RESERVOIR CFD MODELLING – WHAT ARE THE BENEFITS OF USING A 3D MODEL?

Nasrine Tomasi AWT Water Limited ,Robert White AWT Water Limited, Thomas Joseph AWT Water Limited

ABSTRACT

Water utilities own and operate a number of potable water reservoirs throughout their water supply network. The behaviour of water in reservoirs can impact on water quality. It is interesting to understand the movement of water within water tanks in order to ensure that potable water is not held up in “dead zones” as increased residence time results in residual chlorine decay that augments the risk of recontamination in the network.

Reservoir models can assist in finding efficient and cost effective solutions to ensure satisfactory flow path within reservoirs by assessing the impact of baffle curtains and/or comparing different layout options. In general 3D Computational Fluid Dynamics modelling is recommended to provide detailed solutions that involved fluid flows. However 3D models are often complex to build and have long runtimes which can make them prohibitive costs.

The main objective of the work described in this paper was to investigate the possibility of using simpler 2D open-channel software for modelling flows in water tanks where the reservoir layout does not significantly differ in depth. A case study was used to compare the simulation results from a 3D CFD model to those from 2D computations. Comments are made regarding the limitations of 2D modelling and the extent in which 2D models can be used to estimate the movement of the water in reservoirs.

KEYWORDS

Reservoirs, CFD modelling.

1 INTRODUCTION

Water utilities own and operate a number of potable water reservoirs throughout their water supply network. Reservoirs help providing onsite storage to buffer for peak demands and emergency supply situations. However increased residence times in reservoirs result in residual chlorine decay that augments the risk of recontamination in the network and can significantly affect water quality in distribution systems.

Reservoirs design and operation are critical to minimise water age. It is essential to size reservoirs such that residence time is as low as possible whilst presenting sufficient volume to provide security of supply. Volume of water flowing in and out of the reservoirs must be maximised and reservoirs must be regularly filled and emptied through network operation to guarantee renewal of the water stored.

The behaviour of water in reservoirs is another decisive parameter that can impact on water quality. It is important to understand the movement of water within water tanks in order to ensure that potable water is not held up in “dead zones”. Ideal flow pattern through a reservoir is plug flow. In plug flow, all fluid elements entering the reservoir have a constant velocity and present the same hydraulic residence time. In these conditions water passing through the reservoir is not retained in “dead areas” and water age is minimised.

In general three dimensional (3D) Computational Fluid Dynamics modelling is recommended in order to provide detailed solutions that involved fluid flows. However 3D models are often complex to build and have very long runtimes which can make them prohibitive costs. The main objective of the work described in this paper is to investigate the possibility of using simpler 2D open-channel software for modelling flows in water tanks where the reservoir layout does not significantly differ in depth. A case study is used to compare the simulation results from a 3D CFD model to those from 2D computations. Comments are made regarding the limitations of 2D modelling and the extent in which 2D models can be used to estimate the movement of the water in reservoirs.

2 DESCRIPTION OF MODELS

2.1 GSSHA 2D MODEL

The Gridded Surface Subsurface Hydrologic Analysis (GSSHA) software developed by the US Army Engineer Research and Development Center was used to simulate flows in a case study reservoir. This software is a physics-based, distributed, hydrologic, sediment and constituent fate and transport model. Features include two dimensional (2D) overland flow, 1D stream flow, 1D infiltration, 2D groundwater, and full coupling between the groundwater, shallow soils, streams, and overland flow.

The 2D overland flow routing feature was used to model the path of water in a reservoir. GSSHA uses two-step explicit finite volume schemes to route water for 2D overland flow. Flows are computed based on heads, and volumes are updated based on the computed flows.

2.2 FLOWSOLVE 3D MODEL

The computational fluid dynamics (CFD) modelling has been undertaken by Dr David Glynn of Flowsolve Limited, London. Flowsolve undertook this work as an independent engineering software consultancy, specialising in the provision, application and support of CFD models. The company was founded in 1990 to address the flow simulation requirements of industrial and academic organisations within the desktop environment.

