May 17th, 5:00 PM

Numerical Simulation and Assessment of the Effects of Inlet Configuration on the Flushing Time of a Potable Water Service Reservoir

C. O'Neill
c.oneill8@nuigalway.ie

Colin James O'Neill Mr
National University of Ireland, Galway, c.oneill8@nuigalway.ie

Recommended Citation

O'Neill, C. (2018). Numerical Simulation and Assessment of the Effects of Inlet Configuration on the Flushing Time of a Potable Water Service Reservoir. Daniel Bung, Blake Tullis, 7th IAHR International Symposium on Hydraulic Structures, Aachen, Germany, 15-18 May. doi: 10.15142/T32936 (978-0-692-13277-7).

7th International Symposium on Hydraulic Structures Aachen, Germany, 15-18 May 2018
ISBN 978-0-692-13277-7 DOI: 10.15142/T32936

Numerical Simulation and Assessment of the Effects of Inlet Configuration on the Flushing Time of a Potable Water Service Reservoir

C. J. O'Neill[1], S. Nash[2], E. Clifford[2] & S. Mulligan[1]
[1]*College of Engineering and Informatics, National University of Ireland Galway (NUIG), Galway, Ireland*
[2]*Ryan Institute, NUI Galway eoghan.clifford@nuigalway.ie *corresponding author*
E-mail: c.oneill8@nuigalway.ie

Abstract: *Service reservoirs provide the dual function of balancing supply with demand and providing adequate pressure in the water supply network by maintaining sufficient pressure head. In a potable water service reservoir, the flushing time is a major water quality concern due to the potential loss of chlorine residual. The flushing time is closely related to the flow pattern in the reservoir. In this study, the influence of change of water inlet configuration and baffling on flushing time was examined. Multiphase Computational Fluid Dynamics (CFD) was used to simulate service conditions and predict flow patterns. To determine the flushing time, two identical fluids with the properties of water were used in the simulation with one fluid in the reservoir initially and the other fluid introduced through the inlet. The volume fraction of the initial fluid at the ideal flushing time was used to determine the effects of inlet configuration and baffling. A laboratory scale model was used to validate results produced from the CFD model. In the physical model, a saline solution was used as the initial fluid in the reservoir and fresh water was introduced through the inlet. A time series of the conductivity of the solution was measured at a specific point and was compared to a volume fraction monitor point in the numerical simulation to validate results. Four different scenarios were assessed with different inlet configurations and locations including a baffle wall located near the inlet to direct the flow.*

Keywords: *Computational Fluid Dynamics, service reservoir, water quality, flushing time, inlet configuration.*

1. Introduction

Once water has been treated in a water treatment plant and is potable, it must be conveyed to consumers. As demand in each catchment varies greatly throughout the day and over the course of a week, maintaining constant water pressure in an area is achieved using service reservoirs and water tower structures. A trunk main usually supplies a service reservoir directly from the water treatment plant; the service reservoir then in turn supplies the local area through a series of distribution mains. Service reservoirs are usually located at the highest elevation in the area to provide adequate pressure head to the distribution network while also maintaining supply to the area should the treatment plant or pumps breakdown.

After the treatment process, the water maintains a chlorine residual which acts as a safeguard against additional microbial contamination that could occur in the distribution process. This chlorine residual will decay over time; therefore, reducing the residence time of the distribution network is important for ensuring that there is a high quality of water for the consumers. Therefore, the study of flow in a service reservoir is of major interest in the design of the reservoir to improve water distribution and to prevent short circuiting within the reservoir. Completely mixed flow is desirable in a service reservoir as it minimizes the potential loss of disinfectant (Grayman et al. 2004). Ideally this can be achieved simply by modifying the geometry of the tank or the inlet and outlet configuration thus removing the need for mixers or agitators which induce high energy and maintenance costs.

The shape of the reservoir has a significant influence on the flow conditions in the reservoir and previous studies have shown that a rectangular tank with an inlet at one end and an outlet at the other is the best design when considering mixing and age distribution (Lee et al., 2014). Increasing the length to width ratio for rectangular reservoirs causes the flow conditions in a reservoir to resemble plug flow for a vertical inlet (Yeung, 2001). Using a series of four distributed inlets was found to be a viable measure to preserve water quality in a circular tank (Montoya-Pachongo et al., 2016).

The effect of installing baffle walls to improve mixing and residence time in a service reservoir still remains inconclusive with contradictory results found in previous studies. One study found that while baffle walls retrofitted to areas prone to recirculation tended to break up the vortices and shorten the flow path, the water velocity is reduced after flowing past the baffle walls (Zhang et al. 2013). It is recommended that under most circumstances baffles should not be used in service reservoirs where they can inhibit mixing, especially in fill-and-draw reservoirs (Grayman et al. 2004).

