

Using Hydraulic Modeling With Solar Energy To Optimize The Chlorine Contact Tanks

Ali Hadi Ghawi

Al-Qadissiya University , College of Engineering

Abstract

Traditionally, contact time of a chlorine contact tank (CCT) in waster water treatment plants is determined by performing field tracer tests. This paper presents an alternative approach by using computational fluids dynamics (CFD) modeling to predict the contact time. This method is known as numerical tracer testing. Numerical tracer testing was used for determining the contact time at the Al-Dewanyia Wastewater Treatment Plant in Al-Dewanyia The CCT is of 1600 m³ volume, five-pass baffled tank capable of handling up to 1000 m³/ hr . CFD results were compared to field data obtained by a step-feed field tracer tests using Rhodamine-wt. The comparison shows that CFD modeling is capable of accurately predicting contact times in the CCT. Results of CFD modeling, including flow streamlines, velocity contours and vectors, were used to improve velocity distribution and reduce solids deposition, short-circuiting and vortices with the inclusion of SolarBee mixers. The CFD modeling has clearly illustrated the short-circuiting phenomenon that occurs without the SolarBee mixer in place. SolarBees can be effectively used to prevent short-circuiting by being placed directly in the short-circuiting path. The study demonstrates that CFD modeling is a valuable modeling tool for wastewater industry.

Keyword: contact tank, CFD, RTD, SolarBee mixer, tracer, wastewater.

الخلاصة

من الحقائق العلمية الشائعة في حساب زمن بقاء الكلور في حوض التعقيم في محطات معالجة مياه الصرف الصحي هو اختبار اضافة صبغة في الحوض. في هذا البحث تم استخدام موديل رياضي يسمى ديناميكا السوائل الحسابية لغرض حساب زمن بقاء الكلور في حوض التعقيم. هذه الطريقة تسمى الطريقة العددية لأضافة الصبغة في حوض التعقيم. تم استخدام الطريقة العددية لأضافة الصبغة لغرض دراسة إمكانية تحسين أداء حوض التعقيم في محطة معالجة مياه الصرف الصحي في مدينة الديوانية. ان حجم حوض التعقيم في محطة معالجة مياه الصرف الصحي في مدينة الديوانية هو 1600 م³ و يتكون من خمسة مصدات (جدران) بتصريف أكثر من 1000 م³ ساعة. تم مقارنة نتائج الموديل الرياضي مع التجارب الحقلية باستخدام صبغة Rhodamine-wt. المقارنة أثبتت دقة الموديل الرياضي في ايجاد زمن بقاء الكلور في الحوض. نتائج الموديل الرياضي شملت خطوط الجريان و السرعة التي استخدمت لتحسين اداء حوض التعقيم بتحسين توزيع السرعة و تقليل الدوامات والمناطق المعدومة السرعة بعد استخدام خلاطات تعمل بالطاقة الشمسية وزعت على امتداد حوض التعقيم. و في هذه الدراسة تم ايجاد تأثير اضافة وعدم اضافة خلاطات تعمل بالطاقة الشمسية على تقليل الدوامات و زيادة كفاءة حوض التعقيم. اثبتت هذه الدراسة فعالية النموذج الرياضي المستخدم في هذه الدراسة لتصميم محطات معالجة مياه الصرف الصحي.

Introduction

Disinfection is the last process in wastewater treatment process and involves the inactivation of pathogenic microorganisms. This inactivation means biochemical alteration of microorganism to prevent its replication or eliminate activity. Chlorination is the most widely practiced disinfection process for water and waste water treatment. This is primarily done in disinfection chambers where sufficient contact time is provided between treated waste water and chlorine. The US EPA determines the effectiveness of these contactors for disinfection by the CT method, where C is the concentration of the disinfectant at the outlet of tank and T is the T₁₀ value (The time required for 10% of the liquid to leave the tank, or the time at which 90% of the liquid is retained in the tank and subject to at least disinfection level of C). The contactor hydraulic efficiency can be measured by the ratio of T₁₀ and theoretical hydraulic residence time. The configuration of baffles, inlet and outlet conditions, length and width of compartments influence this hydraulic efficiency. Computational

Fluid Dynamics (CFD) analysis provides the better understanding fluid flow phenomena that affect mixing, short circuiting, particle dispersion and other critical issues. It helps to eliminate different configuration of the contactor which are inefficient and evaluate designs in terms of mixing characteristics. It can help to reduce the cost of chlorine by increasing efficiency of its disinfection reservoirs. One of the greatest challenges in designing waste water treatment equipment is that its large size makes it very expensive and time consuming to perform physical experiments (**Hannoun et al. (1998)**).

Tracer test is conducted in full scale plants to calculate this residence time distribution (RTD). The entire RTD can be used to predict the microbial inactivation in the disinfection chamber. Another alternative of this full scale tracer test is to simulate it using CFD models and obtain RTD which will thus same time and money. Only a few CFD studies have been published. The studies of **Hannoun et al. (1998)**, **Crozes et al. (1999)** and **Reddy (2000)** focused on improving existing tanks through the addition and placement of baffles. **Crozes et al. (1999)** demonstrated fair agreement between CFD and experimental results. They indicated that modest gains in hydraulic efficiency can be achieved through inlet/outlet piping arrangement, while significant improvements can be achieved through the addition of multiple plain or perforated baffles. **Chataigner et al. (1999)** compared CFD results with experimental RTD curves and achieved good agreement using conventional turbulence models such as the k- ϵ model. Errors of between 6.7% and 9.3% were reported for the prediction of T_{10} , T_g and T_{90} values. The agreement is sufficient to use CFD not only for qualitative, but also quantitative predictions.