Flowsolve's Principal Consultants Dr David Glynn, Dr Patrick Phelps and Dr Sergei Zhubrin each have over 20 years experience in the construction and application of CFD models for clients in a wide variety of industrial and environmental sectors.

3 METHODOLOGY

3.1 CASE STUDY

A T-shaped reservoir presenting a top inlet pipe and bottom outlet was selected as a case study. As shown in the Figure 1 below the reservoir dimensions were assumed to be 50m wide and 54m long.

Two scenarios were developed: the reservoir was first modelled without a baffle curtain. A second scenario assuming the installation of a baffle curtain located at mid-distance along the inlet-outlet segment was then considered. Both scenarios were modelled in 2D and 3D.

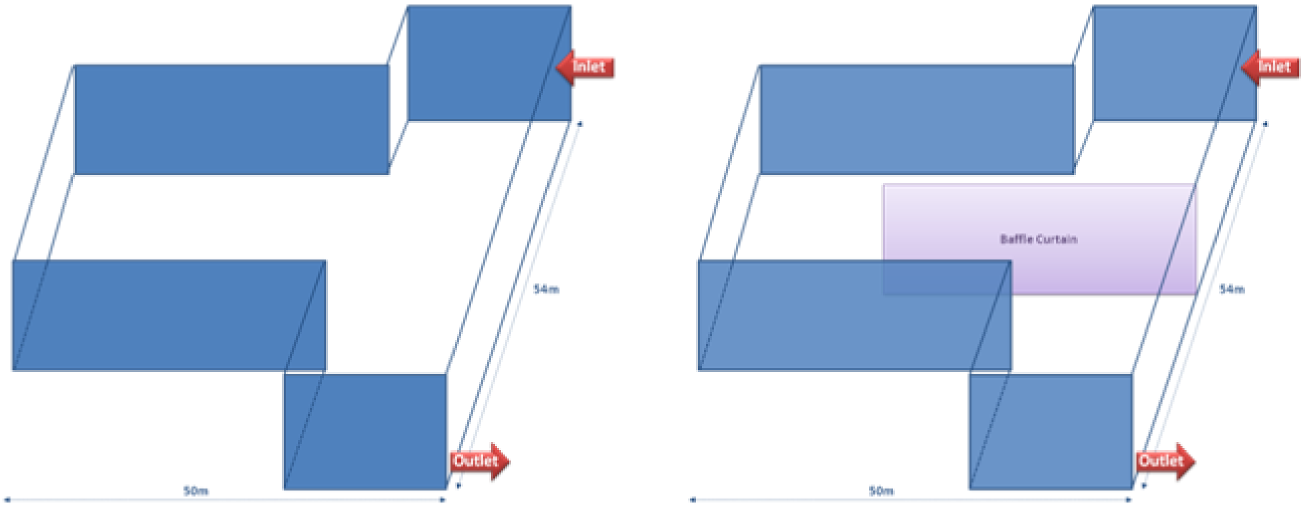

Figure 1: Case Study Reservoir Schematics

3.2 MODELLING ASSUMPTIONS

Simplified models of actual behaviour were developed. Constant flows and head were assumed where, in general, actual inflows and outflows have the potential to change..

3.2.1 2D MODELS

2D reservoir models with and without the baffle were developed, and run with constant and balancing inflow and outflow. The models assumed a constant inflow / outflow of 150l/s, with discharge calculated on a 2 metre by 2 metre grid. Reservoir walls were modelled as high elevated cells. The inlet discharge was loaded to one cell located at the northeast corner of the reservoir. The outlet was defined as one cell located at the southeast end of the reservoir. Reservoir columns were not modelled in the 2D models.

3.2.2 3D MODELS

The models assumed a constant inflow / outflow of 150 l/s, with water entering the northeast corner of the reservoir at a high level and exiting this reservoir at a low level in the southeast corner. Columns were modelled every 6m in the reservoir.

4 MODEL RESULTS

4.1 2D MODEL RESULTS

4.1.1 RESERVOIR WITHOUT BAFFLE

In the option without a baffle, the 2D model suggests that there is little water movment and turnover of water in the area furthest from the inlet and outlet. It is considered that this is exacerbated by the “T” shape of the chamber.