Computational Fluid Dynamics (CFD) offers an effective tool to allow parameterization and optimization of flow control geometry in reservoirs (Jarman et al., 2008; Dewals et al., 2008; Dufresne et al. 2011). The majority of previous studies adopting CFD to evaluate the residence time of a service reservoir used 'tracer tests' to calculate the residence time. This consisted of injecting a pulse of tracer through the inlet and measuring the concentration at the outlet (Yeung 2001, Montoya-Pachongo et al. 2016). However, this method does not give a clear indication as to the flushing time of the reservoir and a tracer test alone may not be indicative of the mixing and age distribution in the reservoir. Flushing time, residence time and age are not consistently defined in literature as transport time scales and they can often be confused (Monsen et al. 2002). In this study, four different inlet configurations are compared to evaluate the effect they have on the flushing time of a reservoir with consistent geometry.

2. Methodology

2.1. Physical Geometry

The physical geometry of the reservoir was based on a new 7.5 Ml reservoir proposed to be constructed in the United Kingdom. This new reservoir is to serve as an ancillary cell to an existing 15 Ml reservoir adjacent to the proposed site. Therefore, the physical geometry of the proposed design had to be similar to the existing structure with the same roof height. The proposed reservoir has dimensions of 50 m in length, 30 m in width and 5 m in height to the top water level. The height of the roof of the reservoir is sufficiently higher (0.59 m) than the top water level. Roof support columns, 0.35 m square, are spaced at approximately 5 m center spanning in both directions. The proposed inlet is a 0.45 m diameter ductile iron pipe that enters the reservoir vertically from underground from one side of the tank and overspills a bell-mouth on top from above the top water level. The outlet is situated at the other end of the tank in a sump. Similar to the inlet, the outlet comprised of a 0.45 m diameter ductile iron pipe with a bell-mouth at the entry, the top of which is level with the base of the reservoir. Both the inlet and outlet are offset to the centerline of the tank on opposite sides. Because making changes to the physical geometry of the reservoir is restrained by the existing reservoir, a number of different inlet configurations were to be evaluated and analyzed to improve flushing time in the reservoir.

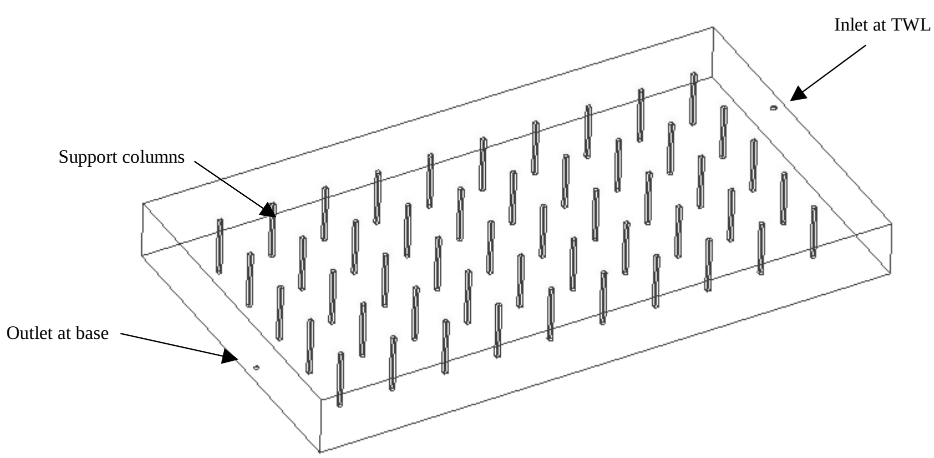


Figure 1. Schematic layout of the proposed service reservoir.

2.1.1. Test Scenarios

Four different inlet configurations were analyzed to determine the most suitable using a 3D numerical model. Scenario 1 (*S1*) is as outlined above in the original proposal with the inlet over-spilling a bell-mouth set above the top water level. Scenario 2 (*S2)* has the inlet in the same $x - y$ position but is located at the base level similar to the outlet. Scenario 3 (*S3*) has the same inlet as *S1* but has a baffle wall across the support columns immediately adjacent to the inlet, transverse to the long side wall. The purpose of the baffle was to direct flow into the corners of the reservoir.

Scenario 4 (*S4*) has a horizontal inlet normal to the back wall of the tank at the same y dimension as the *S1* and 0.5 m above the base. The different test scenarios are outlined in Figure 2 (a-d) below.