Other CFD based disinfection studies have also been performed by **Chiu et al. (1999)** on UV disinfection channels. A number of studies have also been performed on ozone contact tanks such as **Henry and Bennett (1996)** and **Brouckaert et al. (2000)**. None of the studies integrated the hydraulics with the disinfectant decay and the disinfection of organisms. Two steps were involved in this study. The first part was conducted at the Al-Dewanyia Wastewater treatment plant (DWWTP) where tracer test was conducted using Rhodamine-wt to determine the residence time distribution for the chamber. In the second part we simulated the tracer test using Fluent and predicted the residence time distribution. The aim of this study was originally stated as to improve the operation and performance Al-Dewnyia Wastewater Treatment Plants by improving Contact Tank which have been identified as operating poorly, which is achieved by predicting the existing flow distribution of the Contact Tank using Computational Fluid Dynamics (CFD) techniques. The special objective of this study is computational fluid dynamics (CFD) analysis of existing chlorine contact chamber in DWWTP using Fluent 6.3, predict its residence time distribution and compare it with the experimental results.

Materials And Methods

Residences-Time Distribution (RTD) Model

Residence time distribution (RTD) analysis provides means of assessing the homogeneity of water age in a tank. A plot of the fraction of water outlet as a function of time can be used to assess the extent of short-circuiting and the existence of low flow zones in the tank. Traditionally, this has been measured by conducting a physical tracer study on the tank.

The following background equations and definitions are used in the proposed model. Considering a volume V and constant flowrate Q , the theoretical mean residence time T is given by:

$$T = \frac{V}{Q} \dots\dots\dots (1)$$

The normalized residence – time distribution function $E(t)$ after an instantaneous (pulse) tracer input is defined as **(Levenspiel, 1999)**:

$$E = \frac{C_i}{\int_0^\infty C dt} \approx \frac{C_i}{\sum_{i=1}^n C_i \Delta t_i} \dots\dots\dots (2)$$

Where t denotes time, and C_i is the tracer concentration measured at the outlet at time t_i . An alternative to the pulse tracer study is the step tracer study. In this case, at $t = 0$ the tracer is “switched on” at the tank inlet and remains “on” during the study. The normalised concentration of the tracer measured at the tank outlet under steady-state operation is R (Eq. 3). The relationship between E and R is given by Eq. 4. **(Scott, 1999)**:

$$R = \frac{C_i}{\int_0^\infty C dt} \approx \frac{C_i}{\sum_{i=1}^n C_i \Delta t_i} \dots\dots\dots (3)$$

$$R = \int_0^\infty E dt \dots\dots\dots (4)$$

While either a pulse or step physical tracer study can be used to determine the RTD of an operational tank, a physical tracer study can only be undertaken for operating conditions which are feasible in the physical tank. Other scenarios, such as the performance of the tank for operating conditions outside normal operating conditions, or the impact on the flow of physical tank modifications, cannot be assessed in advance using this method. An equation was solved for “residence time”, this being the mean age in the water in each cell since its entry to the tank. Contour plots of this variable give a good indication of the level of water mixing, and whether there are any stagnant zones.

CFD Analysis

The flow inside a contact tank presents usually the feature that the variations of all relevant quantities in the vertical direction, except in the thin boundary layer near channel bottom and possibly near the free surface, are substantially smaller than variations across the width or in stream wise direction. Thus, two- dimensional or depth-averaged models may be applied to describe hydrodynamics and mass transfer processes. These CFD models are based on the mass conservation equation and the Navier-Stokes equations of motion. Since the flow in the tank is turbulent, these equations must be averaged over a small time increment applying Reynolds decomposition, which results in the Reynolds-averaged Navier-Stokes equations (RANS). For a planar, incompressible flow these equations are **(Hjertager et al., 2002)**:

$$\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} = 0 \quad \dots\dots\dots 5$$

$$\bar{\rho} \left(\frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} \right) = \bar{\rho} g_x - \frac{\partial \bar{p}}{\partial x} + (\mu + \rho \nu_t) \nabla^2 \bar{u} \quad \dots\dots\dots 6$$

$$\bar{\rho} \left(\frac{\partial \bar{v}}{\partial t} + \bar{u} \frac{\partial \bar{v}}{\partial x} + \bar{v} \frac{\partial \bar{v}}{\partial y} \right) = \bar{\rho} g_y - \frac{\partial \bar{p}}{\partial y} + (\mu + \rho \nu_t) \nabla^2 \bar{v} \quad \dots\dots\dots 7$$

where ρ and μ are fluid density and viscosity, p is fluid pressure and u, v are velocity components in the x and y directions, respectively. The overbar indicates time-averaged quantities. Notably, in eq. (3) there is the turbulent kinematic viscosity ν_t , that if isotropic turbulence assumption holds could be estimated following the k - ε model approach as:

$$\nu_t = \frac{C_\mu k'^2}{\varepsilon'} \quad \dots\dots\dots 8$$

where k' and ε' are turbulent kinetic energy per mass unit and its dissipation rate, respectively, and $C_\mu=0.09$ (Hjertager et al.,2002). These parameters are estimated with the classical two equations of k - ε model:

$$\frac{\partial k'}{\partial t} + \bar{V} \cdot \nabla k' = \frac{\nu_t}{\sigma_k} \nabla^2 k' + 2\nu_t \bar{D} \bar{D} \quad \dots\dots\dots 9$$

$$\frac{\partial \varepsilon'}{\partial t} + \bar{V} \cdot \nabla \varepsilon' = \frac{\nu_t}{\sigma_\varepsilon} \nabla^2 \varepsilon' + 2\nu_t C_{1\varepsilon} \frac{\varepsilon'}{k'} \bar{D} \bar{D} - C_{2\varepsilon} \frac{\varepsilon'^2}{k'} \quad \dots\dots\dots 10$$

where D is deformation tensor, whereas $C_\mu, \sigma_k, \sigma_\varepsilon, C_{1\varepsilon}$ and $C_{2\varepsilon}$ are constants, and their values are listed in **Table 1**. Transport of solutes within the contact tank could be modelled using the 2D advection-diffusion equation:

$$\frac{\partial \bar{C}}{\partial t} + \bar{u} \frac{\partial \bar{C}}{\partial x} + \bar{v} \frac{\partial \bar{C}}{\partial y} = D_t \left[\frac{\partial^2 \bar{C}}{\partial x^2} + \frac{\partial^2 \bar{C}}{\partial y^2} \right] \quad \dots\dots\dots 11$$

where molecular diffusion was neglected because it is too small and only turbulent diffusion was considered with D_t as turbulent diffusivity and C as solute concentration.