As shown in Figure 2 and 3 below the long strip located on the eastern side of the reservoir presents significant velocities around the inlet and outlet cells. Velocities are decreasing at mid distance of the reservoir. Streamlines are mainly parallel to the main inlet - outlet line although some “turbulences” can be seen closer to the inlet.

The main chamber of the reservoir shows velocities that are decreasing from east to west when moving away from the main inlet – outlet line. Streamlines are mainly oriented from west to east and from north to south confirming only little exchange with water coming from the inlet pipe and water short-circuiting the main reservoir chamber.

Figure 2 below shows the discharge distribution in the reservoir. The discharge is null for the cells representing the walls of the reservoir.

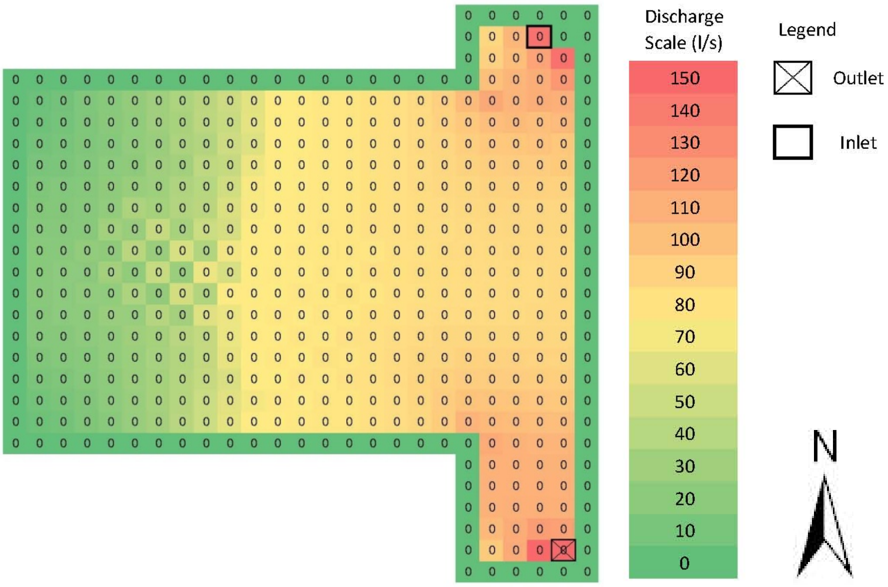

Figure 2: 2D Model - No Baffle – Discharge in Reservoir

Figure 3 below shows the intensity and direction of the velocity vectors in the reservoir. The cells displaying velocity arrows on a white background correspond to the walls of the reservoir where the velocity is null.

