

Research Article

Three Dimensional Model of a Fluid and Tracer Transport in Disinfection Contact Tank

Shaikh Mohamed Ashfaq^{†*}

[†]Department of Chemical Engineering, Z.H.C.E.T [AMU], Aligarh, India

Accepted 07 Feb 2015, Available online 10 Feb 2015, Vol.5, No.1 (Feb 2015)

Abstract

Chlorine is mostly used as disinfect in water treatment. Chlorine contact tanks are designed to maximize the contact time between chlorine and pathogens before the water is delivered to consumers. Thus the design objective is to maximize the time that it takes for water to travel from the tank influent to the effluent. The present study is used to investigate the hydraulic performance of the water tank for any dead spots or low velocity zones, resulting in water quality deterioration by using computational fluid dynamics (CFD) 3D model. In case of any discrepancy in the hydraulic performance of the tank, CFD analysis is used to propose modification to improve the hydraulic performance of the tank. In present analysis Tracer studies are used to determine the hydraulic characteristics of the disinfection contact tank. The results from tracer studies are used to obtain accurate estimates of the effective contact time. Residence time distribution (RTD) which is one of the important parameter of the process performance to achieve desire conversion was estimated using CFD tools. The effects of different tank flow condition are studies as well using CFD.

Keywords: CFD; 3D; RTD; Chlorine Contact tank; Tracer; Hydraulic.

1. Introduction

Chemical, biological and physical processes in nature and in engineered systems usually take place in what we call "reactors." Reactors are defined by a real or imaginary boundary that physically confines the processes. Lakes, segments of a river, and settling tanks in treatment plants are examples of reactors. Most, but not all, reactors experience continuous flow in and out. Sequencing batch reactors have a sequence of states including fill, react, and empty. It is important to know the mixing level and residence time in reactors, since they both affect the degree of process reaction that occurs while the fluid usually water and its components often pollutants pass through the reactor. Fluid flows are inherently complex and governed by equally complex equations. This is certainly the case for many fluid flow problems seen in the water & waste water treatment industry. The rapid acceleration of computer technology in recent decades has resulted in computer hardware and software that is now capable of numerically solving these equations and problems to time scales that fit within project schedules. The science of using computer programs to solve fluid flow problem is known as Computational Fluid Dynamics (CFD). The CFD modeling was used to

investigate the hydraulic performance of the disinfect contact tank for any dead spots or low velocity zones, resulting in water quality deterioration. In case of any discrepancy in the hydraulic performance of the contact tank, CFD analysis was to propose modification to improve the hydraulic performance of the disinfect contact tank. The purpose of this article is to summaries the results of the CFD modeling analysis of disinfect tank. The analysis considered three different flow scenarios in the tank.

2. CFD Modeling of the disinfect tank

2.1 Geometry

Shiono et al. (1991) conducted physical experiments on a tank that has been numerically modeled in this study. The model rectangular tank was a hydraulically scaled replica of the EMBSAY Chlorine Disinfect contact tank - an underground basin sited in West Yorkshire, U.K., and owned and managed by Yorkshire Water plc. The scale ratios of the tank were 1:8, both horizontally and vertically. Three dimensional model of the disinfectant tank was constructed for this analysis based on the dimension of the physical model. **Fig1.** Shows the overall view of the three dimensional (3D) model generated for disinfectant tank.

