

تحسين أداء أحواض التهوية في محطات معالجة مياه الصرف الصحي باستخدام نموذج ديناميكا السوائل الحسابية

د.علي هادي غاوي

جامعة القادسية \ كلية الهندسة \ قسم الهندسة المدنية

الخلاصة

ان التصميم الهيدروليكي لمحطات معالجة مياه الصرف الصحي تعتمد على المحددات التصميمية و الخبرة العملية للمصمم. البحوث والتطوير في مجال تصميم محطات معالجة مياه الصرف الصحي يعتمد بشكل رئيسي على فهم وتحسين العمليات الحيوية في وحدات المعالجة. على أية حال التصميم الهيدروليكي عامل مهم في تقييم أداء محطات معالجة مياه الصرف الصحي. و يعتبر حوض التهوية في محطات معالجة مياه الصرف الصحي والذي يشمل السلوك الهيدروليكي و الحيوي احد أهم مراحل المعالجة التي يمكن ان تقيم بواسطة أداء محطات معالجة مياه الصرف الصحي. في هذا البحث تم تطوير موديل رياضي عام باستخدام نموذج ديناميكا السوائل الحسابية CFD لمعرفة خواص الجريان و تحسين أداء محطات معالجة مياه الصرف عن طريق تحسين أداء حوض التهوية في محطة معالجة مياه الصرف الصحي في مدينة الديوانية. كذلك تم تطوير نموذج RTD للتنبؤ بزمان بقاء مياه الصرف الصحي في حوض التهوية بالاعتماد على معرفة سلوك الجريان. وتم إضافة نموذج المتطلب الحيوي للأوكسجين BOD الى النموذج الرياضي الذي طور في هذه الدراسة لغرض تمثيل إزالة المتطلب الحيوي للأوكسجين في حوض التهوية. تم تطبيق عدد من السيناريوهات لنموذج ديناميكا السوائل الحسابية في هذه الدراسة. ان النتائج التي تم الحصول عليها من هذه الدراسة تعتبر مفيدة جدا لمعرفة تأثير حصول الترسيب و الدوامات القصيرة على أداء حوض التهوية. تم مقارنة النتائج الناتجة من النموذج الرياضي و الناتج من القياسات حيث كانت النتيجة تطابق جيد بين النتائج و القياسات. ان هذه الدراسة أثبتت التصميم الهيدروليكي يحسن أداء تشغيل محطات المعالجة و يقلل كلف الصيانة و الطاقة.

NOMENCLATURE

| | |
|---------------------|-------------------------------------|
| BOD | biological oxygen demand |
| C_e | BOD removal, 100% |
| CFD | computational fluid dynamics |
| RTD | Residence Time Distribution |
| d | diameter, m |
| F_{torque} | torque, N.m |
| k | turbulent kinetic energy, m^2/s^2 |
| L | length, m |
| MRF | multiple reference frame |
| N | rotational speed, rpm or rps |
| NP | power number, dimensionless |
| NQ | flow number, dimensionless |
| P | power, W |
| PIV | particle image velocimetry |
| Q | flow rate, m^3/s |
| r | radius, m |
| Re | Reynolds number, dimensionless |
| S | source term |
| t | time, s or day |
| u | absolute velocity, m/s |
| UDS | user defined scalar |
| v | relative velocity, m/s |
| \bar{v} | average velocity, m/s |
| $V \square \square$ | flow rate, m^3/s |
| x, y, z | Cartesian coordinate |

Greek symbols

| | |
|---------------|---|
| ρ | density, kg/m^3 |
| ϕ | general parameter for CFD model equations |
| Γ_ϕ | transport coefficient |
| ω | angular velocity, rad/s |
| ε | dissipation rate of turbulent kinetic energy, m^2/s^3 |

Subscripts

| | |
|--------|-------------|
| i, k | tensor form |
|--------|-------------|

1. Introduction

Biological treatment of wastewater by Aeration Tanks (AT) has been widely used throughout the world for more than 35 years, relying on unsophisticated technology and minimal maintenance. Wastewater treatment in AT involves a complex combination of physical, chemical, and biological processes that are affected by many parameters such as AT types and configurations, system design, and environmental conditions. In terms of the predominant biological environment that exists, AT can be classified as facultative, aerated, aerobic, and anaerobic AT. Usually, AT serve as an economical means for secondary treatment of wastewater, in which the propellers through mechanical mixing bring the wastewater into contact with microorganisms that decompose the pollutants. Incomplete mixing may result in dead zones in a AT where the actual liquid retention time is more than the designed retention time while in other areas the residence time is less than desired. AT hydraulic performance can be experimentally and theoretically investigated. One conventional method is to inject a dye tracer into the AT inlet and then measure the tracer concentration at the outlet to obtain the Residence Time Distribution (RTD) model. For large AT, however, tracer studies are time-consuming, labor intensive, and expensive (Pougatch et al., 2007). Baleo and Le Cloirec (2000) presented a modeling approach to simulate the spatial distribution of mean residence time in a turbulent axisymmetric flow containing expansions and contractions, and compared the simulations with experimental measurements obtained with a pulse injection method using butane as the trace gas. De Kretser, Matthews and Williams (2002, 2003) examined the flow patterns generated by individual aerators and mixers, and the combinations of three aerators and three mixers in a rectangular block of 70 m² with a depth of 4.3 m instead of the whole AT. The results showed that aerators produce very good surface mixing but are relatively inefficient at depth, whereas the mixers promote good mixing at depth but are less effective on the surface. Pougatch et al. (2007) developed a 3-D computational flow model for large AT including high-speed floating mechanical surface aerators. In this model, the lagoon was divided into two domains with 315.6 m*315 m*2.85 m in each zone. Each aerator was simulated separately and then was applied to the boundary condition in the AT model. Stropky et al. (2007) developed a calculation method based on random-walk Lagrangian particle tracking to predict RTD in large mechanically aerated tank. Several of the studies (Ghawi and Kris 2007 a, and 2007 b), have been carried by use of CFD model. To the author's knowledge, the latest CFD model was reported by Pougatch et al. (2007), in which about 1.6 million computational cells were used to generate grids and 10 days of CPU time on a PC with Pentium 3 at 1.2 GHz was taken to obtain one solution.