Table 1: Recommended typical values of the constants in the k - ε turbulence model (Hjertager et al.,2002).

C_μ [-]	$C_{2\varepsilon}$	$C_{1\varepsilon}$	σ_ε [-]	σ_k [-]
0.09	1.92	0.1256	1.6	1.0

These equations were solved using FLUENT modeling package (FLUENT 6.3, 2006). FLUENT can solve for the same flow domain both motion equations and advection-diffusion equation. Particularly, both the $k-\varepsilon$ model application mode and the advection-diffusion application mode were used. They solve Eqs. from (5) to (11) for the pressure p , the velocity vector components u and v , $k-\varepsilon$ model parameters and solute concentration C within the domain of the flow (FLUENT, 2006). FLUENT was used to simulate the flow through contactor. FLUENT uses finite volume method to solve the differential equations governing fluid flow.

The design drawings of the chlorine disinfection tank was collected from Al-Dewanyia wastewater treatment plant (**Fig. 1**). The layout of the Al-Dewanyia wastewater treatment plant can be summarised by the following stages,

- screening
- grit removal
- aerated eration system
- secondary clarification
- disinfection before discharging into Al-Dewanyia river.
- sludge thickening (sludge from secondary stage)
- sludge digestion (sludge from primary and secondary stages)
- drying bed dewatering

Chlorination Tank being rectangular in shape, Cartesian co-ordinate system used to define various points on the tank. Gambit used as a pre-processor for modeling the tank. The modeled was developed from top to bottom i.e. volumes were generated first by subtracting, adding, splitting finite volumes. Volumes were added to obtain a final geometry of the tank and then surfaces, edges and nodes were created on it. Uniform meshing was applied to the whole domain. Hexagonal elements were used except for the corners. A schematic plan of hydraulic CCT is shown in **Fig. 2**.