*Corresponding author: Shaikh Mohamed Ashfaq

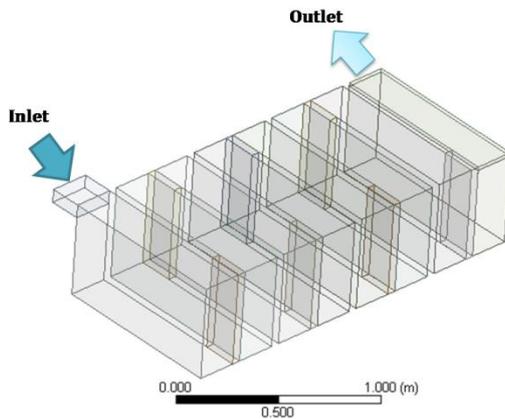


Fig. 1 Three dimensional view of the model generated for disinfectant tank

2.2 Modeling

General Details of the CFD model are as follows

2.2.1 Modeling Approach

Flow in the disinfectant tank is categorized as free surface flow. There are different techniques available within CFD modeling platforms by which this flow condition can be simulated. Considering the little variation in the free surface of the water, the water surface was considered to be steady for the analysis purpose. The software code 'Fluent v14.0' was used for this analysis. The standard $k - \epsilon$ model is used in this simulation. It is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ϵ). In the derivation of the $k - \epsilon$ model, it is assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard $k - \epsilon$ model is therefore valid only for fully turbulent flows. The CFD code fluent v14.0 solver was setup with this turbulence models to obtain the results. For the disinfect contact tank geometry, a grid with approximately 250,000 cells was constructed with cell size ranging from 0.03 m to 0.01 m in order to get adequate resolution of the geometric features. No-slip conditions were prescribed on all solid surfaces, and all the walls in the disinfect contact tank. The fluid domain downstream of the disinfect contact tank was restricted up to the outlet of the disinfect contact tank. To predict the behaviour of the disinfect contact tank, Residence Time Distribution (RTD) analysis was approached. RTD is one of the ways to characterize the non-ideal mixing in the disinfect contact tank. This technique includes the injection of a small amount of the tracer at the inlet stream and the observation of the corresponding response at the exit stream.

2.2.2 Assumptions

The following assumptions were made while studying the performance of the disinfect contact tank. Water is considered to be isothermal, and none of its properties

change during the simulation. The inflow to the disinfect contact tank and outflow from the contact is continuous and steady. Three flow consider are 0.585 kg/s inflow for low flow scenario, 1.170 kg/s inflow for average flow scenario and 2.340 kg/s inflow for high flow scenario. The free surface level of the water has no major variation.

3. Results and Discussion

The following subsections discuss the results of the CFD simulation for the low flow, average flow and high flow scenario and respective figures are included to illustrate the hydraulic performance of the disinfect contact tank at top sections in the simulation domain. The contours of the velocity are coloured by the velocity magnitude and the velocity scale (m/s) is shown on the left side on each figure. The colour bar located to the right side of each velocity contour shows the velocity magnitude ranging from 0.00 m/s as blue to 0.05 m/s as red. Velocities higher than 0.05 m/s are seen as red in each velocity contour.

3.1 Velocity Contours

The velocity contours is plotted at top water levels in order to understand the flow patterns in the disinfect contact tank. Fig 2 to Fig 4 shows the velocity contour at a top level location in the disinfect contact tank at different flow condition. However, there are some dead zones with velocities lower than this range. These zones are scattered throughout the area at this cross sectionsome circulation zones with low velocity in the domain which indicates the existence of the dead zones.

