CHILDREN'S HOSPITAL CFD CHLORINATION TANK ANALYSIS

Paper Presented by:

Franz Jacobsen

Author:

Franz Jacobsen, *Principal Engineer and Owner,*

Engys AUST

38th Annual WIOA Qld Water Industry Operations Conference
Parklands, Gold Coast
4 June to 6 June, 2013

CHILDREN'S HOSPITAL CFD CHLORINATION TANK ANALYSIS

Franz Jacobsen, *Principal Engineer and Owner,* Engys AUST

ABSTRACT

Computational fluid dynamics (CFD) has been used successfully to optimise the design of water storage tanks. Potently these techniques identify possible short-circuiting and eliminate storage dead zones within the tank, which is conducive to achieving required residence times. In many instances a CFD model has been utilised using qualitative methods to study flow patterns which has not translated to improvements in the field.

An innovative low cost CFD modelling technique that incorporates 'passive scalars' allows the designer to introduces a tracer to the CFD model which results in quantitative performance indicators such as baffle factors and Residence Time Distribution (RTD) curves. For large un-baffled tanks the commonly used characteristic contact time (t_{10}) method may lead to misleading results regarding baffle factors. Alternative means of assessing tank performance and contact time will be discussed. In addition to confirming tank design, this new low cost CFD method is frequently used to model pump wet-wells, drop structures, energy dissipation structures, spillways and weirs.

1.0 INTRODUCTION

The new Queensland Children's Hospital in South Brisbane is currently under construction and forecast to open in late 2014. This hospital will bring together both the Royal Children's Hospital and the Mater Children's Hospital into one single facility.

The water storage system design of the hospital includes four (4) 160 m^3 storage tanks as per the Disaster & Emergency Function Post Disaster Facilities Recommendations. Security of water supply is essential for the Queensland Children's Hospital as a post disaster and emergency facility. It is important that there is no deterioration in water quality as a result of the extended water detention time and potential risk this intermediate storage system will impose. Mixing is required within the storage system to promote water circulation and prevent dead zones, water stagnation and stratification, which will result in the degradation of water quality (consumption of disinfectant/chlorine residual).

2.0 DISCUSSION

During the original assessment, the tank configuration and flow patterns were assessed via CFD modelling by the author whilst employee at AECOM. Two options were investigated to improve the flow paths. On the basis of the flow path visualisation (qualitative assessment) and engineering judgement, it was recommended to adopt option 2. All three options are shown in Figure 1. Only tank 4 was modelled as it was considered to have the worst case configuration.

The layout for the base case consists of the inlet and outlet simply placed at close proximity. For option 1 the inlet was raised within the tank and the inlet was rotated to point towards the opposite corner. Option 2 consists of a 1m long perforated diffuser pipe, also elevated within the tank.

For the purpose of this paper, the original work, carried out in 2011 was reassessed by the author using a quantitative method.

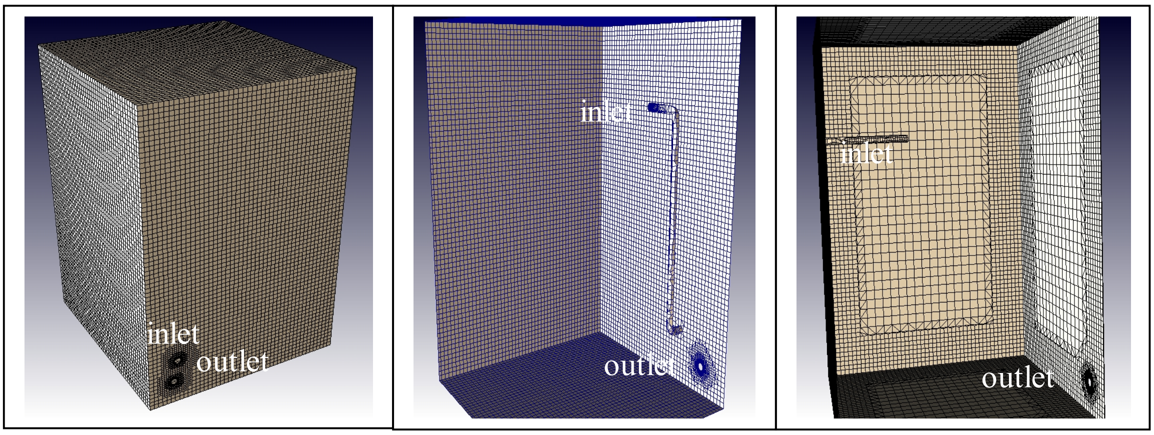

Figure 1: ***Typical Layouts, Base Case (left), Option 1(middle), Option 2 (right)***

This was carried out by plotting Residence Time Distribution curves (RTD) derived using the method of passive scalars, which can be described as similar to adding a trace to the CFD model. This method confirmed the original recommendation of adopting option 2.