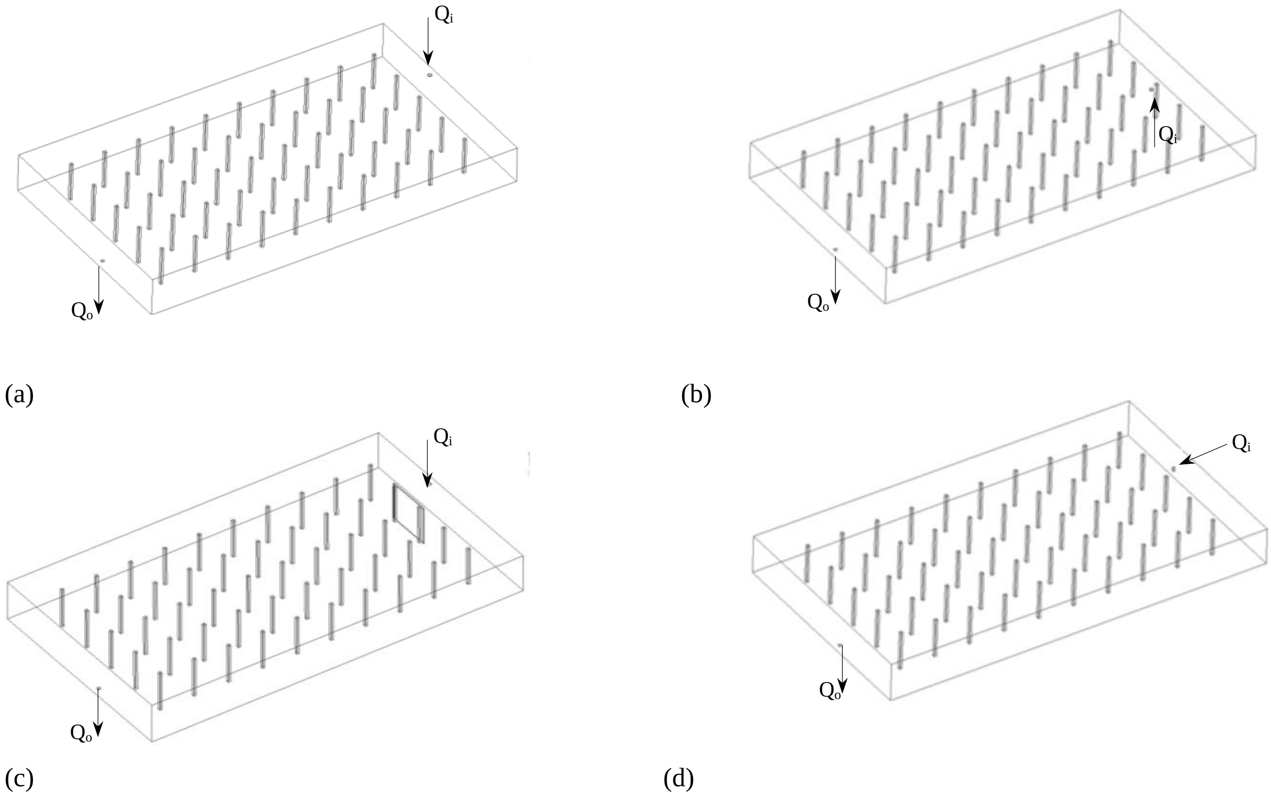


Figure 2. Schematic layout of each test scenario with Q_i showing the inlet location and direction and Q_o showing the outlet. (a) Scenario 1, (b) Scenario 2, (c) Scenario 3 and (d) Scenario 4.

2.2. Numerical Simulation Methodology

Three-dimensional numerical modelling was undertaken using Ansys CFX 18.1 software with a high-performance computer used to solve the simulation. In each simulation, the full tank domain was modelled including the columns

An unstructured tetrahedral dominant mesh grid was constructed to mesh the domain. Grids were refined close walls such as the inlet, outlet, walls and columns through either inflation or face sizing to ensure fine resolution to capture the boundary layer. The mesh comprised of approximately 664,000 nodes and 3,471,000 elements for each scenario, varying slightly depending on the geometric scenario under consideration. A mesh sensitivity analysis was carried out initially to determine to appropriate mesh density to provide a mesh independent solution. For this analysis, monitor points were used to measure time averaged velocity for various mesh refinements starting at a relatively course mesh and refining it each time until the results were independent of the mesh resolution.