Wastewater treatment plant at Al-Dewanyia is considered the primary sources of pollution for the Al-Dewanyia River. This study attempts to highlight the factors leading to the inadequate performance of Al-Dewanyia Waste Water Treatment Plant (DWWTP) in removing organic matter, and solids.

The overall goal of this study was to develop a general CFD model requiring less CPU time for conventional aeration tank simulation. The specific objectives of this study were to:

1. Develop a theoretical model integrating water flow, local RTD, and BOD removal in AT;
2. Validate the power number and flow number of a propeller used in AT against experiments; and
3. Predict flow fields, residence time, and BOD removal in a AT for the existing design.

2. Model Development

The theoretical model was developed based on the following assumptions: (1) Water flow in the AT is 3-D and steady state, (2) Water is an incompressible and isothermal Newtonian fluid. (3) The tracers have the same density as the surrounding fluid and no diffusion occurs, (4) The model is limited to the flow model without considering the heat flow, as the water temperature is constant at 20°C, (5) BOD removal follows first-order kinetics, in which the decay reaction rate is assumed to be 0.23 d⁻¹ at 20°C, (6) The model is single phase without accounting for multiphase interactions, (7) Pumping action of each propeller is represented by one momentum source in the AT simulation, and (8) The effect of wind speed on water flow is negligible.

2.1 Fluid flow

The CFD model is set up in ANSYS-Fluent Inc., (2007) using the standard available k-epsilon turbulence model. The medium is activated sludge, which is a mixture of water, organic matter and small particles. The physical properties of activated sludge are very similar to pure water, therefore the density and viscosity of water are used. In this study, the rotation of the propeller was solved in Multiple Reference Frame (MRF). The general governing equation can be expressed as (Patankar, 1980):

$$\frac{\partial (\rho \phi)}{\partial t} + \text{div} (\rho \vec{u} \phi) = \text{div} (\Gamma_{\phi} \nabla \phi) + S \quad (1)$$

where t is time, ρ is density of wastewater, Γ_{ϕ} is transport coefficient, ϕ represents velocity components, turbulent kinetic energy or its dissipation rate, S is source term dependent on ϕ , and \vec{u} is the absolute velocity which can be calculated by:

$$\vec{u} = (\vec{\omega} \times \vec{r}) + \vec{v} \quad (2)$$

where $\vec{\omega}$ is the angular velocity vector, \vec{r} is the position vector in rotating frame, and \vec{v} is relative velocity vector in rotating frame.

2.2 Turbulence model

Since the Reynolds stress term in the Reynolds-averaged Navier-Stokes equation for turbulent flow is unknown, many turbulence models have been developed to complete the flow equations. The commonly used turbulence models include standard $k-\varepsilon$, RNG $k-\varepsilon$, Realizable $k-\varepsilon$, and Reynolds stress models, among which the realizable $k-\varepsilon$ model was tested to work well for recirculation and rotation flows (Shah, 1979), and thus was used in this study.

2.3 Power number

The power number in a stirred vessel is defined as (Paul, Atiemo-Obeng and Kresta, 2004):

$$N_p = \frac{P}{\rho N^3 D^5} \quad (3)$$

where N is the rotational speed, D is the diameter and P is the power delivered to the fluid through the blades of propeller which can be calculated as:

$$P = 2\pi N F_{torque} \quad (4)$$

2.4 Flow number

The flow number in a stirred vessel is defined in (Paul, Atiemo-Obeng and Kresta, 2004):

$$N_Q = \frac{Q}{ND^3} \quad (5)$$

where Q is flow rate generated by the propeller.