2.0 CFD MODELLING

A CFD model was used to analyse flow paths within the tanks, with the aim of identifying recirculation, dead zones and stratification. The domain of the initial CFD model is shown in Figure 2 which includes all four tanks and associated pipework. However, it was found that the most optimal CFD representation was to base the modelling domain on Tank 4 only, which represents the most conservative case with potentially the most hydraulically adverse arrangement due to close proximity of the inlet and outlet.

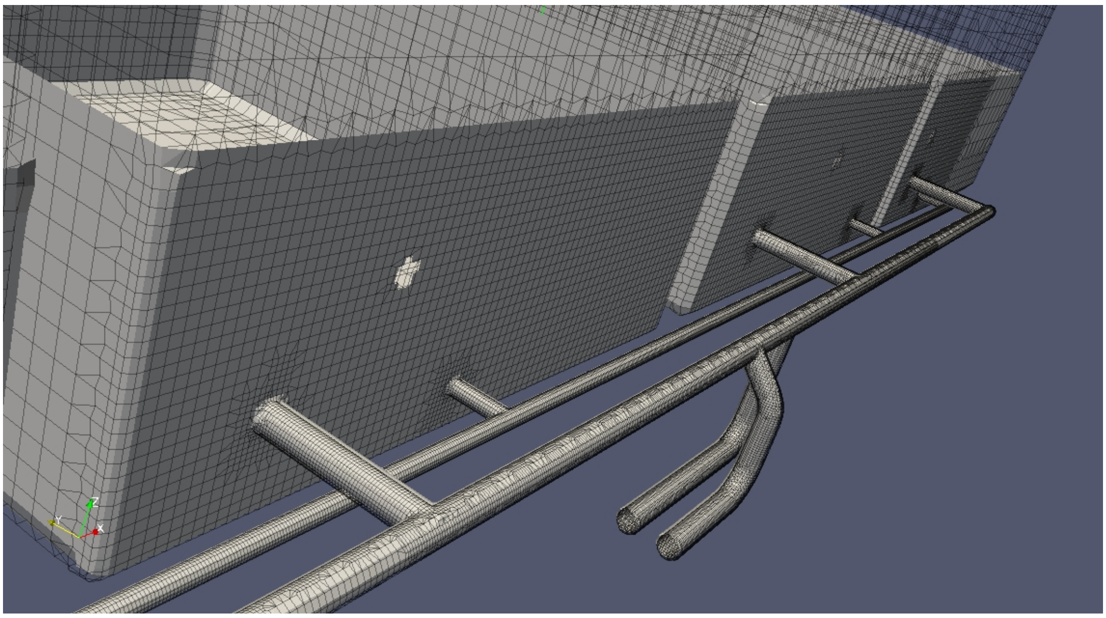

Figure 2: ***Overview of water tank arrangement***

2.1 Model Setup

The numerical software 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 OpenCFD Ltd. The OpenFOAM software package has been the subject of benchmarking studies for the analysis of hydraulic structures including dams. The other model used for the solution was a single phase model, steady state solver utilising Reynolds averaged Navier-Stokes (RANS) turbulence model. The solution model was based on:

- Turbulence modelling using RANS
- Incompressible flow assumptions
- Steady state simulation
- Single phase flow (i.e. the water surface is not computed).

2.2 Computational Mesh

A CAD geometry file was used as the basis to generate the computational mesh which consists of an unstructured mix of polyhedral and hexahedral cells. The spacing of the mesh is 25mm near the wall boundaries and approximately 50mm elsewhere in the region of the tank. The mesh was generated using the snappyHexMesh utility which comes with the openFOAM software package. A background initial mesh is used to define the extent of the computation mesh. The user can define a range of parameters such as regions of refinement and the number of boundary layers.