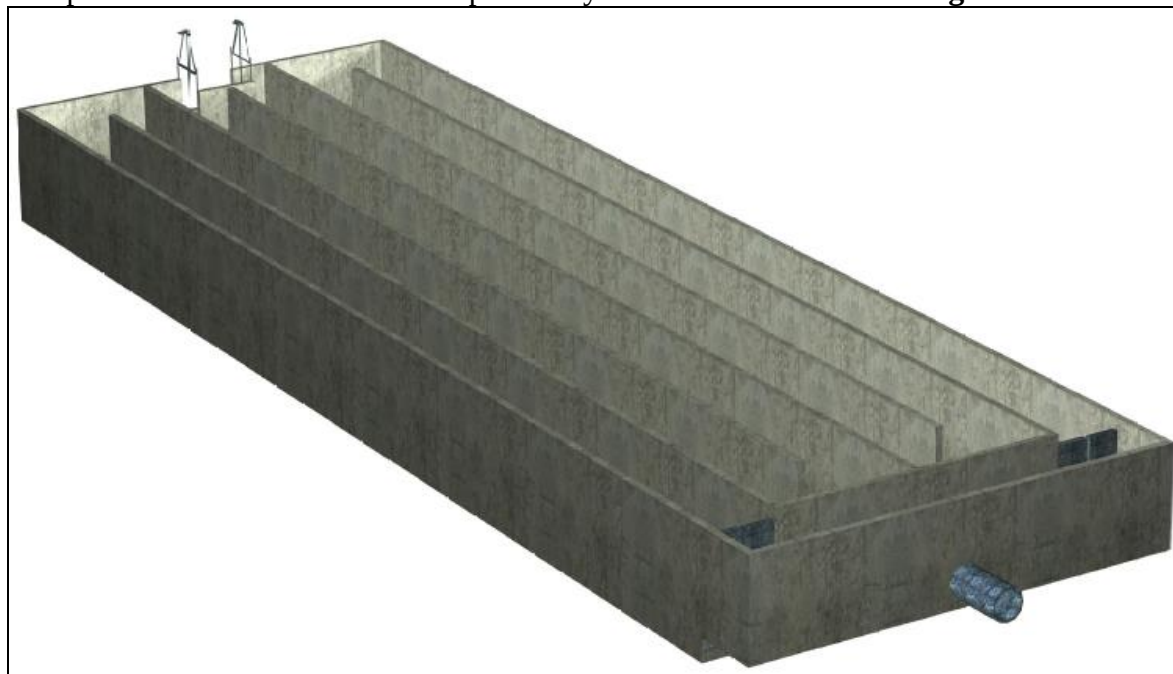


Fig. 1. Schematic of Chlorine Contact Chamber

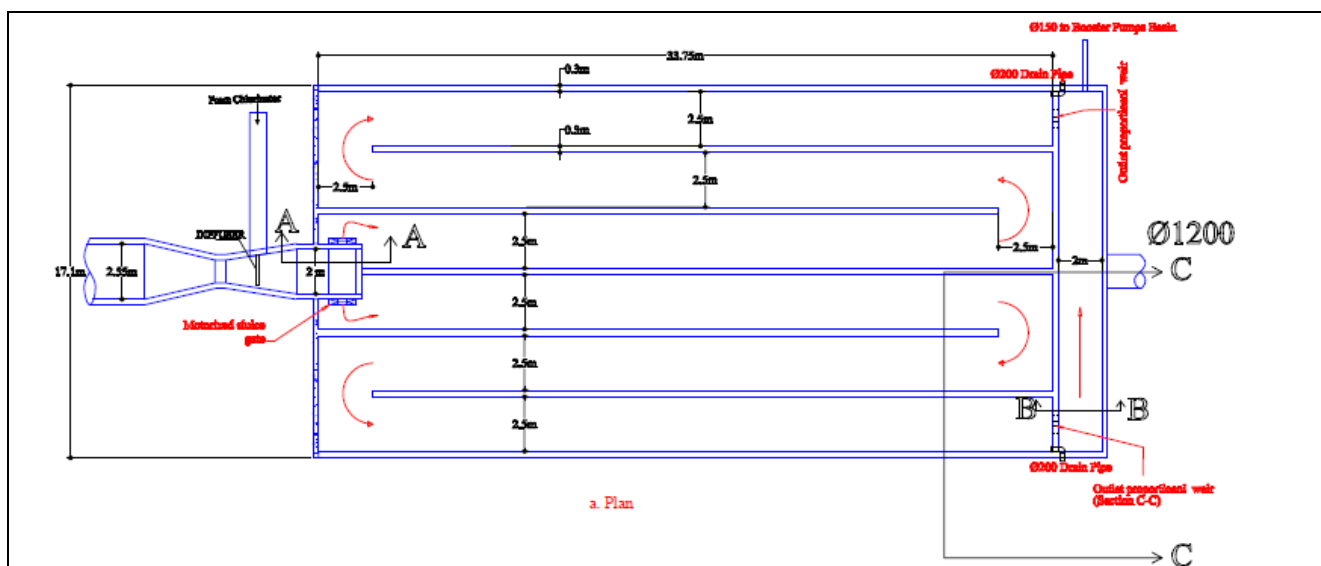


Fig. 2. Outline of Chlorine Contact Chamber

Boundary Conditions:

The inlet boundary had uniform flux across the whole surface. The inlet velocity as calculated from the flow coming in was used in the negative-x direction. The inlet turbulence intensity (Turbulent Kinetic Energy) and turbulence length scale were calculated by:

$$k = \text{kinetic energy} = 0.0026 \text{ m}^2/\text{sec}^2$$

length scale = $0.1644 (k)^{3/2} / (0.1 * \text{entrance width})$ m where k is the kinetic energy as calculated in the first step.

Fluent is very powerful in modeling fluid flow in disinfection tanks. The solution adaptive grid capability is particularly useful for accurately predicting flow fields in regions with large gradients. The neutral mesh file generated using gambit was imported in fluent.

Grid check is applied to see if there's any error in geometry. The minimum volume obtained came out to be positive. Water was used as a fluid material with a density of 998.2 kg/m^3 and viscosity of $0.001003 \text{ kg/(m.s)}$. For tracking the residence time again water was assumed as an injector.

Experimental Tracer Test:

During the experimental period the flow rate ranged from $800\text{-}1500 \text{ m}^3/\text{hr}$, and the average flow rate during the experiment was $1000 \text{ m}^3/\text{hr}$. Before starting experiment, a solution of 10g of the fluorescent dye Rhodamine-wt was prepared. After that the fluorescent dye Rhodamine-wt was added at the influent end of the chlorine contact chamber as a pulse. The fluorometer was set up for the analysis of dye concentration. Fluorometer readings were taken every minute over the course of tracer test, with $t = 0$ being defined as the time of the pulse addition. Fluorometer provided the quantification of the concentration of dye in the contact chamber. The fluorometer data was downloaded in a real time to a connected laptop. The runtime of the test was recorded on a pocket stopwatch. Samples were collected in flasks, and analyzed for the Rhodamine-wt that was added to the system. Dye diffused and moved to the direction of flow. After 45 minute, samples were taken for every minute at designated points. The test was conducted for 2~3 hydraulic detention times. During the experiment, flow rate as a function of time was monitored and recorded at the times when fluorometer readings were taken. The tracer plume was observed as it passed through the contact chamber and its behavior noted (the observation of this

fluorescent dye provided an opportunity for flow visualization-characteristics of the mixing behavior within the system until dye dispersed to the point where it was no longer visible).

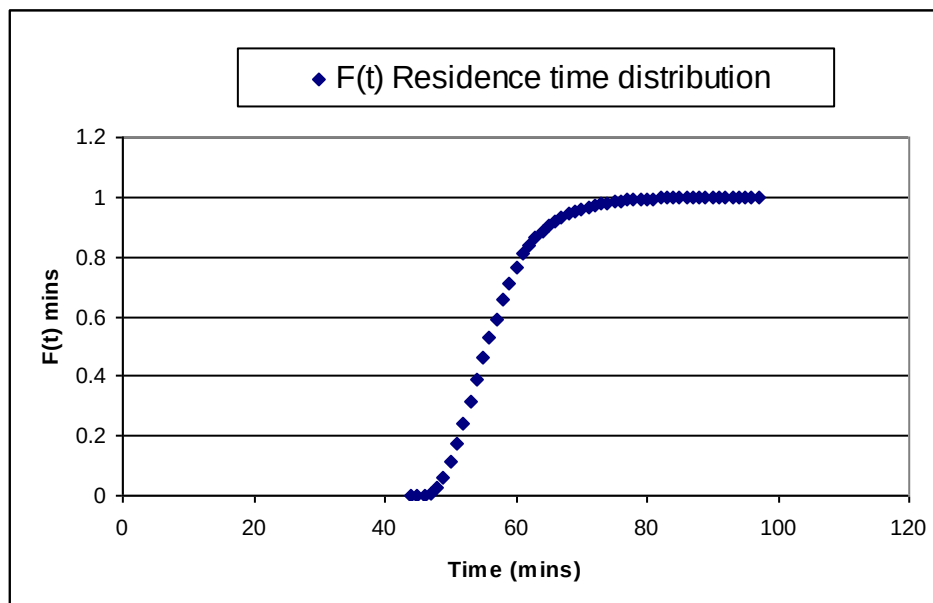


Fig. 3. The residence time distribution in contactor

Fig. 3 represents the residence time distribution ($F(t)$). Residence time distribution ($F(t)$) is the fraction of fluid elements in the effluent that reside in the chamber less than a given time. It is a useful description of the flow characteristics. $F(t)$ is evaluated as the fraction of tracer mass that exits the reactor. Steady-state hydraulics were assumed. It can be observed that, no tracer is detected in the effluent immediately. But, as the time increases, the effluent concentration gradually rises and then falls off after the tracer has begun to be exhausted from the system. From **Fig. 3**, it can be observed that the $\frac{1}{2}$ of the particles stay in the chamber for a time period of around 55 minutes. The mean residence time distribution of all the fluid elements of the tracer was around 55 mins. The above RTD plot was obtained from the Fluent. Fluent gave a mean residence time of 27 min (1620 sec) for the contactor. The bends in the particle track's can be accounted for the bends and corners in the contactor.

