heat.
- **Convergence (or iterative) errors:** CFD software uses iterative solution algorithms to solve the equations of fluid dynamics. Convergence errors occur when this process is stopped before the results are sufficiently close to the final solution, and the CFD modeller should define appropriate convergence criteria to control these.
- **User Errors:** Human factors play a large role in the reliability of the results as the experience of the user is critical. Errors can easily result from a lack of attention to detail, sloppiness, mistakes and blunders. Overly optimistic and uncritical use of CFD due to easily accessible or free codes and compelling visuals can also lead to error. Inexperience and unfamiliarity with particular CFD codes and unfamiliar hydraulics can also lead to errors that the modeller may not be aware of.
- **Other error types:** Other common errors in CFD modelling are problem definition errors (e.g. due to too much simplification in domain geometry), code errors (i.e. bugs in the CFD software) and round-off errors (due to limited number of computer digits available).

All these different sources of errors necessitate that the CFD modeller should take a series of quality assurance actions to ensure increased confidence in the simulations prior to interpretation of the results and before deriving key conclusions and to make important design decisions.

1. **CFD code verification:** Verification in the context of CFD modelling is the process of checking that the mathematical equations are implemented and solved correctly in the CFD code. Ideally the code used in the CFD study is well recognised in industry and academy, with a good track record for the area of application. The verification of such CFD codes have typically been done and recorded by the code developer.
2. **Model validation and calibration:** Validation is done by comparing the CFD predictions against the experimental data or real site measurements. For many CFD modelling studies, experimental data or site measurements is not available. In these cases, it is recommended to refer to the validation studies done on other similar hydraulic structures or to use standard hydraulic formula, such as equations for open channel flow for comparison.

   In some cases, validation studies may not lead to a satisfactory conclusion. In these situations, it may be necessary to calibrate some modelling parameters (model calibration) to improve the agreement between CFD predictions and the measured data. However, calibration of the model to the validation data should be reviewed carefully as it can be easy to inadvertently introduce errors in an effort to improve agreement with the validation data. Where models don't match, this can sometimes be explained by scale errors in the validation data or inherent limitations in the CFD code and shouldn't be hidden.
3. **Monitoring convergence:** It is important to set appropriate convergence criteria and monitor the convergence throughout the run to control convergence errors. It is difficult to estimate how large the current convergence error is as the final solution is unknown. A broad range of approaches can be used in conjunction to ascertain that a model is sufficiently converged. The

CFD modeller monitors residual error and global imbalances of mass, momentum and energy to assess if a satisfactory level of convergence is achieved. The convergence checks may also include monitoring key predicted flow variables in critical aspects of domain to see if these have stabilised. The particular approach taken to judging/confirming convergence will depend upon the particular model implemented.

4. **Model reviews:** In addition to the steps described above, appropriate levels of model reviews carried out by CFD analysts should be undertaken to make sure that user errors are eliminated.

Post-processing and reporting

The final stage of the CFD modelling process is analysis and interpretation of the model predictions. A variety of flow characteristics can be obtained from the solution of the CFD model by means of extracting, deriving and integration, depending on the objective of the analysis. Some of the typical outputs sought in dam and reservoir applications are flow velocities and water levels in a spillway channel, features of a hydraulic jump, discharge capacity vs. upstream water level curve of a weir, energy dissipation performance of stepped spillways, pressure and bed shear forces on various components of hydraulic structures, identification of cross waves or locations of water overtopping the channel walls and many other flow characteristics.

Interpretation of the simulation results requires an appropriate level of theoretical and practical knowledge of the subject as well as modelling experience. The results should be assessed with the consideration of the potential impacts of assumptions made and limitations of the CFD analysis. CFD modelling reports should include relevant caveats while discussing the outputs and conclusions of the study.

SPECIFYING AND REVIEWING CFD MODELS

The specification and procurement of hydraulic models has often relied heavily on input from the laboratory carrying out what was typically a physical model, and has relied on the laboratory for quality assurance.

As CFD modelling can be done almost anywhere, this lowers the barriers to entry and increases the variability in approach, outputs and quality of the work produced. In this environment it becomes more important to adequately specify the qualifications and quality of outputs expected.

Specifying a hydraulic model is not always a straightforward task and may not be written by a modelling expert. The specification should avoid being overly prescriptive and allow for the CFD modeller to utilise their expert advice in responding to the requirements of the CFD model.

This article aims to provide a starting point for reviewers and contains recommendations on which CFD modelling specifications can be based.