2.5 Local RTD model

RTD model is represented by a scalar transport equation. For an arbitrary scalar ϕ_k , the general transport equation can be expressed in (ANSYS-Fluent Inc., 2007):

$$\frac{\partial}{\partial t}(\rho \phi_k) + \frac{\partial}{\partial x_i}(\rho u_i \phi_k - \Gamma_k \frac{\partial \phi_k}{\partial x_i}) = S_{\phi_k} \quad (6)$$

According to the assumption that there is no diffusion between the tracer and the background liquid, if the source term is specified as the density of the background liquid (S_{ϕ_k}), then the resident time of the tracer can be obtained by solving the convection term only under steady state as:

$$\frac{\partial}{\partial x_i}(u_i \phi_k) = 1 \quad (7)$$

where ϕ_k represents the physical resident time of the tracer.

2.6 BOD removal model

The most widely used parameter to represent the organic pollution in wastewater treatment is the BOD. Based on the first-order kinetics, BOD removal in aeration tanks be expressed as (Metcalf and Eddy, 2003):

$$C_e = 1 - 1 / e^{kt} \quad (8)$$

where k is constant rate and t is hydraulic retention time.

3. CFD Strategy

CFD modeling strategies include grid generation and then numerically solving the governing equations. In meshing a DWWTP aeration tank with multiple propellers, controlling mesh quantity and ensuring mesh quality are crucial to a successful simulation. AT meshing was implemented by Gambit software (ANSYS-Fluent Inc., 2007). The main meshing procedures included: (1) constructing AT geometry with inlets, outlet, and propellers, (2) decomposing the whole domain into a few sub-domains, (3) generating tetrahedral mesh for MRF zones containing propellers and hexahedral mesh for all other regions, and (4) using size functions to control mesh growth. Numerical computing was performed by *Fluent* software (ANSYS-Fluent Inc., 2007). The key setups under Fluent were: (1) using Green-Gauss node based solver, (2) specifying the influent as velocity-inlet, the effluent as pressure-outlet, the water top surface as symmetry, and walls as no-slip velocity, and (3) starting iterations from the first order upwind and then switching to the second order upwind scheme. Upon obtaining converged flow fields, user defined scalar (UDS) can be active to solve Eq. (7) to predict residence time, and then the custom field function can be used to calculate BOD removal by Eq. (8).

4. Results And Discussions

20HP Raptor propeller used in this study was tested in DWWTP to obtain its power number and flow number. Power consumption at 1200 rpm of rotational speed (N) was measured. Wattmeter power was multiplied by motor efficiency to determine motor output horsepower, also designated as brake HP. An Esterline Angus Recording Wattmeter was used to measure electrical power to the motor. Flow rate was obtained by measuring the velocity fields through particle image velocimetry (PIV). PIV apparatus consists of a digital camera, a high power laser, and an optical arrangement to

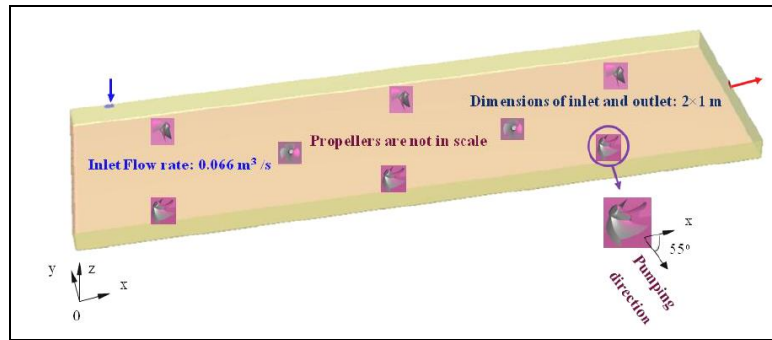
convert the laser output light to a light sheet. A fiber optic cable connects the laser to the cylindrical lens setup. The laser acts as a photographic flash for the digital camera, and the particles in the fluid scatter the light that is detected by the camera. Upon obtaining the power input and flow rate of the propeller, the power number and flow number can be predicted by Eqs. (3) and (5). Comparison of the measured and simulated values of power number and flow number indicates that CFD is capable of predicting propeller-driving flow fields (**Table 1**).

Table 1 Measured and simulated power number and flow number of a propeller.

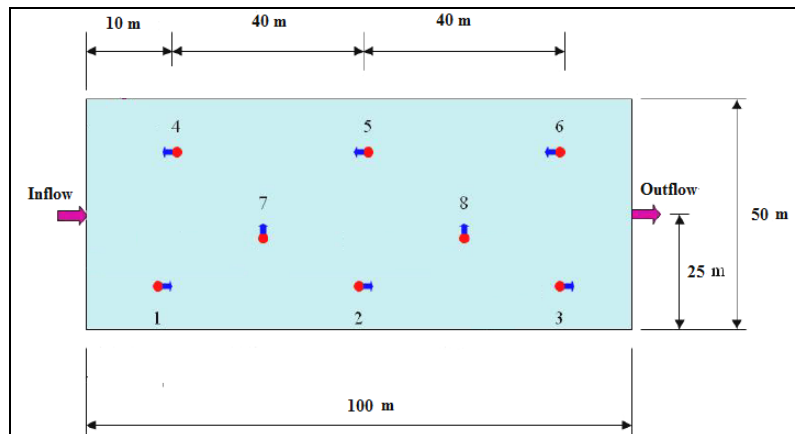
| | Measured | Simulated |
|--------------|----------|-----------|
| Power number | 0.41 | 0.4 |
| Flow number | 0.6 | 0.6 |