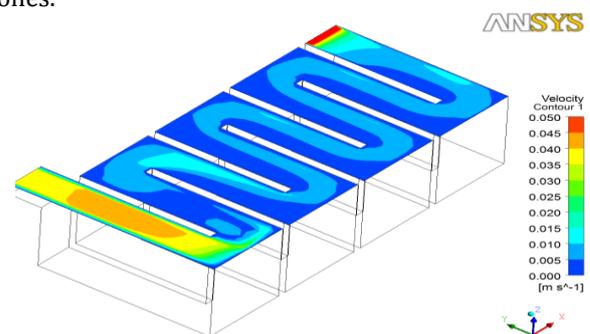


Fig. 2 Velocity contour for 0.585 kg/s inflow

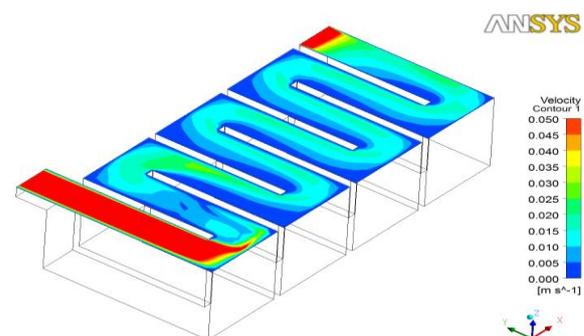


Fig. 3 Velocity contour for 1.170 kg/s inflow

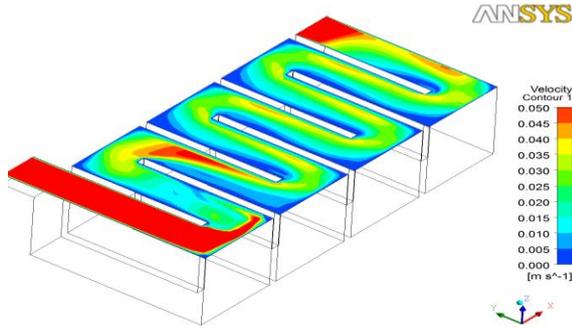


Fig. 4 Velocity contour for 2.340 kg/s inflow

3.2 Streamline

Shows streamline coloured by time in the disinfect contact tank. As seen from the fig 5 to fig 7, the flow from the inlet creates strong currents towards the outlet and flow moves towards the outlet side and the flow tend to move downward creating circulations in the disinfect contact tank.

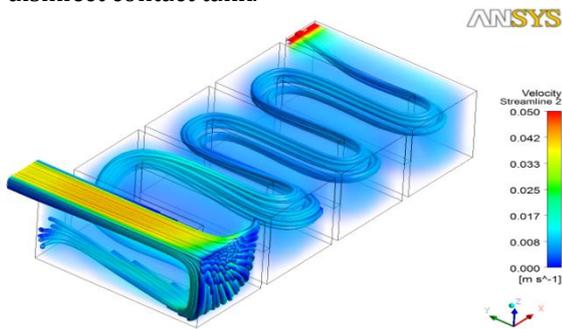


Fig. 5 Streamline for 0.585 kg/s inflow

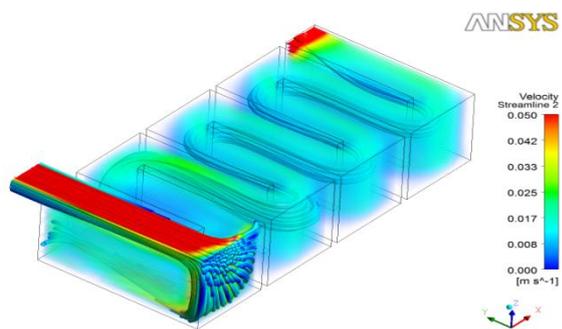


Fig. 6 Streamline for 1.170 kg/s inflow

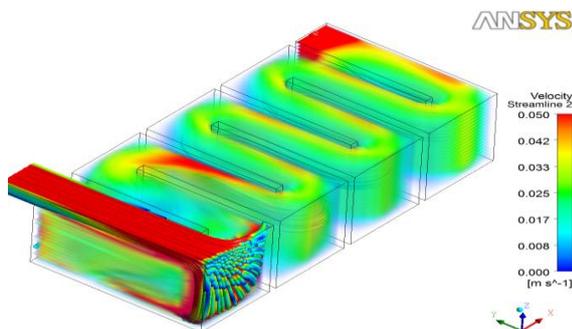


Fig. 7 Streamline for 2.340 kg/s inflow

3.3 Analysis of Residence Time Distribution

The residence time distribution (RTD) is one of the ways to characterize the non-ideal flow in tanks and channels. This technique includes the injection of a small amount of the tagged inert tracer at the inlet stream and the observation of the corresponding response at the exit stream.

The disinfect tank configurations can be determined by evaluating the Axial Dispersion coefficient, D expressed as a dimensionless number, the Vessel Dispersion Number, D/uL. To model real reactors which deviates from the ideal reactors various models are available to represent the behavior of fluid flow and mixing in the process equipments. One such model which is used here is Axial Dispersion model. The model assumes diffusion like process superimposed on plug flow. This can be represented by dispersion or longitudinal dispersion to distinguish from molecular diffusion. The dispersion coefficient represents the spreading of the tracer which is injected at the inlet in the reactor.

1. Large D means rapid spreading of the tracer curve
2. Small D means slow spreading
3. D = 0 means no spreading, hence plug flow

We evaluate D or $\frac{D}{uL}$ by recording the shape of the tracer curve as it pass the exit of the vessel. In particular, we measure mean time of passage.