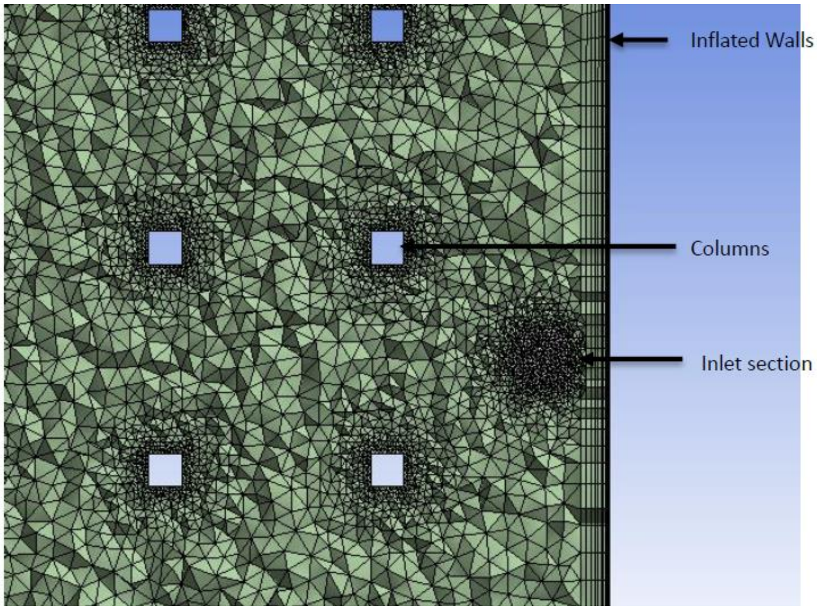


Figure 3. Typical computational mesh used for numerical simulations. Denser mesh grids around boundary conditions and walls.

For the purpose of this simulation, the air-water interface was assumed to be a constant fixed lid or horizontal plane positioned at the top water level. This is a valid approximation given that sloshing of water or other dynamic effects of the air water interface would most likely be absent in order to avoid additional computational cost of modelling the water-air interface (Zhang et al. 2011). The top water level plane is specified as a smooth wall with free-slip conditions.

The main boundary conditions are outlined in Table 1. All walls and columns were specified as smooth no-slip walls. Although the reservoir is a concrete structure, given the low velocity and relatively laminar flow, applying a roughness factor to the surface was deemed unnecessary and for the bed of the reservoir the friction factor is only considered important when the water depth is very low (Dufresne et al. 2011)

Table 1. Boundary Conditions.

Boundary Type	*Conditions*
Inlet	Normal velocity = 0.884 m/s
Outlet	Static Pressure = 0 Pa
Wall	Smooth, no-slip
Top water level	Smooth, slip wall

For the inlet condition, a constant normal speed of 0.884 m/s was applied. This boundary condition was calculated using Eq. (1) to determine the ideal flushing of this tank over a 12 h period where τ_f is the ideal flushing time and V is the volume of the tank. For the outlet condition, a static gauge pressure of 0 Pa was applied.

$$Q = \frac{V}{\tau_f} \tag{1}$$

Multiphase analysis was used to determine the flushing time of the model. Two fluids with the same properties as water were used, with Fluid 1 in the reservoir initially and Fluid 2 being introduced through the inlet after the flow was fully developed. The in-homogenous Eulerian-Eulerian Model was used to model interphase mixing in the system (ANSYS CFX-Solver Theory Guide 2011). The standard Shear Stress Transport (SST) was adopted to model turbulence. The Transient Model was allowed to run for a 12 h period (prototype) plus time (the equivalent of 1 h in prototype scale) for the flow to fully develop initially.

Monitor points were positioned in the center of the tank and the volume fraction of each fluid was monitored over the period of the simulation together with the average volume fraction for each fluid for the whole domain. When the volume fraction of Fluid 1 was equal to zero, the flushing time of the tank would be realized. However, for this study the scenario that had the smallest volume fraction of Fluid 1 remaining after a 12 h period was determined to be the most suitable design.

2.3. Physical Model Methodology

To validate the numerical model simulation, a 1:50 scale physical model of the reservoir was constructed in the laboratory for Scenario 1. The physical scale model was made using perspex for the walls and treated timber to simulate the roof support columns. Although in reality the inlet and outlet discharge fluctuated over time, to simplify the model a steady inlet and outlet flow was assumed. A header tank and ball-valve was used at the inlet to maintain a steady flow and a ball-valve was used at the outlet as shown in Figure 4.

The physical parameters of the model were calculated using Froude similitude due to it being a free surface problem (Chanson, 1999). The inlet discharge (Q_i) was set to 0.01 l/s which would give an ideal flushing time (τ_f) of 12 hours in the full-scale prototype. The outlet discharge (Q_o) was set equal to the inlet to maintain constant depth.

Red food colouring was added to the header tank and a time-lapse video recording of plan view was obtained to give an indication of the flow patterns for qualitative comparisons with the numerical model. To quantitatively measure the flushing time of the scale model, a conductivity probe was set up in the center of the tank at the same location as the monitor point for the numerical model for comparative purposes. Central tank and outlet readings were obtained at one minute and three-minute time intervals, respectively. Table salt was added (0.4 g/l) to the fresh water in the tank to increase the conductivity and the tank was mixed thoroughly to ensure an even distribution. The header tank was filled with fresh water and the outlet was opened. The model was allowed to flow for 10 minutes to allow the flow to fully develop before the probe started recording conductivity measurements. The experiment was repeated three times to ensure repeatability and accuracy of results.