2.3 About Passive Scalars

The passive scalar method resolves the convection-diffusion transport equation for a passive scalar such as temperature or convection, using a user-specified stationary velocity field.

The velocity field is given and not modified by the transported field. In contrast, an active scalar modifies the physical properties, such as density and viscosity, according to the local concentration of the scalar. The passive scalar does not influence the fluid properties, but there is still a transport equation solved for it. It can be considered as a tracer. Also, inertial effects are neglected the passive scalar approach.

3.0 RESULTS

An example of the flow paths visualised with stream tubes is shown in Figure 3. On first inspection, from these figures alone, it may appear difficult to judge which the better option is. A recommendation was made based on solid engineering judgement. However this method does not lend very well for field evaluations to confirm model simulations or to quantify and compare the residence time for the various options. For the revised assessment, a passive scalar or tracer is released at the inlet of the tank while the concentration is recorded at the outlet and plotted.

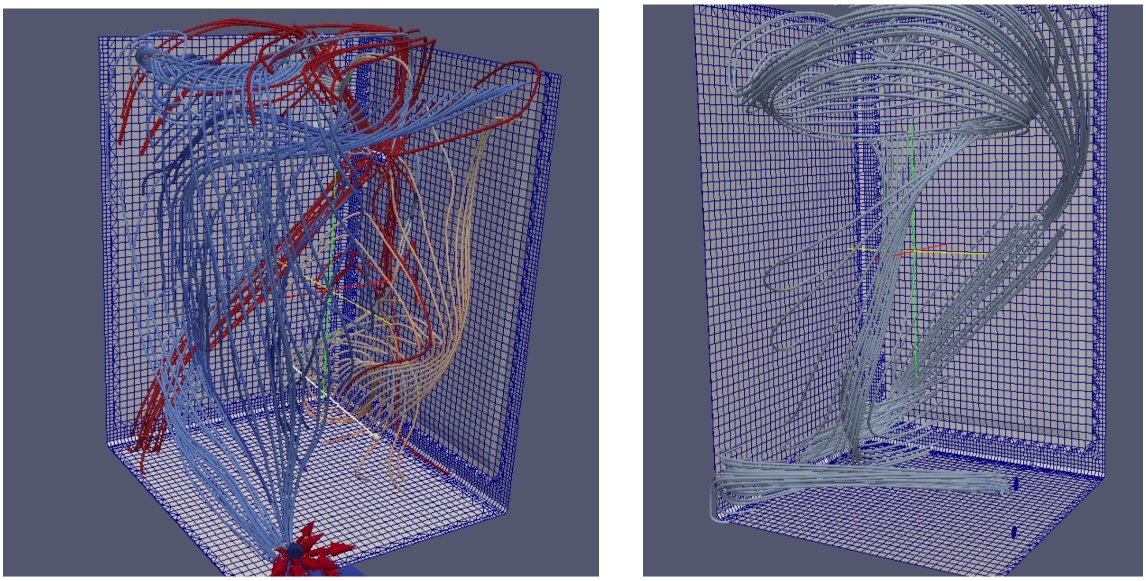

Figure 3: ***Flow paths. Base case (left), Option 2 (right)***

The RTD curves are shown in Figure 4 and the same results are also given in Table 1. The influence of short circuiting can be seen from the shape of the curve in Figure 4 for the base case and option 1.