Expertise of the CFD modeller

The expertise of the CFD modeller should be shown in two demonstrated categories. Firstly, a thorough knowledge of the appropriate area of dams and reservoir hydraulics including design such as open channel weirs, spillway and energy dissipation structure hydraulics or intake and conveyance hydraulics. This should be shown through project experience or training. Secondly, the CFD modeller should also provide demonstrated expertise in use of the CFD code, which would ideally be evidenced by previous project examples and published peer-reviewed work which would then give confidence in the modeller's capability.

Those without demonstrated experience in CFD modelling may not fully appreciate the limitations of the software, while CFD practitioners with no experience in dam and reservoir hydraulics may be unaware of the physics required or the set-up and meshing appropriate for the model. Of course, even if expertise is demonstrated in both areas, budget and time constraints often lead to over-simplification and lower resolution models that can often be under conservative or simply less accurate.

Specification Recommendation: The specification should request a named model lead along with evidence of relevant project or published experience in both CFD modelling and applicable relevant project experience in dam and reservoir hydraulics.

CFD code selection

The code the modeller or modelling team intend to utilise should be stated up front. While there are many codes in use today, they are complex and impossible for a reviewer to thoroughly check for each project. The code selected should be a commercially recognised code with a range of validated dam and reservoir related test cases. If an unrecognised code is proposed, the specifier may require industry standard validation cases to be provided or run prior to acceptance of this code.

Specification Recommendation: CFD code employed must be validated for cases or hydraulics similar to that being modelled. It is reasonable to require the modeller to show published examples, validations and other best practice guidance to support this.

Geometry and boundary conditions

Any specification should detail the level of accuracy required and where (Typically +/- 10mm for hydraulic control surfaces such as weir crests). For existing structures, the modeller should have visited the site before modelling to appreciate the issues that may not show in the survey and good photographic records should be available for examination by a reviewer.

The level of detail captured within a CFD model geometry is likely to be similar to that required from detailed or 90% design. The reviewer should be aware of the design stage and how the results will be used. Later refinements to the design may invalidate the CFD model in ways that may not be immediately obvious.

Specification Recommendation: Model geometries should be 3D with a level of detail and accuracy appropriate to the results and conclusions being drawn. The source (survey, bathometry, lidar, site measurements) and accuracy of the geometry derived from these should be reported. Model geometry files and drawings with key dimensions and any photographs of as-built geometries should be made available for review if requested. A site visit should be included for the design team and key modeller.

It is critical that the reviewer understands and agrees with the boundary and initial conditions. Virtual space is not limited but as a larger domain is more expensive to model in time and cost, there is pressure to constrain the model as much as possible. Incorrect boundary conditions will not always be apparent or visible in the model outputs and require skill and experience to implement correctly. The modeller should fully report their assumptions and be able to provide the boundary condition information along with any sensitivity testing or other calculations when requested.

In the same way, initial conditions can dictate model results and should be selected to ensure that the results reflect reality.

Specification Recommendation: Model boundary type, locations and assumptions should be reported, and assumptions justified. Boundary conditions may require testing to justify location or type and the results of such tests provided to the reviewer on request.

Computational mesh

Specifications should not dictate the type or application of the computational grid but instead leave its design to the experienced modeller. Through the course of the project the modeller will determine the most appropriate mesh based on their understanding of the key flow phenomena, experience or research of similar cases and testing. While many software packages offer automatic meshing and even adaptive meshing (mesh that is automatically refined with iteration) the modeller should always inspect and verify that the mesh is appropriate.

When reviewing visual outputs of the results, the mesh is typically not shown. The mesh should not be able to be inferred by inspection of just velocity or pressure plots, so where steps appear in what should be smooth outputs this should be queried. While the primary outputs may not display the mesh lines these should be reported on. Sections through areas of interest, such as through a weir or bellmouth intake or through a pipe would be shown, and will identify areas of high pressure or velocity gradients that require careful mesh application.

While the geometry may be made within a 3D modelling package the geometry only defines the domain that will be meshed. Dividing this domain into mesh cells introduces facets on what would be smooth curves, and a view of the meshed domain may show problems such as changes to the wetted area or model facets creating separation where flow would otherwise follow a smooth curving surface.

In general, where flow variables like velocity or pressure change rapidly requiring tight contour values to describe them, the mesh resolution should increase accordingly to capture these gradients.

Some pragmatism may be needed in the application of the mesh and some simplifications may be justified where there are smaller flow features that will not affect the overall results. Some flow features that are particularly difficult to resolve are jets, flow separation points, spray and boundary layers, which are all very sensitive to mesh resolution.

The model report should include sections showing mesh lines in areas of most interest and reports on the shape, number of the cells within the model and the range of cell sizes. Typically, it would be good practice to report additional details of mesh sizes in key locations, for example on a stepped spillway it may be appropriate to report details of the number of elements within each step cavity.