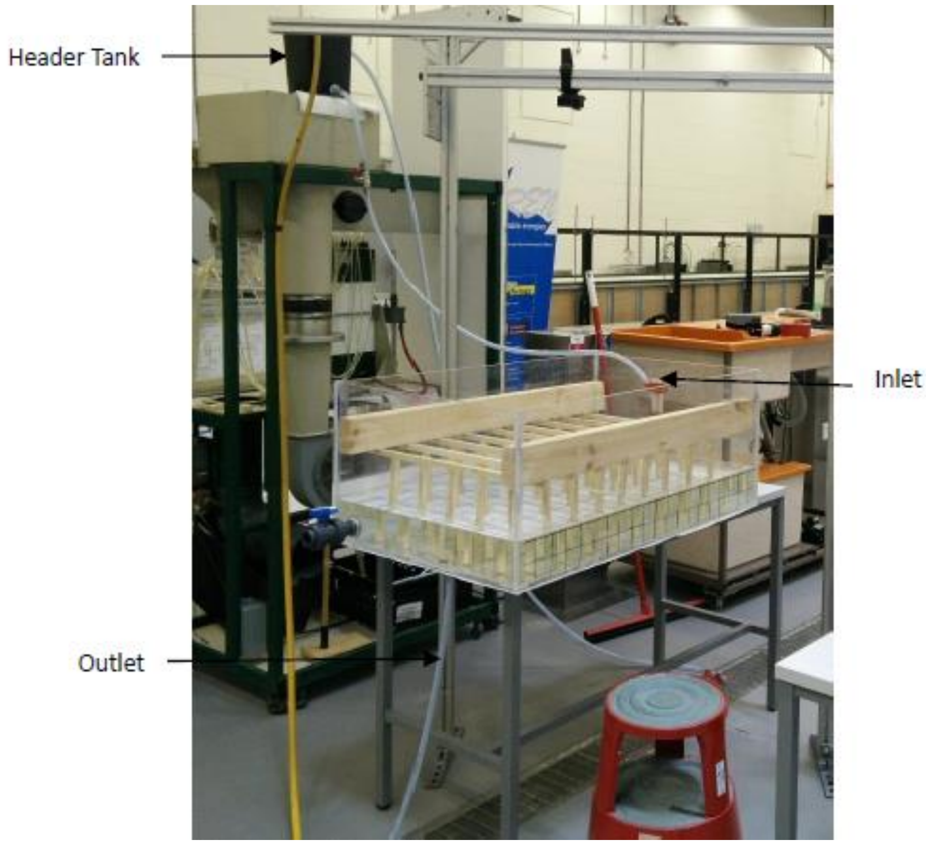
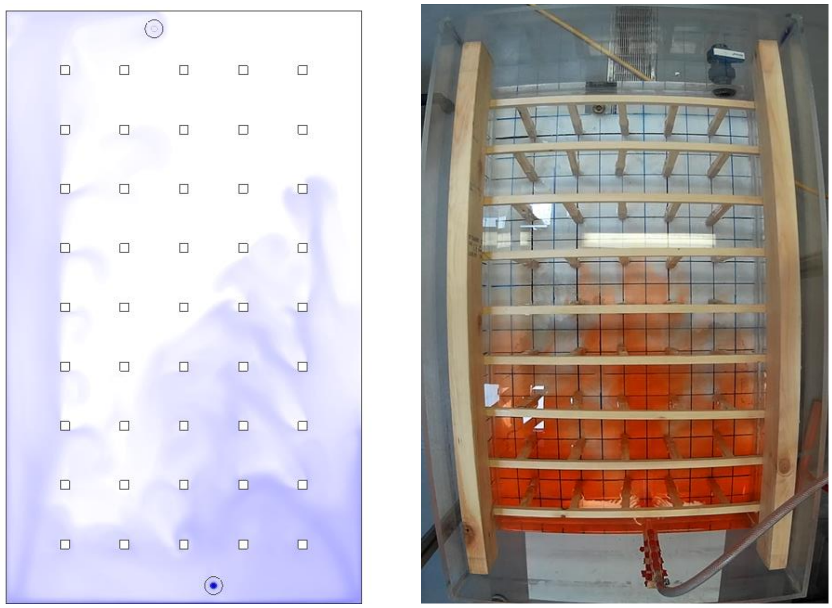


Figure 4. Physical scale model setup in the hydraulics laboratory.

