OPTIMISE TANK DESIGN USING CFD

Paper Presented by:

Lisa Brown

Authors:

Lisa Brown, *General Manager*,
Franz Jacobsen, *Senior Water Engineer,*

Parsons Brinckerhoff

72nd Annual Water Industry Engineers and Operators' Conference
Bendigo Exhibition Centre
1 to 3 September, 2009

OPTIMISE TANK DESIGN USING CFD

Lisa Brown, *Senior Water Engineer*, Parsons Brinckerhoff, Melbourne, VIC
Franz Jacobsen, *Senior Water Engineer*, Parsons Brinckerhoff, Brisbane, QLD

ABSTRACT

The use of computational fluid dynamics (CFD) has proven useful in optimising the design of water storage tanks. In particular, the use of CFD helps to ensure that short-circuiting within the tank does not occur or is minimised, which is vital to maintain correct residence times and eliminate storage dead zones. CFD allows the modeller to manipulate inlet and outlet pipe positions and angles, add baffles or other innovative concepts and rapidly view the results. A skilled CFD practitioner can produce outputs that are highly visual, user friendly and simple to understand. The advantages of CFD are its flexibility, relatively short computational times and overall cost effectiveness. In addition to confirming tank design, CFD is frequently used to model pump wet-wells, drop structures, energy dissipation structures, spillways and weirs. CFD modelling is relatively inexpensive and has a proven track record for providing substantial capital cost savings and affording assurance of design. CFD is no longer seen as a daunting technology and is a cost effective addition to the design process.

KEY WORDS

CFD, short-circuiting, optimisation, tank design

1.0 INTRODUCTION

The importance of water storage tank circulation has not been a major concern in the past because chlorine was the dominant form of disinfection used, and chlorine is forgiving of circulation problems. Chloramines, on the other hand, break down through a nitrification process that can provide nutrients for bacteria if they are allowed to remain in dead spots. Chloramination is commonly used in water systems throughout Victoria, Australia and overseas, offering benefits over chlorine in certain situations. Water utilities moving to chloramine disinfectants face the challenge of evaluating and in many cases improving circulation within their water storage tanks. The concerns for adequate circulation are widespread because most existing reservoirs follow very simple and economical designs in which water enters and exits the tank either from a single inlet/outlet or separate inlet and outlet pipes that are positioned on opposing sides of the storage tank where site geometry allows.

For water authorities changing to chloramine disinfectants from chlorine, or simply experiencing problems with apparent short-circuiting, the first task would be to evaluate the circulation currently achieved within the tanks, to identify any dead zones or short-circuiting. In the past, the only method engineers had to evaluate reservoir circulation was to build a scale model and perform experiments. This approach is expensive and time-consuming and may provide inaccurate results in many cases, because of scaling factors and a very small number of point measurements. On the other hand, a CFD analysis provides fluid velocity, pressure and solute concentration values throughout the solution domain for problems with complex geometries and boundary conditions. As part of the analysis, an engineer may change the geometry of the system or the boundary conditions, such as inlet velocity or flow rate, and view the effect on fluid flow patterns or concentration distributions.

This paper discusses a CFD model that has been created to demonstrate the ability of CFD to easily identify any dead zones or short-circuiting. The model has been based on a typical water storage tank, constructed with a standard separate inlet and outlet arrangement. The model has been further tailored to show the change to circulation patterns able to be made by simple modifications to the inlet pipe, along with the inclusion of a straightforward baffle wall. Through viewing the output from the CFD modelling, the changes made to flow distribution within each scenario can be seen, thus enabling realistic optimisation of the tank design with minimal cost outlay.

2.0 DISCUSSION

It is crucial to ensure the water storage tanks preserve water quality, not degrade it, in order to maintain safe drinking water throughout the entire distribution system. There are three distinct hydraulic conditions that must be addressed to preserve water quality in distribution storage tanks:

- eliminating short-circuiting,
- achieving complete mixing, and
- turning over (fluctuating) the tank levels to minimise water age.