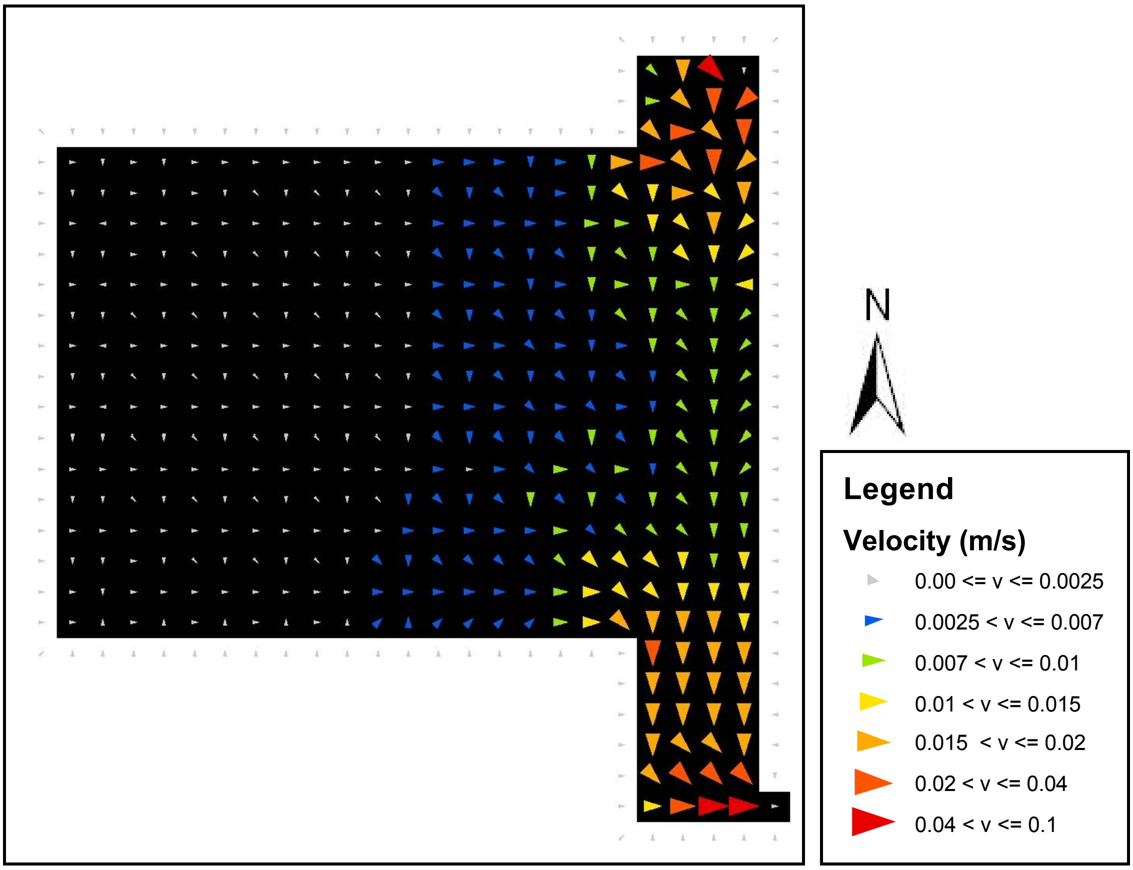

Figure 3: 2D Model - No Baffle – Velocities in Reservoir

4.1.2 RESERVOIR WITH BAFFLE

Inspection of the model output suggests that the construction of a baffle wall in the reservoir would be beneficial in achieving more even flow through the reservoir and minimising the opportunity for “dead areas” to occur, and the opportunity for chlorine residual drops below required levels.

As shown in Figure 4 and 5 below the reservoir presents significant velocities around the inlet and outlet cells. Velocities slightly decrease in the northern half of the reservoir and increases in the southern half of the reservoir located behind the baffle curtain. Four (4) “dead corners” showing low velocities can be observed along the baffle curtain and the western wall of the reservoir. Streamlines are mainly parallel, winding through the reservoir although some “turbulences” can be seen closer to the inlet.

Figure 4 below shows the discharge distribution in the reservoir. The discharge is null for the cells representing the walls of the reservoir.