Specification Recommendation: Computational mesh should be applied and checked by an experienced modeller and tests carried out to ensure mesh independence and reported or evidence provided of similar studies. Tests should be carried out using the key outputs the model is expected to report as a sensitivity parameter.

Physics

Compared to other fields, the range of physics encountered in dam and reservoir modelling is usually limited to water flows and pressures and the interactions between water, air and the control structures in question. More complicated flows may require additional models to be activated within the software package and most codes have a range of standard models such as surface tension, bubble models and water surface models (VOF) that can be activated.

There are certain 'physics' that are more common place and that CFD can be expected to predict well, whereas there are also 'physics' (e.g. flows with large amounts of air entrainment/flow bulking) which are more challenging due to the complex physical interactions and may involve larger uncertainties in model predictions.

The model specification should indicate what general flow conditions and outputs are most desired as these will indicate the physics models that should be employed. The specification could include statements such as, 'the model should define the location of the water surface and effectiveness of the hydraulic jump basin', or maybe 'the model should be used to identify flow separation and potential for cavitation and air demand in the tunnel'. Physics models the specification may ask the modeller to consider could include, turbulence, free surface, air movement, surface tension, bubble models, air entrainment, sediment transport, pollutant or tracer transport and concentration, moving bodies.

Specification Recommendation: The modeller should propose the use of physics models within the CFD code to address the objectives of the modelling project and to provide the required outputs. Adequate documentation supporting these decisions and the limitations of the models should be reported and made available for review where required.

Scenarios

The number of scenarios that are modelled may include varying flow, geometry or different operational cases such as gates followed by modifications to the geometry once any issues are known. While physical models can run many flows relatively quickly, measurements need to be taken then and there. Numerical models can be batched, but the run itself can take quite some time and longer runs, if they are needed, can be expensive. The flow rates and flow rate combinations that may be required would typically include PMF, 10,000yr and 1,000yr flood flows for spillways and numbers and combinations of pumps or gates that could be in operation.

As pointed out in ICOLD Bulletin 172 ([4] ICOLD, 2016), derivation of a rating curve for a spillway can take a large number of runs and care should be taken to ensure that all relevant features on the curve are captured. It is unlikely that the model can be run to simulate the full hydrograph time series. However, some hydraulic systems show hysteresis where differences between capacity and performance are observed between the rising and falling limb of the hydrograph, so care should be taken when setting up initial conditions to account for this.

Typically, once the modeller has been engaged and has built the geometry, an early meeting with the owner should be organised to discuss the scenario matrix that will be used and any changes to what had originally been proposed can be agreed.

Specification Recommendation: The following assessment [eg. Rating curve, water levels, velocities, stilling basin performance etc] are required to satisfy the regulatory or operational outputs necessary. Early meetings should be included to review the set-up and discuss any additional model runs that may be required as identified by the modeller, owner, Panel Engineer or Independent Reviewer. This will be followed by [number] of additional meetings to view early results or changes. Rates and timeframes for additional model runs and associated reporting should be provided to provide for additional runs to be added if needed.

Accuracy, validation and verification

The accuracy of any model is critical to those investing in the modelling exercise and even more so for those relying on its results for their safety. The accuracy of any model must be shown by both verifying that that the model is working as expected and through validation against real world results as discussed earlier.

However, the term 'accuracy' should be carefully applied and interpreted if used. For those specifying models, where validation data is available (an equivalent physical model) a margin of error may be requested (say +/- 5% of physical model water levels). The margin of error should be selected with care as over specification can lead to needless cost. A good understanding of the validation data accuracy is required (measurement methodology, model scale, prototype conditions etc.) and the modeller should be able to demonstrate a good understanding of the physical model or prototype data they are calibrating against, including the instrumentation and method of physical model data capture and reliability.

Lack of computational power or resolution should not be an excuse for inaccurate results and a CFD modeller should recommend or conclude that physical modelling is required if hydraulics is not able to be resolved as soon as this becomes apparent. Quality assurance should be carried out during modelling to review the modellers assumptions and make sure that the accuracy the modeller is claiming is correctly interpreted. Finally, model data should be summarised in a form useful to the reviewer and following modellers looking to reproduce results.

Specification Recommendation: The following validation data is available for review [previous physical model studies, photographs of prototype performance, manufacturer data etc]. Outputs shall show agreement with prototype/validation data to within [eg. -/+ 5%, 250mm or other parameter] and outliers or areas of divergence due to model effects should be discussed and explained.