In order to investigate the feasibility of substituting momentum source for each propeller, a AT (100 m*50 m* 5.0 m) with 8 propellers was simulated (**Fig. 1**) (**Existing Design**). Arrangements of 8 propellers were made in three rows. Propellers 1, 2 and 3 in the first row were pumping towards the outlet, opposite to the pumping direction of propellers 4, 5, and 6 in the third row. To enhance mixing in the central region, propellers 7 and 8 were placed in the second row pumping towards the inlet. All propellers were operated at $N=1200$ rpm. Taking propeller 3 as an example, average velocity magnitude (v) that is normal to the exit plane for each MRF zone can be predicted as 4.6 m/s for $N=1200$ rpm illustrated in **Fig. 2(a)**, and then the momentum source can be specified by fixing two velocity components illustrated in **Fig. 2(b)**. Following this procedure, the velocity components for all other momentum sources can be specified. All the momentum sources have the same velocity magnitudes but the directions are different for different locations. **Fig. 3** shows qualitative and quantitative comparisons of velocity contours at some specified planes for propellers and momentum sources. In all displays of velocity contours the velocity magnitudes in the red area are greater than (or equal to) the maximum value specified in the velocity contour bar.

From **Fig. 3** it can be seen that the overall flow patterns are similar although velocity magnitudes in some areas are different. These differences could be attributed to the fact that only two velocity components are fixed in each momentum source while propeller-driving has three velocity components. Generally, a momentum source acting as a propeller is acceptable. In this study, AT meshing and CFD simulation were performed with 6 dual processors (2 GB memory of each processor), and every solution was checked for grid independence. CFD simulation using momentum sources, evaluated in terms of CPU time and number of mesh cells, shows higher efficiency than that using propellers (**Table 2**).

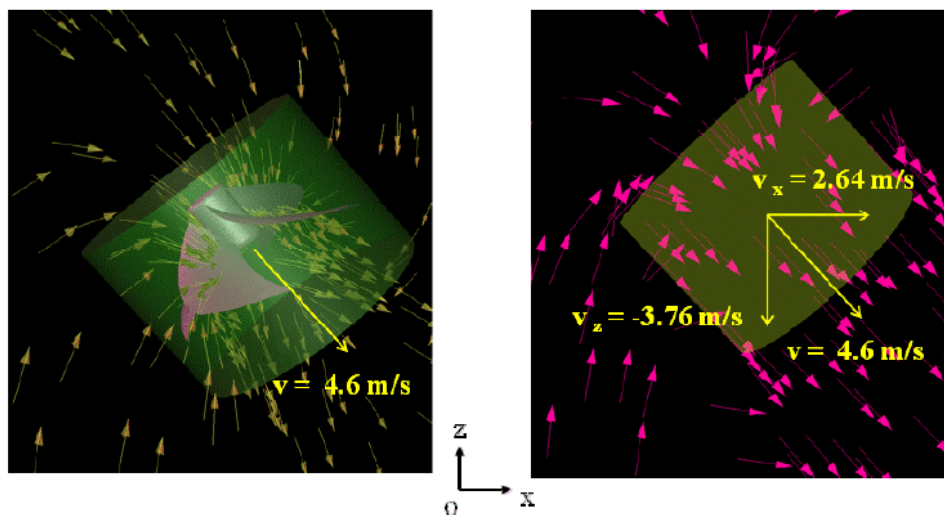


(a) Aeration tank configuration



(b) Arrangement of propellers

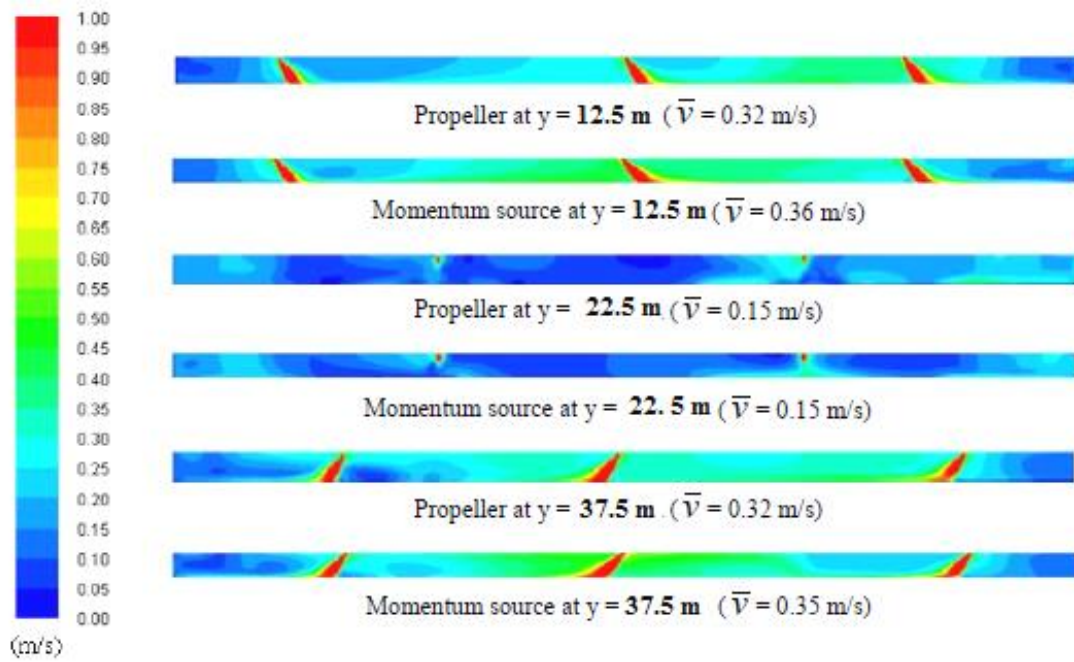
Fig. 1 Aeration tank with 8 propellers (Existing Design).