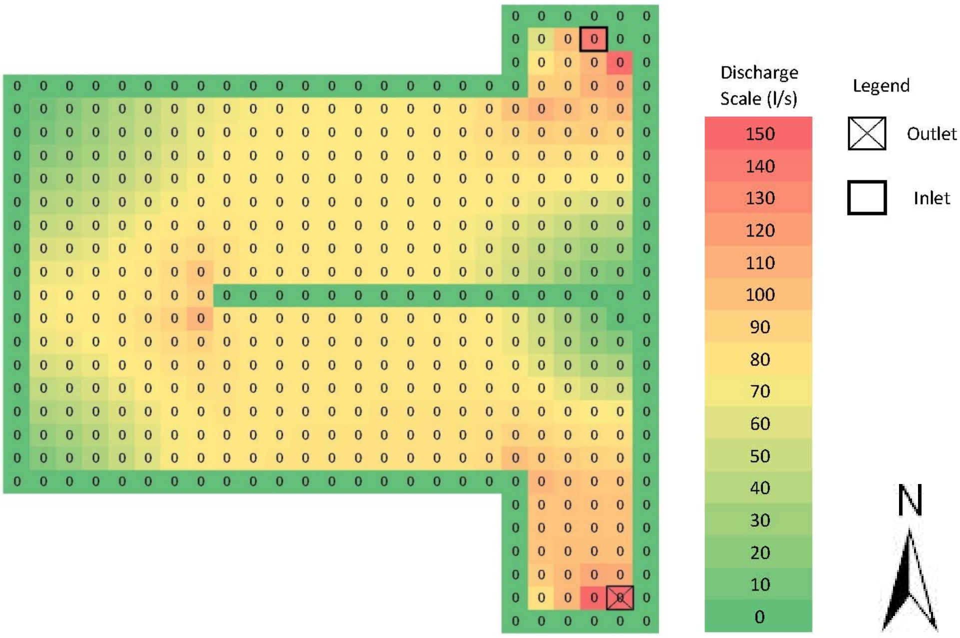

Figure 4: 2D Model - With Baffle – Discharge in Reservoir

Figure 5 below shows the intensity and direction of the velocity vectors in the reservoir. The cells displaying velocity arrows on a white background correspond to the walls of the reservoir where the velocity is null.

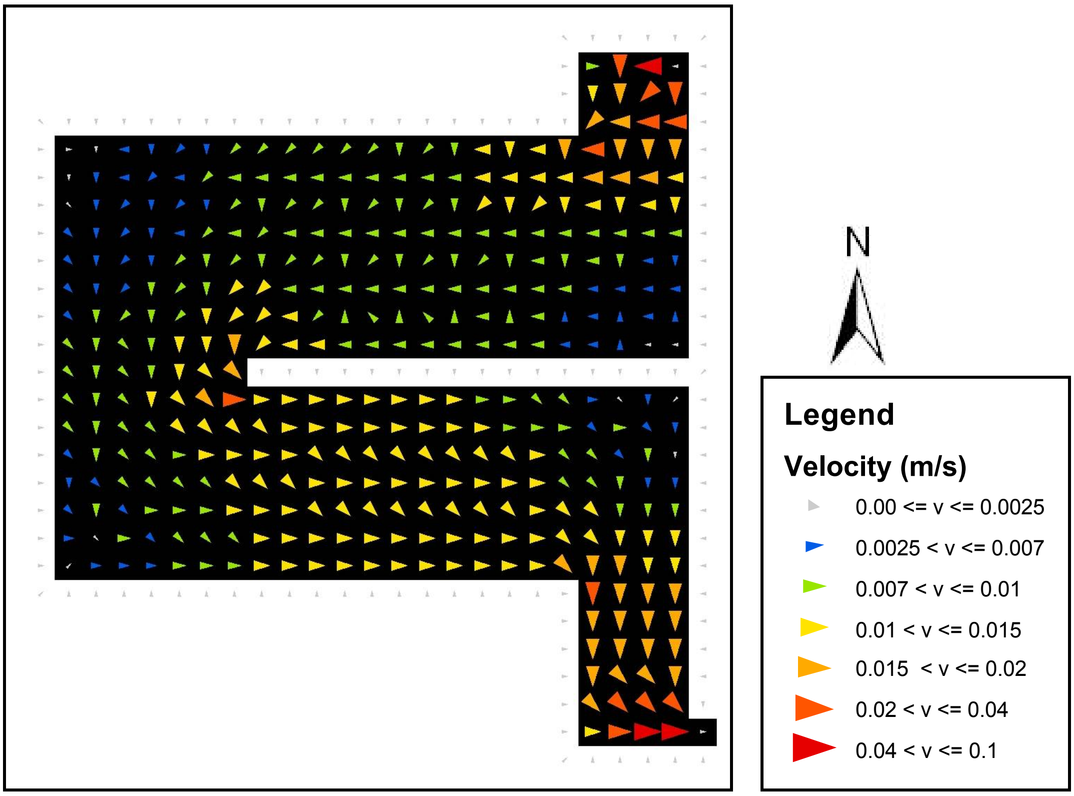

Figure 5: 2D Model - With Baffle – Velocities in Reservoir

4.2 3D MODEL RESULTS

The output graphs indicate water age by colour and velocity by the length of the arrows. Where an arrow passes through the reservoir wall, this indicates the velocity of the water at the start of the arrow, not that the water is exiting the reservoir.

For each case two plots have been produced, showing respectively the bottom of the reservoir and the surface. Contours of "TIME" in hours have been plotted showing the mean age of the water since entry to the reservoir.