To prevent any water quality issues, all three need to be addressed. However, short-circuiting is seen to be most severe with common inlet/outlet pipes or when the inlet and outlet are in close proximity to each other. The last water put into the tank is the first water that is drawn out. Stagnant areas develop outside this area of influence and the water in these areas can be days or weeks old, resulting in the loss of disinfectant residual.

Locating the inlet and outlet pipes at opposite sides of the tank will lessen short-circuiting, but can also introduce large dead zones depending on inlet flow conditions. This trial CFD study was undertaken in order to consider the effectiveness of simple, cost effective solutions for short-circuiting, that can be implemented in both existing tanks, and incorporated into the design of new clear water storage tanks, to optimise the design from a short-circuiting perspective.

To gain an understanding of short-circuiting within storage tanks, three differing scenario CFD models were run. The results of these scenarios were used to indicate the impact that the configuration of the inlet and outlet pipes, along with any obstructions within the tank, has on circulation.

2.1 Description of Model Used

The CFD program used to undertake this study is called OpenFOAM. The OpenFOAM (Open Field Operation and Manipulation) CFD toolbox can simulate complex fluid flows involving chemical reactions, turbulence and heat transfer. OpenFOAM is produced by Open CFD Ltd and is freely available and open source, licensed under the GNU General Public Licence. Using open source software significantly reduces the cost of undertaking a CFD analysis, making it cost effective to employ on even the smallest scale projects.

Of course, before the choice of software can be made, confidence in the results obtained through the program must be proven through thorough checking and validation. Parsons Brinckerhoff has previously measured OpenFOAM's reliability by comparing results

obtained from OpenFOAM to results obtained from physical models and final construction assessment on a large scale pump station inlet project.
The results obtained from OpenFOAM, mirror those results achieved through all other methods of modelling employed on the project. Therefore, its use on small scale projects has been sanctioned in order to make CFD modelling an affordable design confidence tool.

OpenFOAM solves equations of fluid dynamics using a finite volume analysis. The model calculates the Navier-Stokes equations, which describe the motion of the fluid. These equations establish that changes in momentum in infinitesimal volumes of fluid are simply the sum of dissipative viscous forces (similar to friction). Changes in pressure, gravity, and other forces acting inside the fluid are calculated by the application of Newton's second law. The method used to solve for turbulence in the study models was the Large Eddy Simulation (LES) method. LES is a numerical technique used to solve the partial differential equations governing turbulent fluid flow.

The final mesh used in each of the scenarios consists of an unstructured grid. The mesh is more 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 final model mesh for Scenario 2 is represented in Figure 1.