\bar{t} = mean time of passage, or when the curve passes by the exit

The mean, for continuous or discrete data, is defined as

$$\bar{t} = \frac{\int_0^{\infty} t C dt}{\int_0^{\infty} C dt} = \frac{\sum t_i C_i \Delta t_i}{\sum C_i \Delta t_i}$$

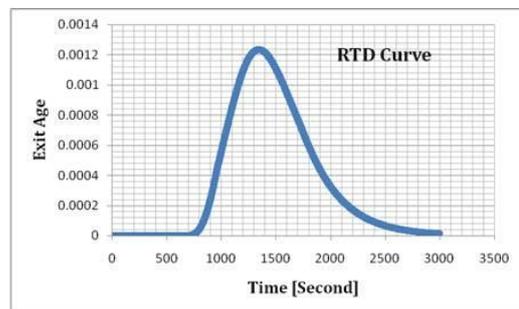


Fig.8 RTD curve for Low flow scenario

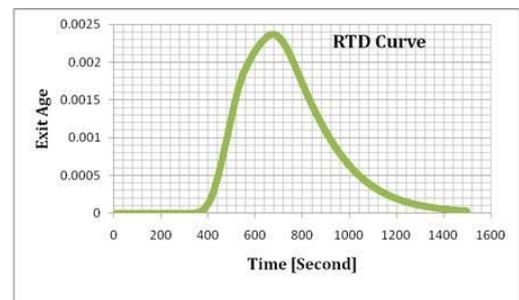


Fig. 9 RTD Curve for Average flow scenario

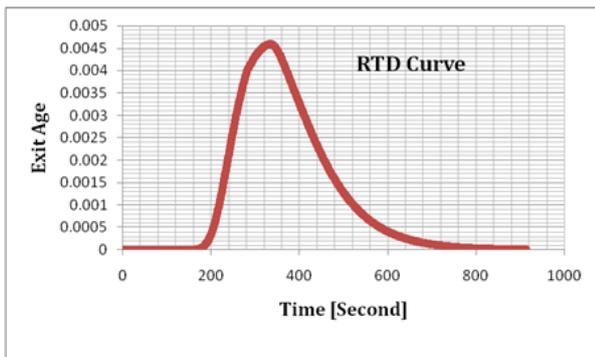


Fig. 10 RTD Curve for High flow scenario

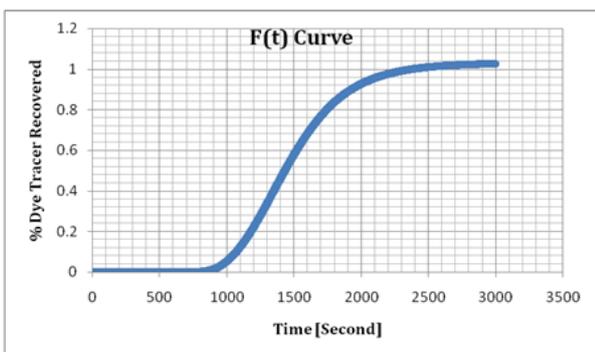


Fig. 11 Cumulative distribution curve for Low flow scenario

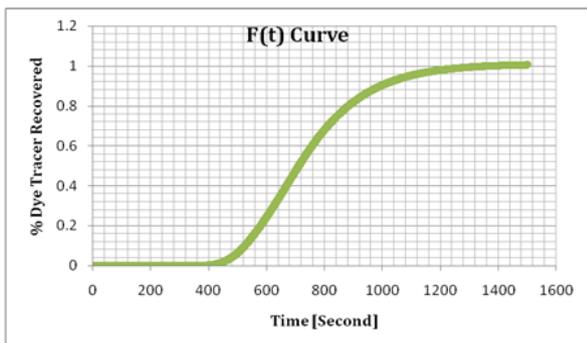


Fig. 12 Cumulative distribution curve for Average flow scenario

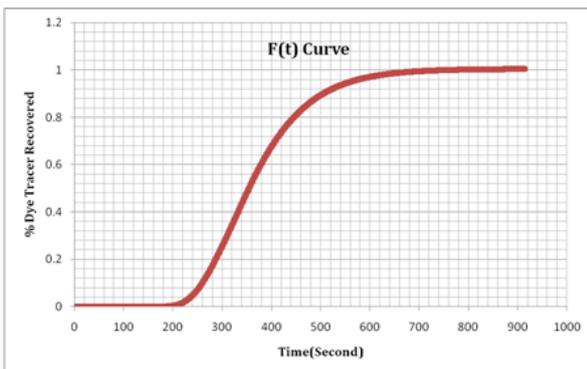


Fig.13 Cumulative distribution curve for High flow scenario

The Tracer CFD Simulation reproduces the passage of a tracer through the disinfect contact tank. Fig 8 to Fig 13 show the tracer results obtained for the three flow scenario. The T10, the time for 10% of the tracer to pass through the tank, can be evaluated by plotting the concentration versus time of the tracer at the outlet of the tank. The T10 for the low flow scenario is 879 second. The hydraulic efficiency baffling efficiency of the tank can be calculated by dividing the T10 time by the theoretical retention time for the tank. The ratio of the T10/HRT is 0.5945. Similarly for average the ratio is 0.5844 and for high flow is 0.5742.

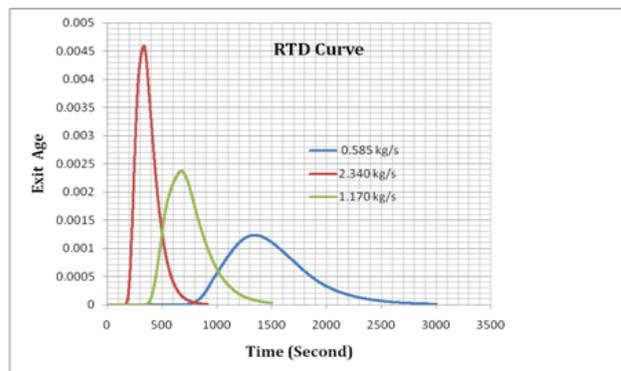


Fig. 14 Comparison Residence time distribution curve

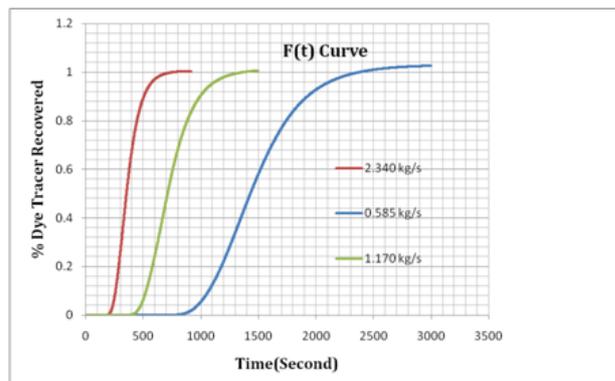


Fig. 15 Comparison of Cumulative distribution curve

Minimum detention time for 90% of the flow

Table 1 Parameters obtained from CFD analysis

S. No	Parameters	Condition		