A robust quality assurance process should be outlined, executed and documented and produced if requested. This should include model logs, check sheets and evidence of internal review of the model inputs and model approvals by other CFD analysts, in addition to model report checks and approvals. A summary of the model inputs and assumptions should be included as an aid to future reproducibility.

Analysis and outputs

Specify the 1D data required such as flow rates, water levels or hydrodynamic loading. Additional outputs may include cross sections (2D plots) and 3D visualisations which can be useful in understanding general performance. When undertaking specific assessments, limited outputs may be the most efficient but do not help the reviewer understand the model set-up or catch problems that may be just ‘off-screen’. A full set of outputs is better as would be typical from both a physical or numerical model and may be useful to answer questions that may arise that were not in the original scope. Specifying the outputs required is important as they may not be provided if not requested from the outset.

For dam spillways, outputs would typically include, graphed outputs of spillway capacity, flow depths, velocity, energy and Froude number. These would typically include graphed flow depths against walls set against spillway station and elevation and include key cross sections. These outputs might be supplemented with direct visualisations of the model showing 2D velocity, pressure or vector fields. 3D outputs would include views of the flow surfaces, flow streamlines or other key variables and should include animations of the flow to help give the reviewer an appreciation for the structure’s performance.

Chapter 6 “User Error” of the ERCOFTAC Best Practice Guidelines ([2] Casy, Wintergerste, & ERCOFTAC, 2000) and Section 6.2.4 of the same document provides good checklists of outputs for model interpretation.

The type and extent of reporting can affect the cost of the modelling outputs. However, requesting a short form report may limit the ability of the modeller to convey the accuracy and testing undertaken, complexity of the hydraulics and may reduce the usefulness of the work to the client in the long term. Limited reporting can often result in remodelling in later phases. Reporting should include model interpretation and the modeller should use their experience to describe the flow phenomena, their origin and impact on structures performance. Anomalies in the results should be pointed out and discussed.

Specification Recommendation: The following data should be reported from the model: 1D tabulated or graphed data including [eg. Rating curves, forces, water depths, summary outputs], 2D graphed and data visualisations including [water level profiles, model sections showing contoured variables], 3D visualisations [both static and animated if possible] and any additional visualisations required to aid in the interpretation and understanding of the model results.

Reviews

CFD modelling of hydraulic structures is typically employed for the assessment of unusual designs or significant departures from standard, novel methods of analysis and involves considerable exercise of engineering judgement. As such, this article recommends independent review similar to a Category 3 review ([5] BSI, 2019) of any CFD model that is being used to inform decisions surrounding dam safety or to respond to recommendations on matters in the interest of safety.

This should be considered in addition to and in support of the review and oversight of the Panel Engineer.

Specification Recommendation: The CFD model report and outputs may be required to be provided for external peer review on completion. The model input files, mesh and geometry may be requested for this review, in addition to the reporting provided.

SUMMARY AND CONCLUSIONS

CFD for dams is no longer new but the quality still varies dramatically, and further guidance may still be needed. CFD modelling has not been subject to the same scrutiny as physical modelling but it is being relied upon more frequently to make decisions about matters in the interest of safety.

Engineers who would typically review CFD model reports need to understand the issues surrounding computational modelling or find supporting expertise to ensure they are obtaining relevant and accurate results. Reviewers need to be aware of what is being presented so as not to be blinded by unfamiliar science.

Those specifying CFD models should also familiarise themselves with CFD terminology and capabilities. While some reliance on modellers is needed to propose methodologies and approaches, more directed specifications will encourage comparable bids and quality of work.

Computational fluid dynamics may be even less transparent in years to come, but black box modelling should be treated with some scepticism and honest questions asked about any results that seem out of the ordinary. Despite its complex implementation, CFD modelling and its reporting should be aimed at communicating complex flows simply and effectively. Hydraulics of spillways and other often observed flows will remain intuitive and approachable and should be presented as such.

REFERENCES

[1] Goveau, N., Ledin, H. S., & Lea, C. J. (2002). Guidance for HSE Inspectors: Smoke movement in complex enclosed spaces-Assessment of Computational Fluid Dynamics. Health and Safety Laboratory.

[2] Casy, M., Wintergerste, T., & ERCOFTAC. (2000). ERCOFTAC Best Practice Guidelines: ERCOFTAC Special Interest Group on "quality and Trust in Industrial CFD". ERCOFTAC.

[3] Ferziger, J. H., & Peric, M. (2002). Computational Methods for Fluid Dynamics 3rd Ed. (3rd ed.). Berlin: Springer.

[4] ICOLD. (2016). Technical Advances in Spillway Design, Progress and Innovations from 1985 to 2015.

[5] BSI. (2019). BS 5975:2019 Code of practice for temporary works procedures and the permissible stress design of falsework.