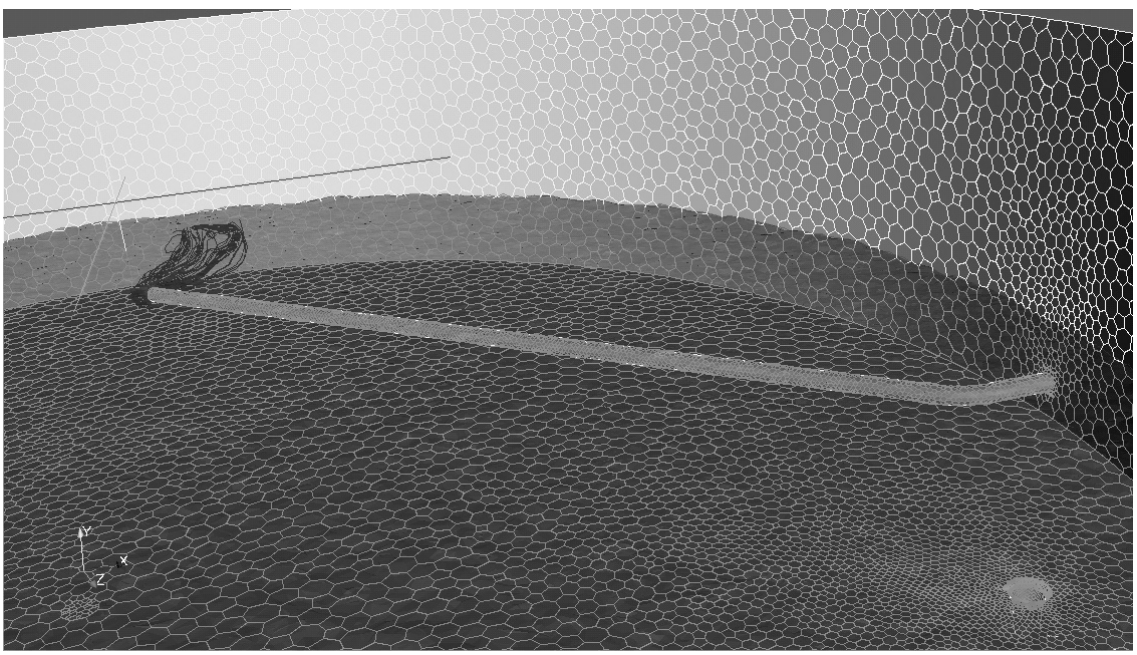

Figure 1: ***Final model mesh***

2.2 Model Scenarios

Three scenarios have been developed for this study to represent existing design practice, as well as options for altering the inlet pipe configuration and changing the internal geometry of the tank by installing a baffle wall. These options are designed to demonstrate the flexibility that CFD allows.

Scenario 1 – Base Model - Inlet and Outlet on Opposite Sides of Tank

The data used to define the base model layout was obtained from a standard water storage tank layout, incorporating the inlet and outlet pipes on opposing sides of the tank, with the inlet pipe penetrating through the side of the tank below bottom water level, and the outlet pipe comprising of a bellmouth outlet located slightly above the tank floor level. The cross section of the model replicates the full extent of the clear water storage, covering both the inlet and outlet pipe work configurations. The model also extends along the outlet pipe to 3 m beyond the bellmouth outlet. This arrangement ensures that flow characteristics around the entrance of the outlet pipe are realistically modelled.

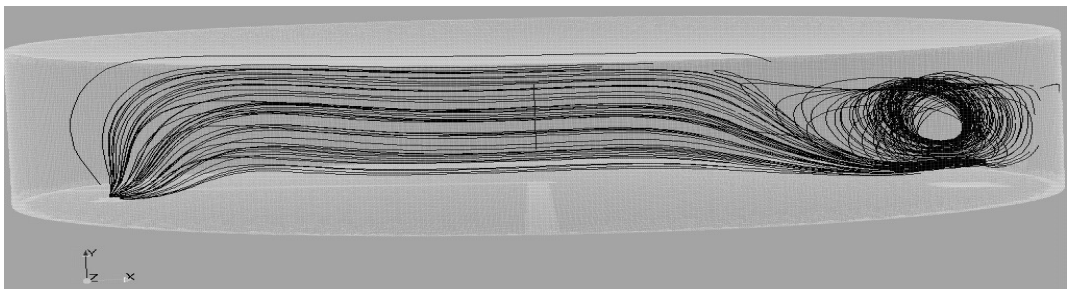

Figure 2: ***Scenario 1 – long section dye trace***

Figures 2 and 3 show the results of a dye trace simulation in the base model scenario from a section and plan perspective respectively. As can be seen by the circular motion of the fluid adjacent to the inlet pipe, areas of recirculation exist on either side of the end of the inlet pipe. This results in significantly differing flow path lengths from the inlet to the outlet pipe which indicates a potential for short-circuiting to occur under these flow conditions.