The velocity vectors have been superimposed on the age plots to show the flow direction. The velocity vector arrows are limited at a maximum of 1 cm/s, otherwise the vectors near the inlets would be excessively long.

4.2.1 RESERVOIR WITHOUT BAFFLE

The output shows significant recirculation of the water and areas of increased water age (yellow and orange areas). Mixing takes place resulting in an increased average age of water.

As shown in Figure 6 below and similar to the 2D results presented above the long strip located on the eastern side of the reservoir presents significant velocities, especially around the inlet and outlet pipes. In general the main chamber of the reservoir shows low velocities.

Streamlines differ between the bottom and the water surface of the reservoir. Circular streamlines resulting in new water mixing with aging water are observed at the inlet of the reservoir (water surface) and in the main chamber (bottom of reservoir).

The mean age (from initial entry) of water within the reservoir is approximately 50 hours. Aging water is mainly located along the external walls of the main chamber (especially along the south wall).

Figure 6 below shows velocities and water age at bottom and water surface of the reservoir:

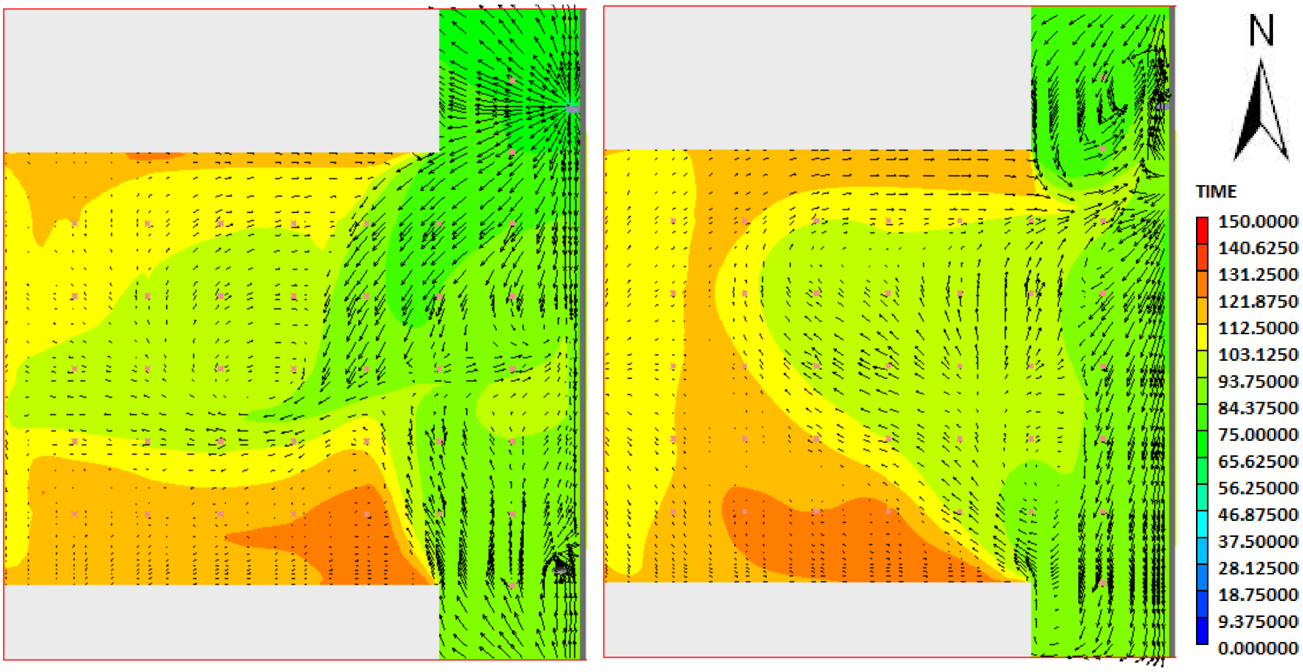

Figure 6: 3D Model - No Baffle - Water Age at Bottom (Left) and Water Surface (Right) of Reservoir

4.2.2 RESERVOIR WITH BAFFLE

This option is considered significantly better than the option without baffle. The central baffle option appears to be beneficial as it provides more consistent flows behind the baffle to the outlet point. This is reflected by the mean average water age and the water age pattern.