Result And Dissuasion

Simulation of existing CCT

When one observes the movement of water in a CCT, it is evident that there are several eddies (zones of recirculation). Some eddies are large and rotate slowly, other eddies are small and rotate fast (**Fig. 4**). Eddies are continuously combined and shed as the fluid moves through the CCT channel. In very simplistic terms the production and dissipation of turbulence appear to behave as follows:

- ◆ Turbulence is produced at a large scale such as when a fluid impinges on a baffle wall.
- ◆ The large-scale turbulence is then dissipated through the successive shedding of decreasing eddy size.
- ◆ The eddy shedding continues until eddies become so small that the viscous forces on the boundaries dissipate the energy into heat.
- ◆ Eddies of similar size coalesce to form larger eddies.

The spatial and temporal unsteady particle interaction inside the eddies as well as the particle interaction between the neighbouring eddies provides opportunity for CCT.

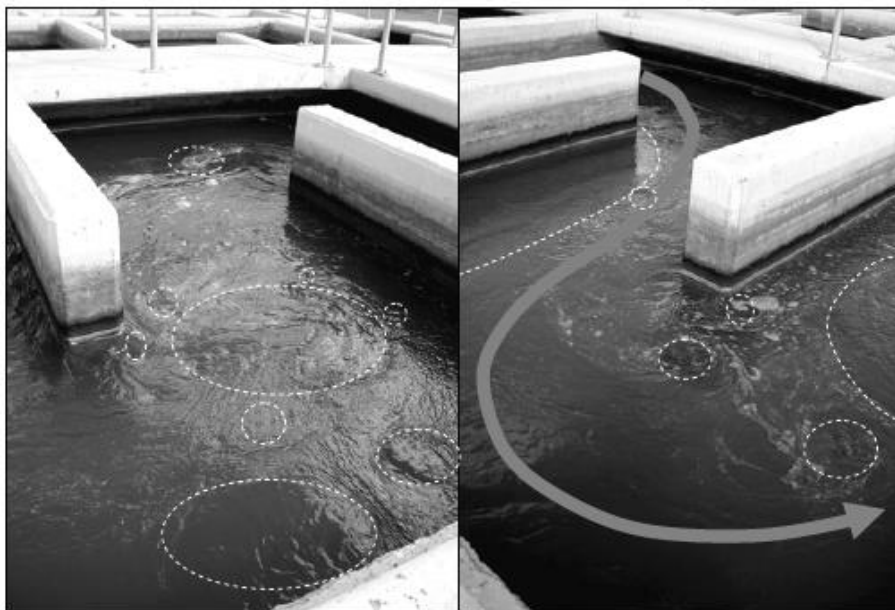


Fig. 4. Photographs present different eddy sizes.

To establish which eddy size and particle size are optimal for CCT remains a mystery, although several attempts have been made to determine this. The modelling efforts of **Lawler (1993)** have, for instance, demonstrated that the fluid shear does not play such an important role as originally thought. He argued that differential settling plays a more important role. However, his approach did not take the turbulent transport of fluid into account and was applied on laboratory scale experiments with ideal mixing conditions.

The CFD model provides an illuminating perspective on the turbulent transport of a fluid in a full-scale tank. Although the CFD approach of this study is limited to the fluid dynamic aspects, future research can perhaps link the fluid-particle and particle-particle interactions with the fluid dynamic behaviour.

Behaviour of CCT with Modifications

Not only does CFD provide new insight into the hydraulics and expected wastewater quality of a contact tank, it also enables the designer to optimise contact tank capital and treatment cost. As the designer has little control over the incoming wastewater quality the degrees of freedom are limited to the hydraulic efficiency of the tank and type of disinfectant that can be used.

In summer conditions, the maximum inflow from main sewer about 1200 m³/hr. Wastewater. At this flow rate, the wastewater in the tank would optimally turn over more than three times per day. However, disinfection efficiency was unacceptably low. Thermal stratification and high flow rates were causing short-circuiting of the flow directly from inlet across the tank to the outlet. Contact times (CT) were about 30 minutes. Several steps were undertaken to address the concerns. These were:

1. Installed a SolarBee mixer (**Fig. 5**) in the center of the tank inline between the inlet and outlet to increase CT and break up thermal stratification. The SolarBee SB10000v12PW was installed in June 2009. The intake hose for the SolarBee was placed on the floor near the center of the tank. The intent of the SolarBee is to increase contact time and efficiency by intercepting the plume of water transiting

across the bottom of the tank and pull it to the surface, thereby mixing the incoming flow, breaking thermal stratification in the water column and eliminating stagnation;
2. Conducted a computational fluid dynamics model of the flow conditions with and without the SolarBee.