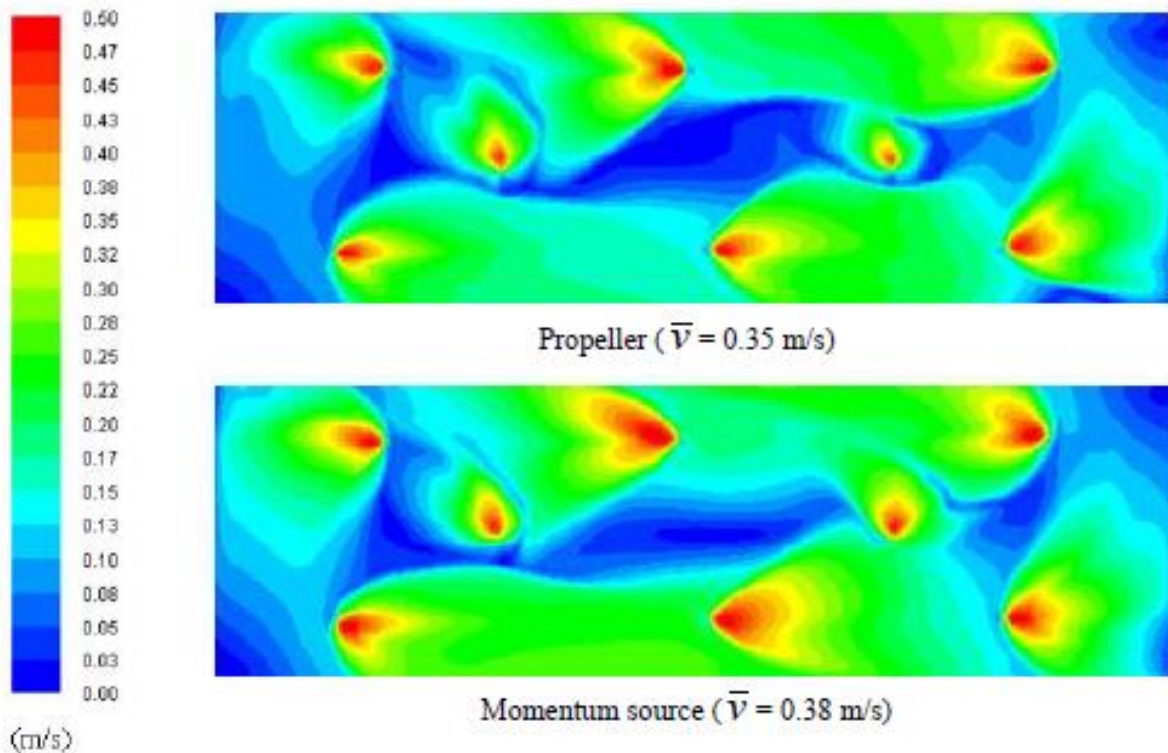
(a) MRF zone containing propeller

(b) Momentum source zone

Fig. 2 Velocity vectors in MRF and momentum source zones



(a) z-x planes respectively at $y = 12.5, 22.5$, and 37.5 m



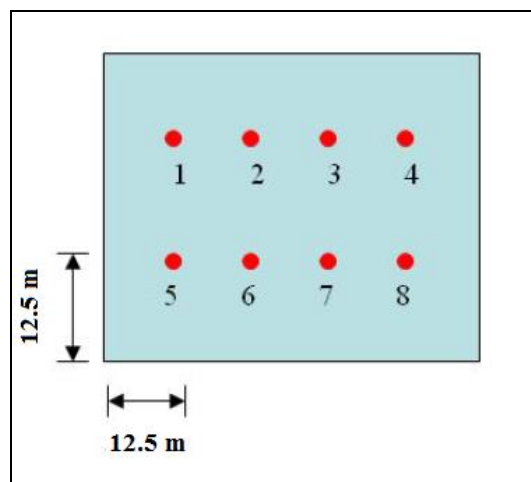
(b) x-y plane at $z = 0.1 \text{ m}$

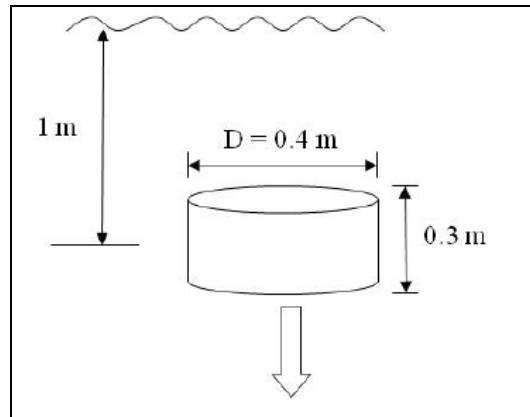
Fig. 3 Velocity contours

Table 2 Mesh cells and CPU time for a AT.