		Low Flow scenario	Average Flow scenario	High Flow scenario
1	Mass flow rate (kg/s)	0.5839	1.1678	2.3356
2	Volume of tank(m3)	0.86325	0.86325	0.8632
3	HRT (Second)	1478.42	739.214	369.6
4	CFD(MRT) (Second)	1362.5	742.418	372.589
5	T10 (Second)	879	432	212
6	T90 (Second)	1926	993	505
7	Baffling factor T10/HRT	0.5945	0.5844	0.57452

Conclusion

Following the CFD analysis, the following conclusions can be made.

1. The steady state simulation was carried out for the three dimensional model.
2. The theoretical residence time for the low flow scenario is 1478.42 second. The values of T_{10} and T_{90} are 879 second and 1926 second respectively. The nature of the curve is improved than average and high flow condition.
3. The theoretical residence time for the average flow scenario is 739.214 second. The values of T_{10} and T_{90} are 432 second and 993 second respectively. The curve is tighter and has an initial peak indicating a short circuiting tendency of the disinfect contact tank.
4. The theoretical residence time for the high flow scenario is 369.60 second. The values of T_{10} and T_{90} are 212 second and 505 second respectively. The curve is tighter and has an initial peak indicating a short circuiting tendency of the disinfect contact tank.
5. The CFD mean residence time values for low, average and high flow are 1362.5 second, 742.418 second and 372.589 second. Are close to the theoretical value of the hydraulic residence time for low, average and high flow are 1478.42 second, 739.214 second and 369.60 second.
6. From table 1 it can be seen that low flow scenario provides an improved performance over the average and high flow scenario with a lower tracer concentration at the outlet and a T_{10} time of 879 second.