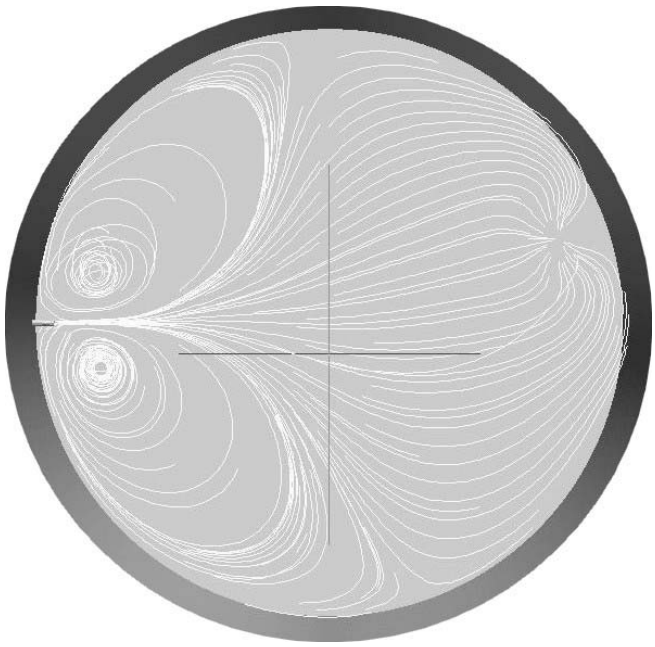

Figure 3: ***Scenario 1 – plan view dye trace***

Modifications to the inlet pipe were made in the second scenario to attempt to eliminate these recirculation zones and improve flow path consistency to reduce short-circuiting.

Scenario 2 – Modifications to the Inlet Pipe

In an effort in improve tank flow path times and reduce the degree of recirculation occurring, the second scenario considered the situation where additional pipe was added internally within the tank to direct flow away from an adjacent inlet and towards the opposite wall of the clear water storage tank. The additional pipe was positioned such to direct flow in a manner to encourage circulation around the tank periphery. Figure 4 shows, in plan view, the results of a dye trace simulation in the second model scenario.

Figure 4: ***Scenario 2 – plan view dye trace***

As evidenced by the larger circular motion of the fluid from the inlet pipe towards the outlet, flow paths in this scenario are more uniform with short-circuiting having been reduced. A small area of recirculation can be seen on the internal side of the inlet pipe, along with a very small dead-zone adjacent to the tank wall. These problems could be further reduced by reducing the length of the inlet pipe and positioning it such that the flow is tangential to the internal tank wall.

Scenario 3 – Inclusion of a Baffle Wall to alter the Tank Geometry

To further demonstrate the benefit of CFD modelling to optimise the design of clear water storage tanks, a third scenario was considered that included the installation of a straightforward baffle wall, along with a reduction in the length of the inlet pipe.

Figure 5 shows the resulting dye trace of the third model scenario. As can be seen by the flow patterns, short-circuiting has been further reduced with long, circumferential flow being evidenced throughout most of the tank. The exception to this is the region adjacent to the baffle wall.

The interference to the flow patterns around the baffle could be further optimised by adjusting the inlet pipe so that the flow from it is tangential to the internal wall of the tank and amending the baffle wall shape to follow the induced flow path. This could also serve to reduce the small region of flow recirculation immediately adjacent to the end of the baffle wall as demonstrated in Figure 5.

Figure 5: ***Scenario 3 –plan view dye trace***

3.0 CONCLUSION

CFD modelling has already demonstrated that it is a useful tool for engineers on complex hydraulic projects such as restricted pump station inlet arrangements, dam spillways and sewer drop structures. The use of open source CFD software, at a significantly reduced cost, is now making it possible for engineers to apply computational fluid dynamics to smaller scale projects to afford assurance in design. As demonstrated in this small scale study, multiple changes can be made to standard scenarios to model the effect of change without the need for expensive physical modelling or full scale testing. Optimisation of design can be achieved, or specific site obstructions considered, for even the smallest of capital works projects without blowing the budget. The opportunity for CFD to become an everyday tool for engineers is on hand. CFD is no longer seen as a daunting technology and is a cost effective addition to the design process.

4.0 REFERENCES

http://www.opencfd.co.uk/openfoam/