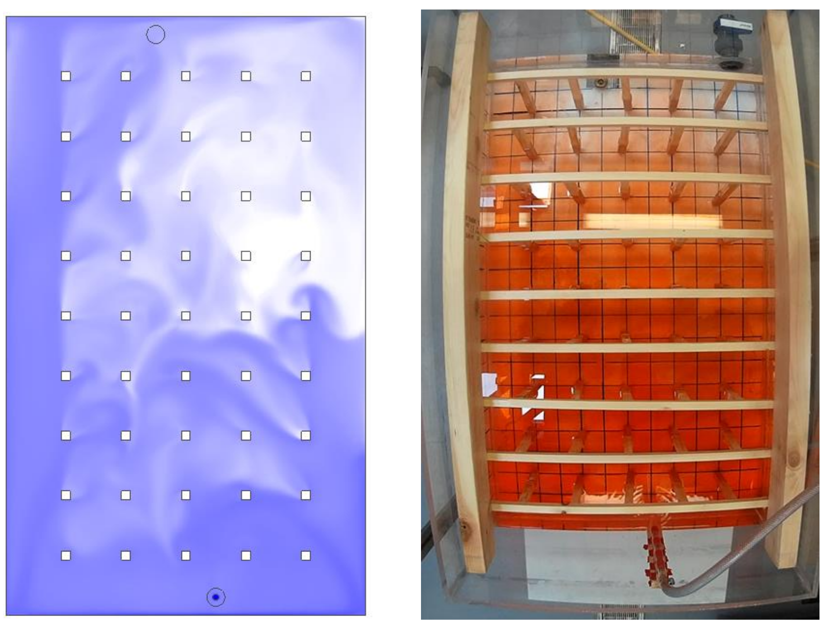
3. Results and Discussion

3.1. Validation of Numerical Model Methodology

The physical model results were used to initially assess the validity of the numerical model. Figure 5 below shows a qualitative comparison between the flow in the physical model compared to the numerical model. Food colouring was added to the header tank in the physical model to show the flow pattern in the model and volume rendering was used in the numerical model at specific time steps to show the flow of Fluid 2 in the domain. From visual inspection alone, these results agree well qualitatively with poorly mixed regions matching approximately at the various timesteps



(a)



(b)

Figure 5. CFD volume rendering and food colouring test for time steps (a) t = 4 h (b) t = 10 h.

The conductivity readings from the probe in the center of the tank were used to calculate the flushing time in the physical model. The initial conductivity (σ_i) reading of the saline solution minus the conductivity of fresh water (σ_w), was set equal to one and the subsequent readings ($\frac{\partial\sigma}{\partial t}$) were taken as a fraction of the initial reading. This allowed the conductivity results from the physical model to be comparable to the volume fraction (VF) results for each monitor point in the numerical model (Eq. (2)). Figure 6 shows a graph of these results for *S1* with an error bar on the measured results calculated by taking the standard deviation of the three physical model tests results plus the tolerance of the probe.

$$\frac{\frac{\partial\sigma}{\partial t}}{(\sigma i-\sigma w)} = VF \quad (2)$$

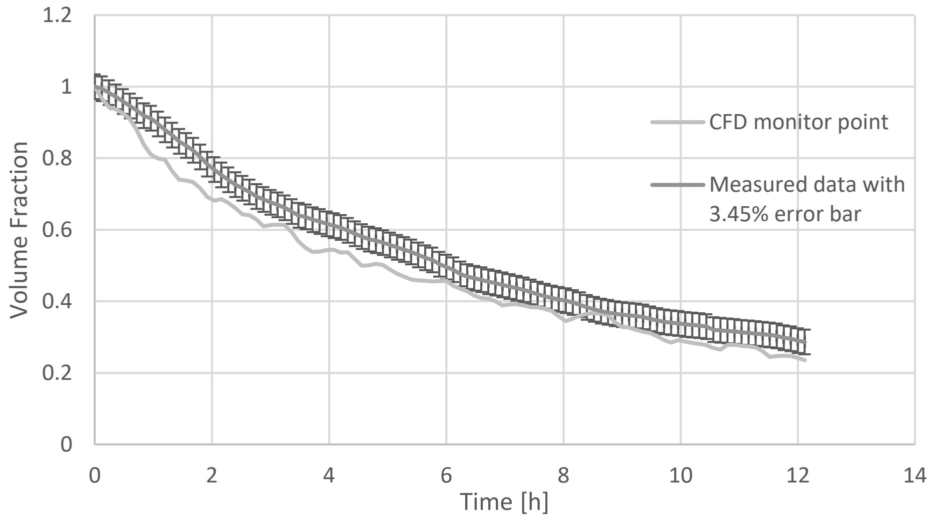
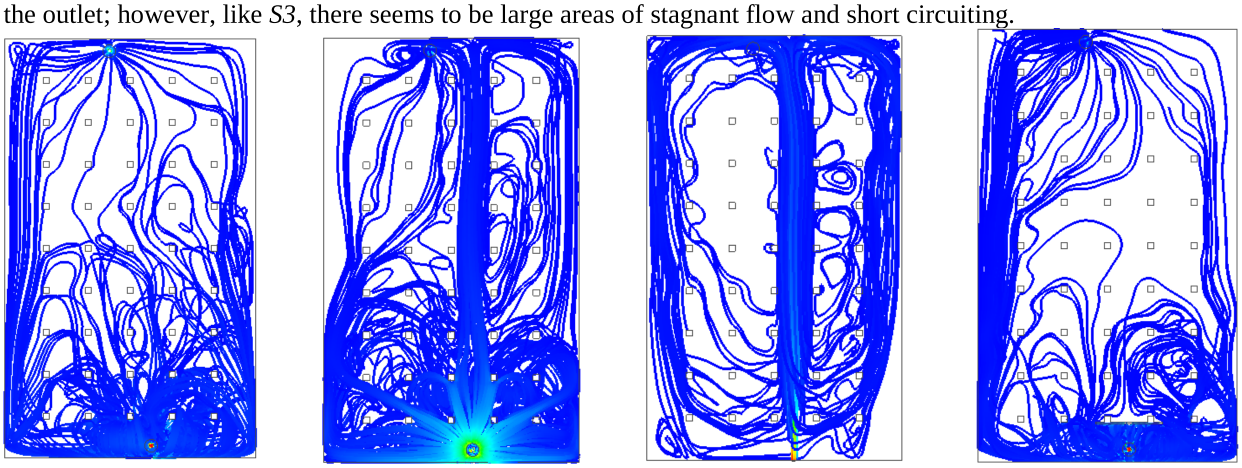