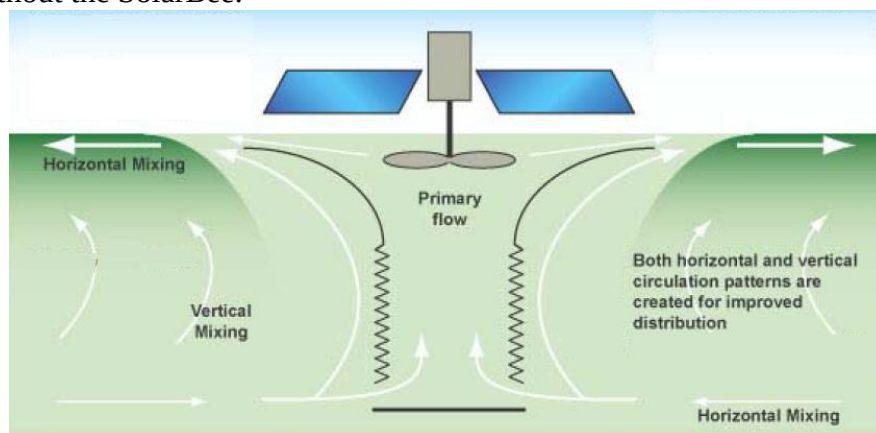


Fig. 5. SolarBee mixer

A computational fluid dynamics model of the tank was developed in this study. The model was developed to illustrate the hydrodynamic conditions as verified in the tracer study and to illustrate the short-circuiting phenomena without the SolarBee mixer in place. The results of the CFD modeling included the production of images based on the flow velocities as determined in the tracer study. **Figure 6** shows the CFD modeling of the tank without a SolarBee mixer. Short-circuiting is clearly evident as the plume of cool, incoming water transits directly across the bottom of the tank. The modeling shows that the new water entering the tank stays low, near the bottom, and short-circuits right to the outlet structure. Note that the upper levels of the tank, near the inlet structure, are at very low velocities and not mixing well with the new water coming into the tank. This condition is further aggravated with thermal stratification where these upper layers of the water column are warmer and less dense, and therefore do not easily mix with cooler denser water coming in.

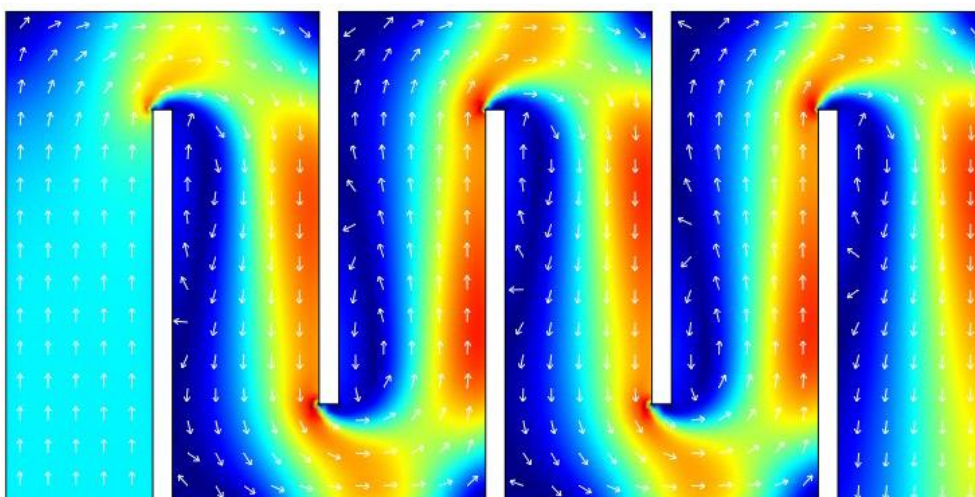
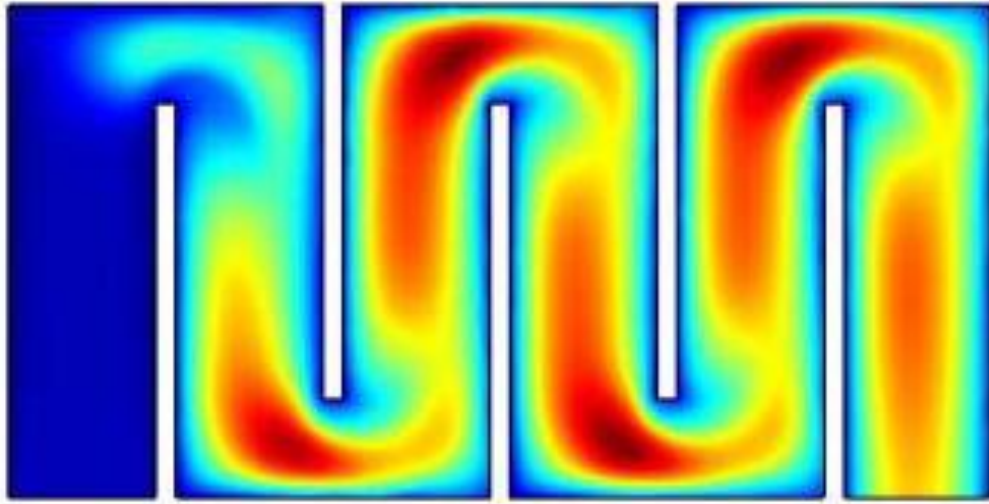


Fig. 6. Simulated flow field and velocity vectors in the contact tank without SolarBee

Fig. 7 shows the CFD modeling of the tank with the SolarBee mixer in place. The SolarBee is located in the middle of the tank, with the intake placed directly in

the short-circuiting path between the inlet/outlet structures. The CFD modeling shows that much of the new water entering the tank is captured by the SolarBee in the middle and then dispersed to the surface of the tank radially, including back towards the inlet structure. The yellow color just past the SolarBee, in line with the outlet structure, shows some 'blow by' past the SolarBee, indicating that the short-circuiting is not completely eliminated, but rather significantly controlled.



Fig, 7. Simulated flow field in the contact tank with SolarBee

The Effect Of The Solarbee Mixer Number And Place On Tank Efficiency

It is evident from the previous section, that the inclusion of SolarBee mixer can significantly improve the hydraulic performance of a contact tank. The question that arises is how many SolarBee mixer should be added? Several CFD simulations were performed for a number of different SolarBee mixer options, in an attempt to establish this. It is evident that as the number of SolarBee mixer increase, plug flow is approached. However, **Fig. 8** shows that the benefit derived from the addition of SolarBee mixer diminishes with increasing number of SolarBee mixer (based on the Morrill Index). Beyond about 5 to 7 SolarBee mixer accepted efficiency is gained.