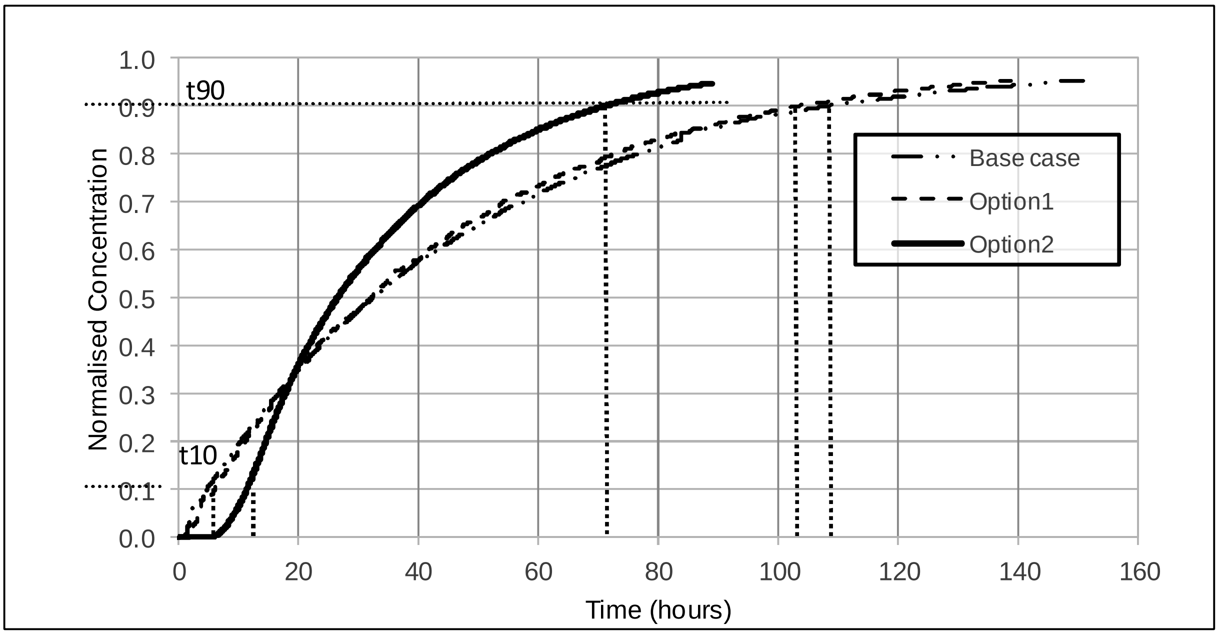

Figure 4: ***Residence Time Distribution Curves***

The theoretical residence (or detention) time (TRT) is the ratio of the tank volume to the average inlet flow rate. The variable t_{10} is the time taken for 10% of the input concentration to be observed at the outlet. The BF, which is widely assessed from site sampling, scale modelling or simply estimated, is the ratio of t_{10} to the TRT and typically ranges from 0.1 such as an un-baffled tank with short-circuiting to maximum value of 1.0 representing ideal plug flow conditions. Values of t_{10} and BF, determined from CFD simulation using passive scalar, are listed in Table 1. Both options represent improvements in BF and hence residence time over the base case, with option 2 being significantly better. These results confirm the original assessment.

A full RTD curve, with both t_{10} and t_{90} indicated, gives a clearer indication of the internal flow dynamics of a system; which gives a better insight than using BF as to whether the flow is short-circuited or plug flow (Jordan et al 2010). A steeper gradient of the curve represents conditions closer to ideal plug flow. The ratio of t_{10}/t_{90} can be used in place of the BF to assess a tank. In cases where the tank has no baffles the BF calculation will be under estimated compared to the t_{10}/ t_{90} ratio.

Another advantage of using passive scalars is that it is easy to output a video representation of the spread of the passive tracer which leads to valuable insights. For example short circuiting is visually apparent.

Table 1: ***CFD Results***

Case	**t_{10} (hrs)**	**t_{90} (hrs)**	**BF**	**t_{10}/ t_{90}**	**% improvement compared to the base case**
base case	4.7	108.0	0.14	0.044	NA
option 1	6.1	103.3	0.18	0.059	28
option 2	11.5	71.0	0.34	0.161	142

3.0 CONCLUSION

A CFD model was used to qualitatively assess the performance of the water storage tanks by incorporating a trace (or passive scalar) into the CFD simulation. This is a better method than commonly used flow paths to assess residence time. It is ideal to use both methods in conjunction.

An alternative means of assessing tank performance and contact time was discussed using the t_{10}/ t_{90} ratio which is more accurate for un-baffled tanks, instead of using BF.

The use of CFD modelling 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 (Jacobsen et al 2009).

4.0 ACKNOWLEDGEMENTS

I wish to thank Abigroup and AECOM for permission to present this paper and Caroline Alepin for her kind help regarding all things regarding water quality.

5.0 REFERENCES

Jordan M, Wilson & Subhask Venayagamoorthy. *Evaluation of Hydraulic Efficiency of Disinfection Systems Based on Residence Time Distribution Curves,* American Chemical Society, Environ. Sci. Technol, 44 (24), pp 9377–9382 (2010)

Jacobsen F, Brown L, O*ptimise tank design using CFD, 72nd Annual Water Industry Engineers and Operators' Conference Bendigo Exhibition Centre.* September, 2009