As shown in Figure 7 below the side located north of the baffle wall presents significant velocities, especially around the inlet point. The southern side of the curtain shows low velocities.

Here again, streamlines differ between the bottom and the water surface of the reservoir. Circular streamlines are observed at the inlet of the reservoir (water surface) and on the northern side of the baffle wall (bottom and water surface of reservoir).

The mean age (from initial entry) of water within the reservoir is approximately 30 hours. Water is gradually aging towards the outlet point of the reservoir.

Figure 7 below shows velocities and water age at bottom and water surface of the reservoir:

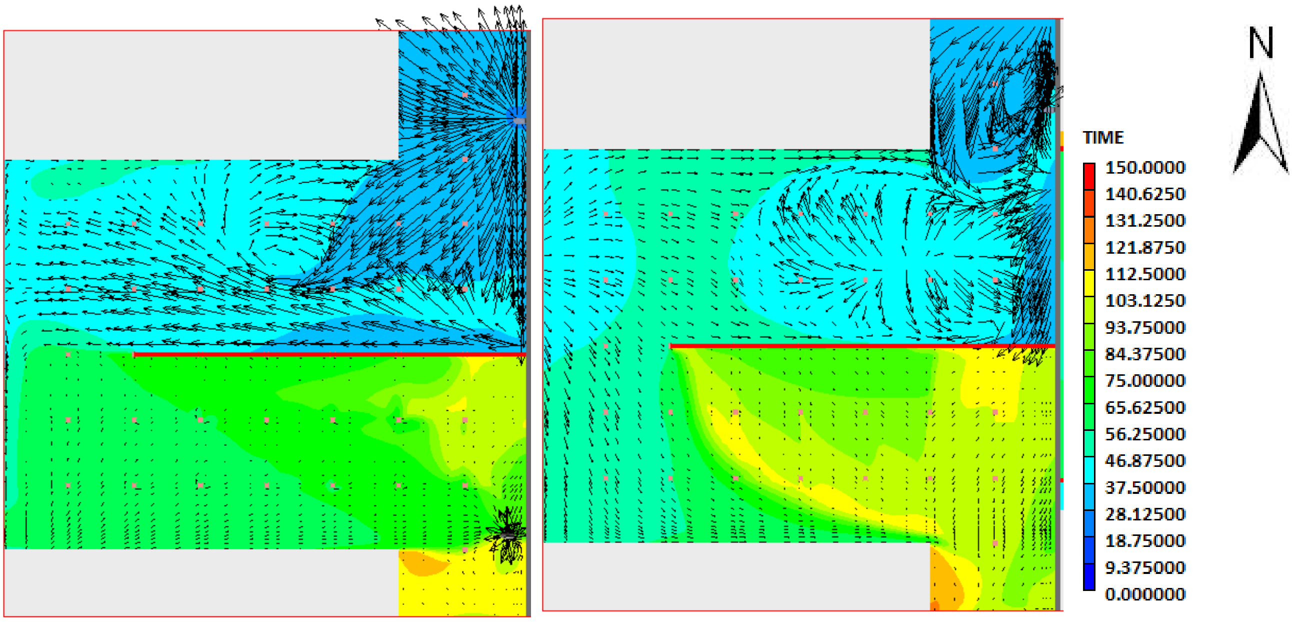

Figure 7: 3D Model - With Baffle - Water Age at Bottom (Left) and Water Surface (Right) of Reservoir

4.3 RESULTS COMPARISON AND DISCUSSION

In terms of option comparison both the 2D and 3D model gave similar results. In both cases the output of the modelling has shown that the installation of baffles minimises ‘dead areas’ and reduces the mean water age in the reservoir. However a detailed comparison of the model results shows differences between the two simulations. At this stage only the velocity results from the two models were compared. Although the 2D model can simulate constituent fate and transport to estimate the age of water this was not undertaken for this study.