Figure 6. Physical model measured data and CFD output data.

This numerical model methodology was subsequently applied to *S2*, *S3* and *S4* to compare the effect of inlet configuration on the flushing time and flow pattern in the model. Figure 7 shows the 3D streamlines from the inlet for each scenario. In *S1*, the jet from the inlet dissipates homogenously for the first half of the tank and the flow seems well dispersed. *S2* streamlines indicate that when the inlet is at the base the flow follows a more direct route to the outlet but is still mixed reasonably well. In *S3*, the jet from the inlet continues until the flow hits the opposite wall and is dispersed into the corners and back down the side walls. There also seems to be large areas of stagnant flow either side of the jet in *S3*. Finally, *S4* shows that the baffle directs the flow into the corners initially and up the walls towards the outlet; however, like *S3*, there seems to be large areas of stagnant flow and short circuiting.



(a) (b) (c) (d)

Figure 7. Plan view of three dimensional streamlines of flow from the inlet for scenarios (a) Scenario 1, (b) Scenario 2, (c) Scenario 3 and (d) Scenario 4.

Figure 8 presents the relationship between the volume fractions of Fluid 1 over time for each of the scenarios. From this graph, it is clear that *S1* will have the shortest flushing time as the flow is almost fully dispersed and was observed in the streamlines of Figure 7. *S2* and *S3* appear to perform poorly as the flow from the inlet is conveyed almost directly to the outlet leaving large areas of the tank stagnant. In *S4* the baffle wall blocks the water from going directly to the outlet and disperses it into the corners and along the side walls, but the baffle creates recirculation flow around it and in the center of the tank. All scenarios show that the rate of flushing reduces exponentially over time.