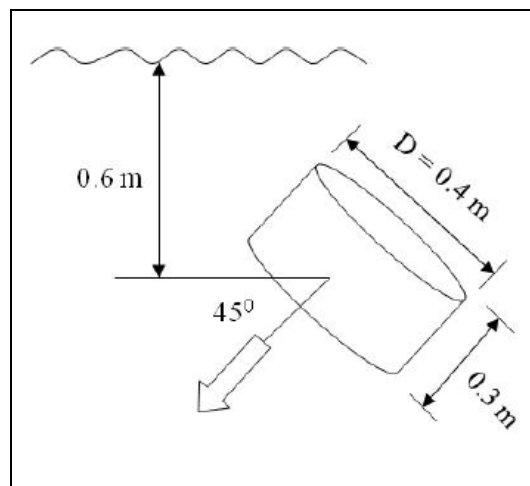
| | Propeller | Momentum source |
|---|-----------|-----------------|
| Mesh cells for each propeller / momentum source | 54,000 | 74 |
| Total mesh cells | 1,224,000 | 602,000 |
| CPU time | 8 hours | 4 hours |

Figure 4 shows the AT proposal design. The proposed design using 8 propellers aimed 45° down. The propellers used in the existing and proposed designs, represented by the momentum sources, are depicted in **Fig. 5**. Total mesh cells and CPU time for both designs are shown in **Table 2**. All propellers in the AT for the existing design were placed in two rows with uniform spacing. It can be observed from **Fig. 6** that the mixing surrounding each propeller is very intensive but weak along the side walls. The overall flow patterns from top view (x-y planes) at different depths are similar. However, the average velocity is 0.06 m/s at $z=0.1$ m which is greater than 0.04 m/s at $z=1.4$ m, indicating good mixing at depth.

**Fig. 4 Proposed Design**



a) Existing Design



(b) Proposed Design

Fig. 5 Propellers represented by momentum sources

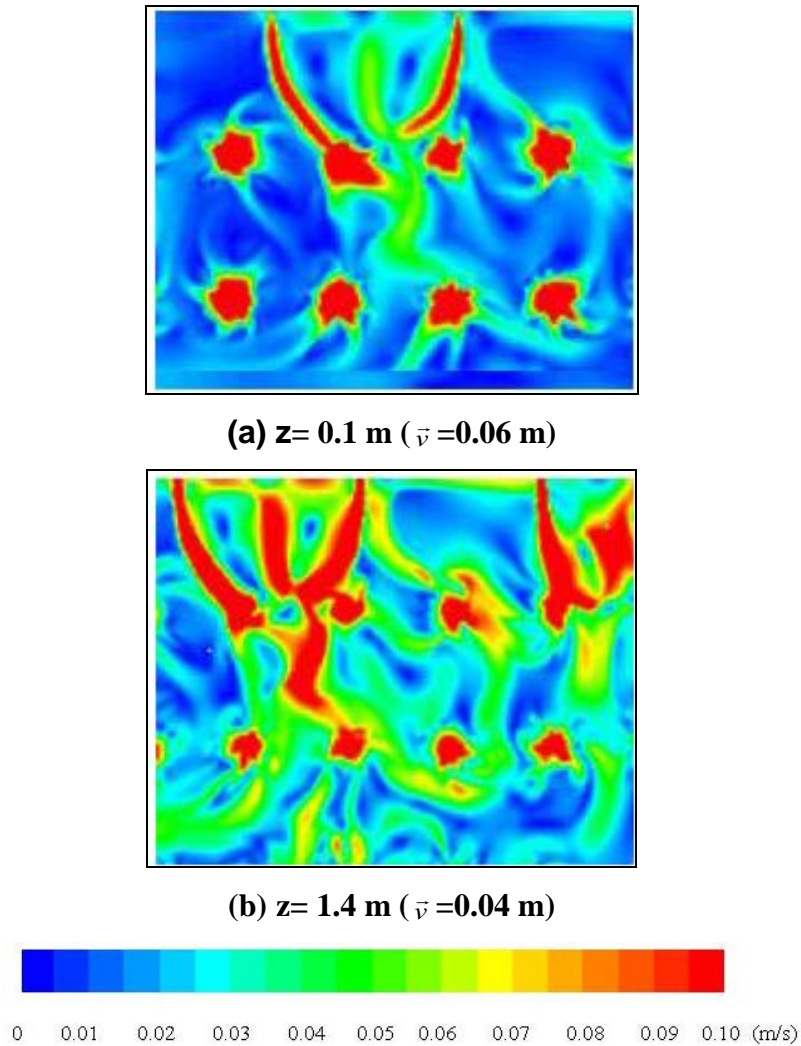
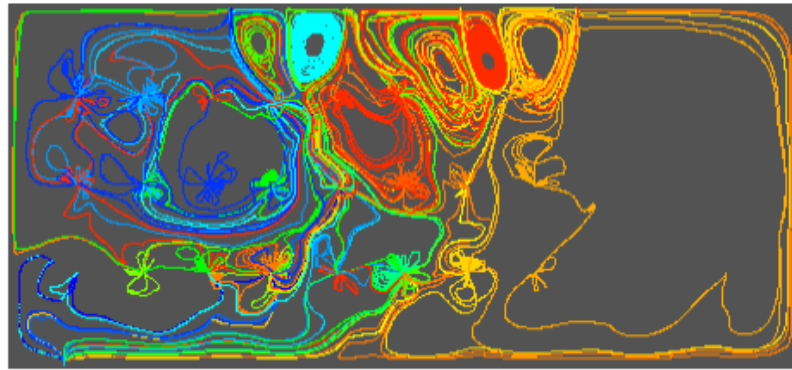