Nomenclature

\bar{t}	Mean time of passage, Second
HRT	Hydraulic Residence Time, Second
MRT	Mean Residence Time, Second
T_{10}	10% of the flow passed, Second
T_{90}	90% of the flow passed, Second
D	Diffusion coefficient, m^2/s
U	Velocity, m/s
L	length of the reactor, m
C	Concentration of Tracer, kg/m^3
C(t)	Concentration function of time, kg/m^3
C_i	Tracer concentration at time i, kg/m^3
k	Turbulence kinetic energy, m^2/s^2
Δt_i	Time increment, Second
ϵ	Dissipation rate, m^2/s^3

Acknowledgment

The author would like to thanks Almighty Allah for giving the wonderful life. Theauthor would also like to thank Dr. Koji Shiono of the University of Loughborough for making available their experimental data. The author is also grateful to his parents.

References

- Shiono, K, Teixeira, E.C, Falconer, R.A (1991). Turbulent measurements in chlorine contact tank. In: *Proc. first international conference on water pollution: Modelling, measuring and predicting*. Southampton, UK: Wessex Institution of Technology: 519-531.
- Shiono, K, Texeira, E.C (2000), Turbulent characteristics in a baffled contact tank, *Journal of Hydraulic Research (JHR)*, Vol 38 No.6, pp 403-416.
- Stevenson D.G (1995), The design of disinfection contact tanks, *Journal of Chartered Institution of Water Environment Management (CIWEM)*, Vol.9 No.2, pp 146-152.
- Wang H. and Falconer R.A (1998a), Simulating disinfection processes in chlorine contact tanks using various turbulence models and high-order accurate difference schemes, *Water Research*, Vol. 32 No.5, pp 1529-1543.
- Wang H. and Falconer R.A (1998b), Numerical modeling of flow in chlorine disinfection tanks, *Journal of Hydraulic Engineering ASCE*, Vol.124 No.9, pp 918-931.
- Wang H., Shao X. and Falconer R.A (2003), Flow and transport simulation models for prediction of chlorine contact tank flow-through curves, *Water Environment Research*, Vol. 75 No.5, pp 455-471.
- Falconer R.A. and Liu S.Q (1987), Mathematical model study of plug flow in a chlorine contact tank, *Journal Water Environmental Management*, Vol. No.1, pp 279-290.
- Falconer R.A. and Ismail A.I.B.M (1997), Numerical modeling of tracer transport in a contact tank, *Environmental International*, Vol. 23 No.6, pp 763-773.
- Gyurek L.L. and Finch G.R (1998), Modeling water treatment chemical disinfection kinetics, *Journal of Environmental Engineering ASCE*, Vol.124 No.9, pp 783-793.
- Hannoun I.A. and Boulos P.F (1997), Optimizing distribution storage water quality: a hydrodynamic approach, *Journal of Applied Mathematical Modeling*, Vol.No.21, pp 495-502.
- Falconer, R.A.; Liu, S (1988). Modelling solute transport using QUICK scheme. *J. Environ. Eng., ASCE*, Vol.114 No.1: pp 3-20.
- Fischer, H.B (1973). Longitudinal dispersion and turbulent mixing in open channel flow. *Ann. Rev. Fluid Mech*, Vol.No. 5: pp 59-78.
- French, R.H. Open-channel hydraulics (1994). *New York, NY: McGraw-Hill, Inc.*, pp 1-739.
- H.Scott Fogler (2005), Elements of Chemical Reaction Engineering, *Prentice-hall of India*, pp 809-870.
- Octave Levenspiel (1999), Chemical Reaction Engineering, *John Wiley & Sons*, pp 294-296 Fluent v 14.0 - User Manual.