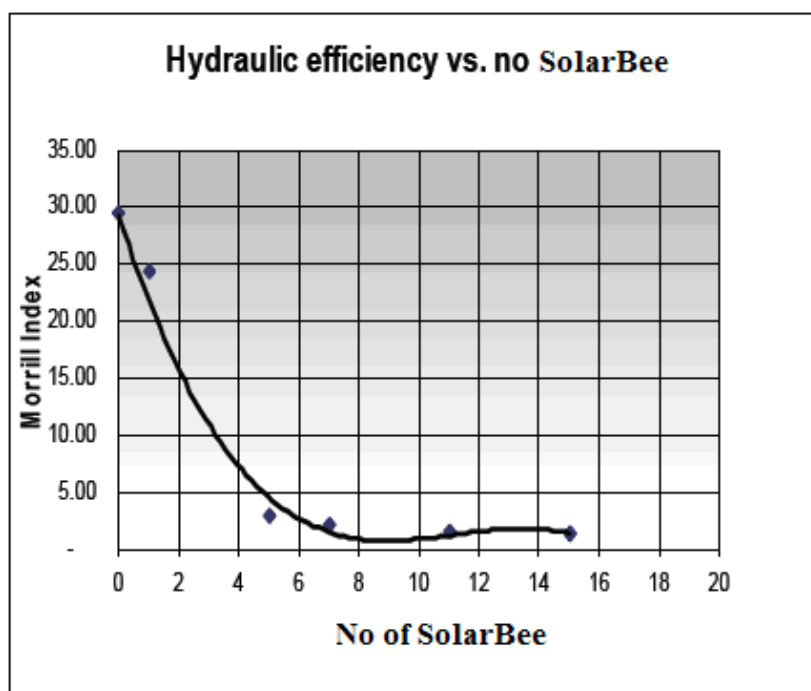


Fig. 8. Number SolarBee mixer vs. Morrill Index.

The previous section proves conclusively that an increasing number of SolarBee mixer improves the hydraulic efficiency. For a rectangular tank, SolarBee mixer should be placed parallel to the longest dimension of the tank. The hydraulic efficiency of a tank with SolarBee mixer positioned in the direction of the longest tank dimension is superior to SolarBee mixer placed along the shortest dimension. This improves the level of plug flow and reduces the number of turns that forms zones of recirculation.

When a step or pulse of tracer is injected at the inlet of the tank, a RTD curve can be simulated. This unsteady tracer dispersion and the derived RTD curves provide a better 'feel' for the expected wastewater quality. The resulting RTD curves for with and without SolarBee mixer tanks are compared in **Fig. 9**. It is evident that the T_{10} value for the tank without SolarBee mixer is much shorter than for the tank with SolarBee mixer. This is due to the fact that the tracer injected into the tank without SolarBee mixer is mixed into the recirculation zone and results in a short circuit. The tank with SolarBee mixer exhibits near plug flow as the water is forced to follow a predetermined path through the tank. The T_{10} value for the tank is 17 minutes while the T_{10} of the tank with SolarBee mixer is 134 minutes – almost 8 times longer. For this specific case the tank without SolarBee mixer needs to be almost 8 times larger in order to achieve a similar level of inactivation (based on contact time).

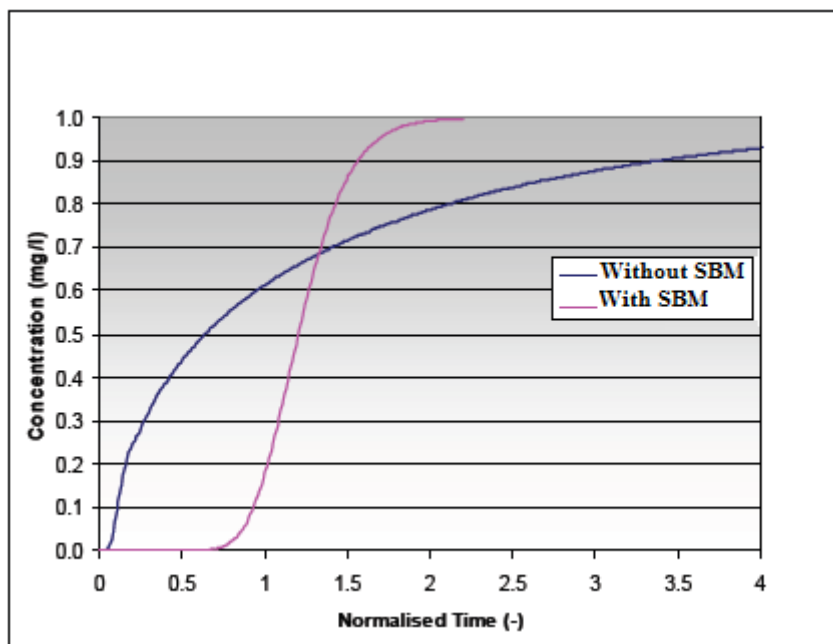


Fig. 9. Tracer dispersion in tank with and without SolarBee mixer (SBM)

As from the velocity vector plot the velocity decreased at corners and ends which is in consonance with the actual flow in the contactor. The results for RTD predicted from Fluent showed some variation from the actual results. This variation in results may be accounted for various reasons. The prediction on RTD depends upon geometry, boundary conditions, flow regime and description of flow fluid. Any error in these parameters can lead to variation in RTD. RNG k- ϵ model was used which might not represent the true flow regime inside the contactor. The assumptions for mixing condition of chlorine to treated influent might have caused some deviations. Any experimental error (human and apparatus) while conducting this experiment can result in different RTD. To conclude more research is required to adequately determine the flow conditions inside the contactor. A better understanding of the effect of boundary conditions, flow regime, model will help in predicting the residence time distribution.

Validation of the Model

Fig. 10 shows the comparison of the experimental and model results. The results of the experimental tracer studies show excellent agreement with CFD results. The reasons for the slight variations between the experimental and model results can be attributed to the over-prediction of turbulent diffusion by the k- ϵ turbulence model. This is a known shortcoming of this turbulence model, but the differences were found to be negligible in this particular case.