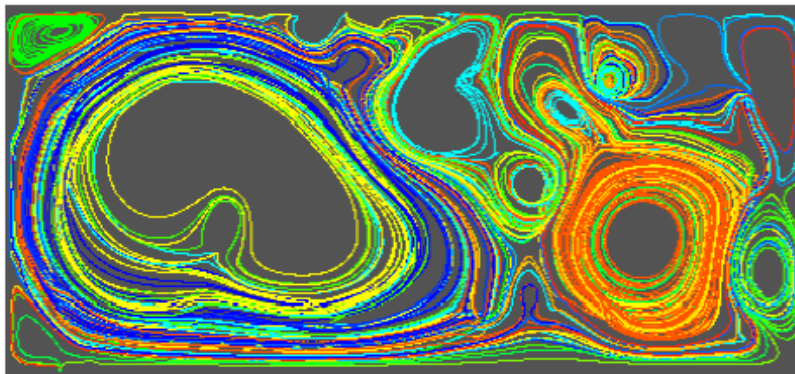
Fig. 6 Velocity contours at x-y planes

To trace the flow paths, some colored particles were assumed to be injected at the inlets. For steady state flow, each pathline refers to the trace of a single particle in space. **Fig. 7** shows the flow paths from the top view in both designs. In the existing design the density of pathlines in the vicinity of each inlet is higher than that in other regions, and few particles flow to the right side of the AT. In the proposed design, however, pathlines are relatively uniform over the whole domain and two significant flow circulations and several small ones can be observed.

Simulations of local residence times in the AT were conducted based on the converged flow fields. **Fig. 8(a)** shows the contours of residence time at $z=0.1 \text{ m}$ in the existing design, in which the residence time in the right side is greater than that in the left side. This phenomenon is because of the hydraulic short-circuiting. The objective of AT hydraulic design is to obtain the longest uniform retention time of water. To increase the uniformity of the residence time, one possible way is to maximize the water flow path. In the proposed design, the residence time in the left side of AT depicted in **Fig. 8(b)** has been improved with the increase of flow paths in this region.

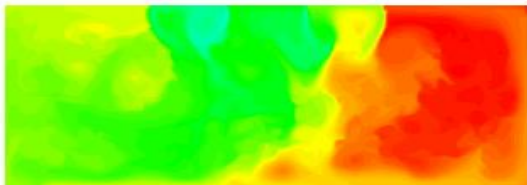


a) Existing Design

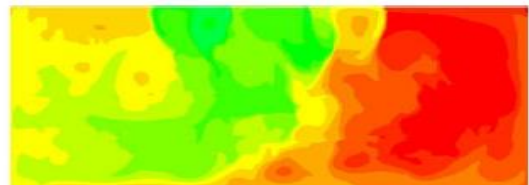


b) Proposal Design

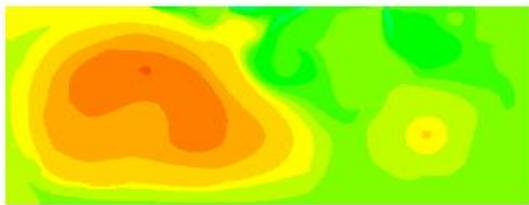
Fig. 7 Path-lines colored by particles releasing from influent



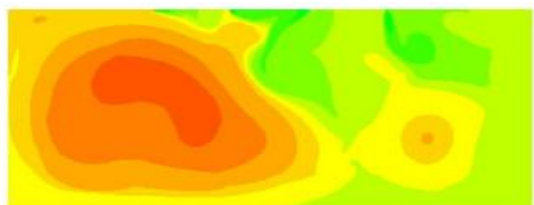
(a) Existing design



(a) Existing design



(b) Proposed design



(b) Proposed design

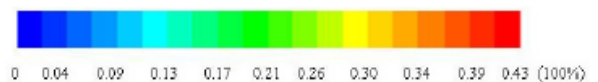
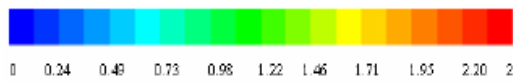


Fig. 8 Contours of residence time in x-y plane at $z = 0.1$ m

Fig. 9 Contours of BOD removal in x-y plane at $z = 0.1$ m.