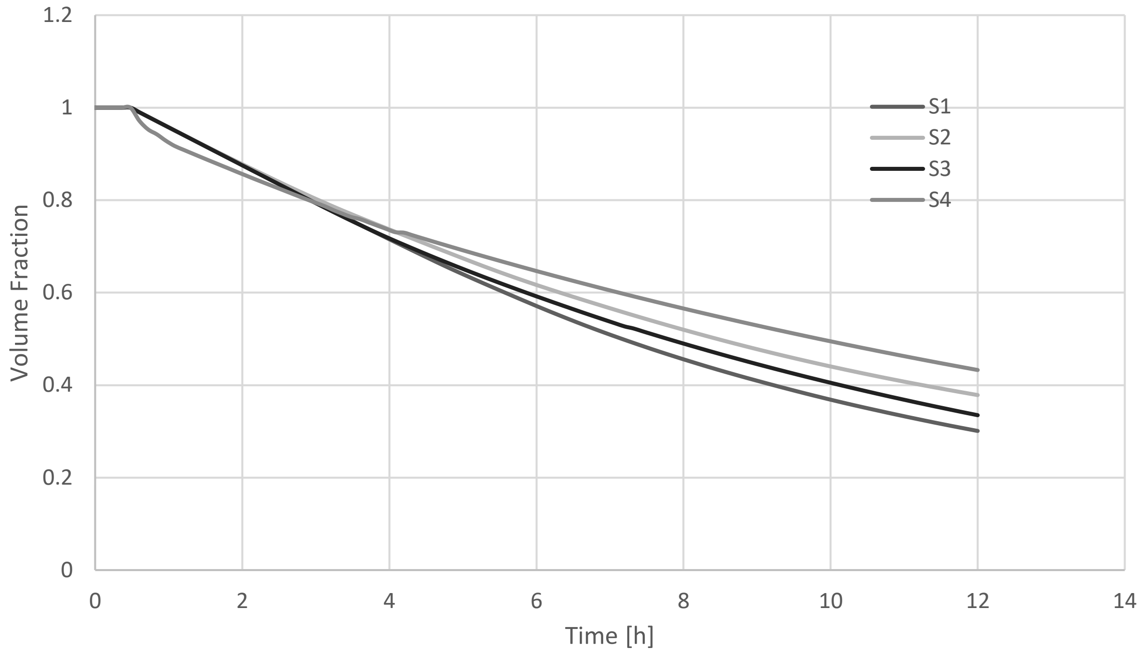


Figure 8. Average volume fraction of Fluid 1 in the domain for each scenario tested.

4. Conclusion

In this study, a three-dimensional multiphase numerical model of a reservoir is undertaken to assess mixing patterns and flushing times. The numerical model is validated using a 1:50 physical model of the hydraulic structure. The numerical and physical models were found to compare qualitatively well. Four inlet scenarios were investigated based on varying the inlet configuration and geometry. The results found that the inlet configuration of a reservoir can have a relatively considerable influence on the flushing time and flow patterns. A bell-mouth inlet over-spilling above the top water level was found to be the most suitable inlet configuration for this specific reservoir. This was demonstrated for examining the volume fractions of mixed fluid phases over a 12-hour simulation period where only 30% of the fluid initially in the tank remained. This was almost 13% better than a horizontal inlet under the same conditions and also performed more effectively than with a baffle or a base level vertical inlet.

To improve on this study, further investigation into suitable numerical schemes to represent fluid level variations is required to accurately represent the flow conditions in reservoirs. It is also recommended that the influence of the varying turbulence Schmidt number on the mass turbulent transport be investigated as previous studies have found that mixing is highly dependent on this parameter (Valero and Bung, 2016). Nonetheless, this study highlights that the proposed numerical approach is an effective method for calculating the flushing time of a reactor such as this. It is recommended that for future studies a smaller scale difference be used for validation and possible use of a chemical tracer for quantifying the flushing time.

5. Acknowledgments

The authors of this paper extend their appreciation to National University of Ireland, Galway, Ward and Burke Construction Ltd., and Irish Centre for High End Computing (ICHEC). Special thanks are also given to Sean Cranney and Jerry Lin.

6. References

"ANSYS CFX-Solver Theory Guide." 2011. 15317(November): 724–46.

Chanson, Hubert. 1999. "Chapter 14: Physical Modelling of Hydraulics." *The Hydraulics of Open Channel Flow*.

Dufresne et al. (2011). "Numerical investigation of flow patterns in rectangular shallow reservoirs." Engineering Applications of Computational Fluid Mechanics 5 (2), 247-258.

Dewals et al. (2008). "Experimental and numerical analysis of flow instabilities in rectangular shallow basins." *Environmental Fluid Mechanics*, 8 (1), 31-54

Grayman, W M et al. 2004. "Mixing and Aging of Water in Distribution System Storage Facilities." *AWWA* 96(9): 70–80.

Lee, Hp et al. 2014. "Shape Effect on Mixing and Age Distributions in Service Reservoirs Shape Effect on Mixing and Age Distributions in Service Reservoirs." (November).

Monsen, N E, J E Cloern, and L V Lucas. 2002. "A Comment on the Use of Flushing Time, Residence Time, and Age as Transport Time Scales." 47(5): 1545–53.

Montoya-Pachongo, C, S Laín-Beatove, P Torres-Lozada, and C H Cruz-Vélez. 2016. "Effects of Water Inlet Configuration in a Service Reservoir Applying CFD Modelling." 2016: 31–40.

Valero, Daniel, and Daniel B Bung. 2016. "Sensitivity of Turbulent Schmidt Number and Turbulence Model to Simulations of Jets in Cross Flow." *Environmental Modelling and Software* 82: 218–28. http://dx.doi.org/10.1016/j.envsoft.2016.04.030.

Yeung, Hoi. 2001. "Modelling of Service Reservoirs." 165–72.

Zhang, Jun-mei et al. 2011. "Modeling and Simulations of Flow Pattern, Chlorine Concentration, and Mean Age Distributions in Potable Water Service Reservoir of Singapore." 137(July): 575–84.

Zhang, Jun-mei, Boo Cheong Khoo, Heow Pueh Lee, and Chit Pin Teo. 2013. "Numerical Simulation and Assessment of the Effects of Operation and Baffling on a Potable Water Service Reservoir." 139 (March): 341–48.