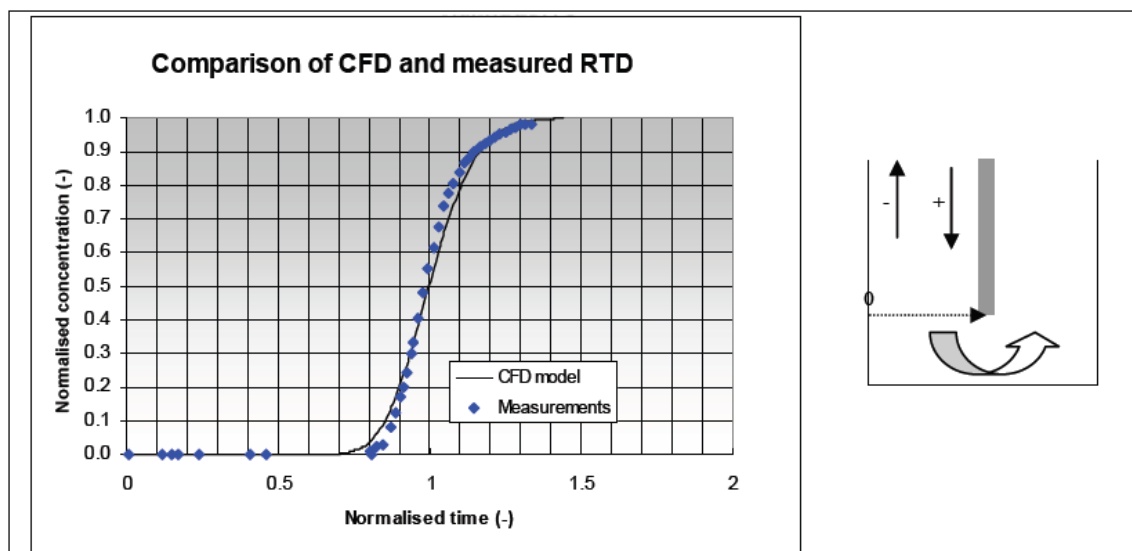


Fig. 10. Comparison of measured and modeled RTD

Conclusion

Understanding hydrodynamics and mass transfer characteristics within a contact tank is very useful in the design of this type of unit which are present in wastewater treatment plants. The approach based on Computational Fluid Dynamics (CFD) methods allows a detailed insight into momentum and mass transport in the tank which provides useful information for the designers to improve the hydraulic efficiency of the tank.

CFD modeling has effectively replicated mathematically the results of the tracer study, and demonstrated the effectiveness of the SolarBee mixer. The results of the experimental tracer studies show excellent agreement with CFD results. RTD curve realising that the hydraulics play an important role, RTD curves were used to infer the hydraulic behaviour. The RTD was subsequently used to serve as a substitute or a model for the hydraulic behaviour of the tank.

The CFD modeling has clearly illustrated the short-circuiting phenomenon that occurs without the SolarBee mixer in place. SolarBees can be effectively used to prevent short-circuiting by being placed directly in the short-circuiting path. The size and number of SolarBee required in other systems will be determined by geometry of the tank and flow rates.

References

- BROUCKAERT, C.J. PRYOR, M. BROUCKAERT, B.M. & BUCKLEY, C.A. (2000) A computational fluid dynamics study of an ozone contactor, *Proceedings 2000 WISA Conference*, Sun City, South Africa
- CHATAIGNER, O. GATEL, D. BONNE, P. & CAVARD, J. (1999) Better disinfection through optimised clearwells, *Proceedings 1999 AWWA annual conference*, Chicago, Illinois
- CHIU, K. LYN, D.A. SAVOYE, P. & BLATCHLEY, E.R (1999) Integrated UV disinfection model based on particle tracking, *Journal of environmental engineering*, 125(1), 7-16.
- CROZES, G.F. HAGSTROM, J.P. CLARK, M.M. DUCOSTE, J & BURNS, C. (1999) *Improving clearwell design for CT compliance*, (Executive summary) AWWARF & AWWA, Denver, Colorado, 154p
- Ducoste, Joel J and Peplinski Kevin Daniel. (2002) *Modeling of Disinfection Contactor of Hydraulics under Uncertainty*. J of Environmental Engineering.
- Fluent Help Manual 2006
- HANNOUN, I.A. BOULOS, P.F. & LIST, E.J. (1998) Using hydraulic modelling to optimize contact time, *Journal AWWA*, August, 77-87.
- HENRY, D.J. & BENNETT, M.W. (1996) Sizing of ozone contactors: a case study for the Joseph Jensen filtration plant, *Proceedings 1996 AWWA annual conference*, p427-451, Toronto, Canada.
- Hjertager, L. K., B. H. Hjertager, et al. (2002). CFD modelling of fast chemical reactions in turbulent liquid flows. *Computers and Chemical Engineering*, 26, 507-515.
- LAWLER, D.F., IZURIETA, E & KAO, C. (1983) Changes in particle size distribution in batch flocculation, *J AWWA*, Dec., 605-612.
- Levinspiel, O., 1999 "Chemical Reactor Engineering," Academic Press, New York.
- REDDY, S. (2000) Disinfection savings through simulation, *Water* 21, Oct. 46-47.
- Scott F. H., 1999, "Elements of Chemical Reaction Engineering," 3rd ed., Prentice Hall PTR, New Jersey.

Nomenclatures

	Symbol	Description
C		The solute concentration (mg/l)
C_D		The drag coefficient
C_i		the tracer concentration measured at the outlet
C_μ		A model constant
$C_\mu, \sigma_k, \sigma_\epsilon, C_{1\epsilon}$ and $C_{2\epsilon}$		constants
D		deformation tensor
D_t		turbulent diffusivity
k, k'		The turbulent kinetic energy (m^2/s^2)
Q		Flowrate (m^3/s)
t		Time (s)
V		Volume (m^3)
v, u		Velocity (m/s)
ϵ, ϵ'		The dissipation of turbulent energy (m^2/s^3)
ν_t		The turbulent viscosity
ρ		The fluid density (kg/m^3)
CFD		Computational Fluid Dynamics
RTD		Residence Time Distribution