Upon obtaining the residence time, the goal of predicting BOD removal (C_e) in AT can be achieved. C_e was calculated using the custom field function in Fluent software without iterations. From Eq. (8) it can be seen that C_e is the function of constant rate (k) and residence time (t). If k is kept constant, then C_e increases nonlinearly with the increase of t . As a result, BOD removal shown in **Fig. 9** corresponds to the residence time shown in **Fig. 8**. **Table 3** gives quantitative comparisons of BOD removal inside the AT and its BOD reduction in the effluent for both designs. Even though BOD removal for the existing design is only a little better than that for the proposed design, the total power consumption for the existing design is much higher than that for the proposed design (**Table 4**).

In the present research stage, the AT temperature was assumed to be constant in calculating BOD removal. Actually, the AT temperature varies with weather conditions, and solar radiation, etc. Also the prevailing wind speed should affect the accuracy of the local RTD. To complete this study, there is a need for further research that would integrate the AT hydraulic model with heat transfer and wind flow models. Controlling the mesh quantity to reduce CFD computing time is very important for AT simulations. CFD along with local RTD methodology developed in this study provides an effective simulation tool to optimize AT through relocating propellers. Prediction of residence time would assist environmental engineers in analyzing the hydraulic performance by evaluating a AT mixing characteristics instead of using the conventional RTD method that treats a AT as a black box. In essence this program opens the black box for a more efficient design.

Table 3 BOD removal for two design

| | Existing design | Proposed design |
|-------------------------------------|-----------------|-----------------|
| Average BOD removal in the whole AT | 31.3% | 25.4% |
| BOD reduction in the effluent | 29% | 35% |

Table 4 Power consumption for two designs.

| | Existing design | Proposed design |
|-----------------------------------|-----------------------|----------------------|
| Total power consumption | 295 kW | 140 kW |
| Power consumption per unit volume | 1.23 W/m ³ | 0.6 W/m ³ |

5. Conclusion

From the CFD analysis of aerated tank, the following conclusions can be drawn:

1. The pumping action of the propeller can be represented by a momentum source effectively.
2. A hydrodynamic model that integrates a simple biological model has been successfully developed to evaluate aeration tank mixing performance.
3. For a aerated tank, mixing performance in the proposed design with less propellers and relocations has been improved in terms of BOD removal per unit power consumption.

6. Acknowledgements

The authors wish to thank the Al-Dewanyia Wastewater Treatment Plant staff for accessing to the required data.

7. References

ANSYS-Fluent Inc. (2007). *Fluent 6.3*. Lebanon, N.H.

Baleo JN, Le Cloirec P (2000). Validating a prediction method of mean residence time spatial distributions. *AIChE J* 46(4):675–683.

Baleo JN, Humeau P, Le Cloirec P (2001). Numerical and experimental hydrodynamic studies of a lagoon pilot. *Water Research* 35(9):2268–2276.

De Kretser D, Matthews B, Williams M (2002). Maryvale mill uses CFD to investigate natural and mechanically induced flows in an aerated lagoon. *Paper Technology* 43(9):35–38.

De Kretser D, Matthews B, Williams M (2003). Use of computational fluid dynamics to investigate natural and mechanically induced flows in an aerated lagoon. *Tappi Journal* 2(4):12–14.

Ghawi A. Hadi Jozef Kris (2007a). Design and Optimization of Sedimentation Tank in Slovakia with CFD Modeling. 10th International Symposium on Water Management and Hydraulic

Engineering 2007 with special emphasis on the impact of hydraulic engineering construction on the environment 4 – 9 September 2007. Šibenik , Croatia.

Ghawi A. Hadi, Jozef Kris (2007b). A Numerical Model of Flow in Sedimentation Tanks in Slovakia Third International PhD Symposium in Engineering 25-26 October 2007, Hungary University of Pollack Mihály Faculty of Engineering Pécs, Hungary.

Metcalf and Eddy (2003). Wastewater Engineering (fourth edition). McGraw-Hill, New York.

Patankar SV (1980). Numerical Heat Transfer and Fluid Flow. Hemisphere/ McGraw-Hill, New York.

Paul EL, Atiemo-Obeng VA, Kresta SM (2004). *Handbook of Industrial Mixing*. John Wiley & Sons, Inc, New Jersey.

Pougatch K, Salcudean M, Gartshore I, Pagoria P (2007). Computational modelling of large aerated lagoon hydraulics. *Water Research* 41:2109–2116.

Shah NG (1979). A new method of computation of radiant heat transfer in combustion chambers. PhD Dissertation. Imperial College of Science and Technology, London, England.

Stropky D, Pougatch K, Nowak P, Salcudean M, Pagoria P, Gartshore I, Yuan JW (2007). RTD (residence time distribution) predictions in large mechanically aerated lagoons. *Water Science and Technology* 55(11):29–36.