The 2D and 3D model runs gave very similar results for the scenario without baffle in terms of velocity range. The two models show that the long strip located on the eastern side of the reservoir presents significant velocities, especially around the inlet and outlet pipes whereas the main chamber of the reservoir shows low velocities. The 3D model predicts the apparition of circular streamlines at the inlet of the reservoir and in the main chamber. The 2D model shows some turbulence in the streamlines at the inlet of the reservoir but the mixing effect in the main chamber is not depicted.

This difference is emphasised in the second scenario results. According to the 3D model the installation of a baffle wall increases the jetting effect observed at the inlet of the reservoir and causes turbulence on the northern side of the reservoir that is not depicted in the 2D model. In the 3D model velocities are greater on the northern half of the reservoir than in the southern half whereas the 2D model shows relatively homogeneous velocities across the reservoir. Another point of difference is that the “dead corners” showing low velocities in the 2D model along the northern side of thebaffle curtain and the western wall of the reservoir are not observed in the 3D model. The dead zone on the southern side of the baffle curtain is shown in both models.

The primary assumption when predicting water path using a 2D model is that flow is uniform in the third dimension, i.e. the depth of the reservoir. The 3D model results show different velocities between the bottom and the water surface of the reservoir which contradicts this hypothesis. This indicates that flows in the reservoir studied strongly diverge from the ideal plug flow pattern where all fluid elements entering the reservoir have a constant velocity and present the same hydraulic residence time.

Successful applications of 2D modelling to predict flow pattern and residence time distribution have been proven in the past for the Calgary Glenmore Northeast clearwell (Yu et al., 2010). In this example the location and size of the dead zones compared well to results of the scale model tracer test. The major difference between the Calgary reservoir and our case study is that water enters the clearwell after moving through a perforated baffle wall which certainly gives a more uniform profile of velocities at the inlet of the reservoir and reduces the jet effect observed in our study.

5 CONCLUSIONS

The objective of the work described in this paper was to investigate the possibility of using 2D open-channel software for modelling flow paths in water tanks. A case study was used to compare the simulation results from a 3D CFD model to those from 2D computations.

Although the general conclusion in terms of option recommendations were the same for the 2D and 3D models, the 3D model identified jetting and mixing effects that simply could not predicted by the 2D simulations. 2D simulations can only be used where flows do not significantly differ in depth. This assumption was proven wrong by the 3D model in the study reservoir which showed significantly different velocities between the bottom and the water surface of the reservoir.

In general 3D models are complex to build and have very long runtimes which can make them prohibitive costs. 2D models can be successfully used as easy first approximations to predict flow paths and water age in a reservoir. However 2D model predictions will only provide a good representation of real flows in water tanks where the reservoir layout does not significantly differ in depth and the primary flow regime through the reservoir closely mimics plug flow. In other conditions 3D CFD modelling is recommended in order to provide detailed solutions that involved fluid flows.

ACKNOWLEDGEMENTS

The authors are thankful for the contributions of Dr David Glynn of Flowsolve Limited, London, who developed the 3D CFD models for our case study reservoir.

REFERENCES

Yu X., Mazurek K.A.,Putz G., and Albers C. (2010) ‘Physical and computational modelling of residence and flow development time in a large municipal disinfection clearwell’ *Canadian Journal of Civil Engineering*.

Ecuru J., Okot-Okumu J., O Okurut T. (2011) ‘Monitoring Residual Chlorine Decay and Coliform Contamination in Water Distribution Network of Kampala, Uganda’ *Journal of Applied Sciences & Environmental Management.*

Martínez-Solano F. J., Iglesias Rey P.L., Gualtieri C., López-Jiménez P. A. (2010) ‘Modelling flow and concentration field in rectangular water tanks’, International Congress on Environmental Modelling and Software.

Glynn D.R. and Shelton A. (2000), ‘Reservoir Design Improvement Using CFD’, *PHOENICS User Conference Proceedings*, Luxembourg, May 2000.

Crowther J. M. and Dandy G. C. (2010), ‘Simulation of water flow and tracer experiments for a clear water storage tank’, *Proceedings of First European Congress of the IAHR*, Edinburgh, UK, 4-6 May 2010, pp.1-6.