CFD modelling and optimisation of a Waste Water Treatment Plant Bioreactor- a case study

Andrew Elshaw
School of Engineering and Technology
Central Queensland University
Cairns, Queensland, Australia
andrew.elshaw@cqumail.com

N.M.S. Hassan*
Centre for Intelligence System
School of Engineering and Technology
Central Queensland University
Cairns, Queensland, Australia
n.hassan@cqu.edu.au
*Corresponding Author

M.M.K. Khan
School of Engineering and Technology
Central Queensland University
Rockhampton, Queensland, Australia
m.khan@cqu.edu.au

Abstract— This This study aims to determine the optimal configuration for wall mounted mixers based on the comparison of computational fluid dynamics (CFD) modelling results to physical data collected from the treatment plant. The study was developing a CFD model of an anoxic zone-1 and simulating the fluid flow using ANSYS code 'Fluent'. A 2D model of an activated sludge process bioreactor anoxic zone was simulated to evaluate the hydrodynamic performance and influence of the inflow through the various inlets. Furthermore, the simulation also sought to evaluate the influence on the hydrodynamic performance from structure geometry. The 2D model was able to simulate the flow pattern within the zone and results from the CFD model varied between 3% and 10% at key locations.

A 3D model was also developed of the anoxic zone to further evaluate the hydrodynamic performance. The 3D model produced consistent results to the physical data collected from the plant. The hydrodynamic performance of the anoxic zone was able to be evaluated from the CFD simulations and from the physical samples collected for velocity readings and suspended solids. In the key locations, the CFD simulation showed the consistent results with the physical data. The anoxic zone was subject to velocity lower than the desired 0.3 meters per second. However suspended solid samples suggest that the zone is still within the acceptable range for specific power dissipation. Therefore, an increase in operating parameter which increases the inflow into the zone can mitigate the need for submersible mixers.

Keywords— *computational fluid dynamics; hydrodynamic performance; specific power dissipation; anoxic zone*

I. INTRODUCTION (*HEADING 1*)

A Waste Water Treatment Plants (WWTP) is an industrial facility where a combination of mechanical, physical, chemical and biological processes is used to achieve pollutant removal from the incoming wastewater [1]. WWTP are used to treat and process raw sewage prior to discharging into water ways. These facilities are commonly operated and resourced by local governments and are critical infrastructure assets that need to be fully operational, all year round in Australia. Adjustments to the operational parameters can influence the treatment process downstream and with heavy penalties for breaching environmental license conditions, achieving efficient operation requires a detailed analysis of the treatment system.

Between 1900's to early 1970's waste water treatment objectives focused on the removal of colloidal, suspended and floatable materials, treatment of biodegradable organics and the elimination of pathogenic organisms. It was not until mid-1970 and 1980 that the treatment methods evolved and the focus shifted to the reduction of biological oxygen demand (BOD), total suspended solids (TSS) and pathogenic organisms. Furthermore, the influence of nutrients such as, nitrogen and phosphorous and their impacts which they had on aquatic environments were beginning to be better understood [2]. There are several different types of methods for WWTPs which include chemical oxidation, chemical reduction, biological oxidation and biological reduction [3].

This facility requires a significant amount of electricity. A review of historical data by the Parliament of Australia [4] indicates that over time electricity prices are predicted to increase. As Council is financially supported by ratepayers, excessive spending is likely to impact the ability for the Council to delivery capital works projects. The cost of electricity for the facility has increased over the last three years in Australia. Anticipating this trend, the cost of operating the treatment plant is also forecasted to increase. The increase in cost will ultimately be borne by ratepayers either in the form of a rate increase or at the expense of little or no capital works projects.

978-1-5090-5753-5/16 $31.00 © 2016 IEEE
DOI 10.1109/APWC.on.CSE.2016.44

The WWTP in this study is operated and resourced by the Cairns Regional Council (CRC) in Queensland, Australia. The bioreactor consists of three 25kW VSD A-recycle pumps and eight 7.6kW mixers. During 2015, the WWTP had an average monthly electricity cost of $54,000 up from $52,000 in 2014.

As the price of electricity is anticipated to increase over the next decade along with the population of Cairns, an investigation into the optimal configuration of submersible mixers may result in a decrease of operation time for the mixers, or the number of mixers required for the bioreactor. Therefore a review of the current submersible mixer configuration may result in a decrease in the electricity cost to operate the facility, potentially providing financial savings to Council. This benefit could be compounded as the Council operates another treatment facility of similar configuration and are also planning a new facility.

A common type of treatment method is the activated sludge process. Raw sewage is treated in the bioreactor by replicating biological reactions similar to that which occurs naturally. The raw sewage flows through the bioreactor with each zone providing different biological reactions to assist in the removal of pollutants such as nitrogen and phosphorus compounds [5]. The WWTP bioreactor investigated is a compartmentalized activated sludge process and the main treatment component of the WWTP is the bioreactor which is shown in Figure 1 and Figure 2. The bioreactor is a multi-compartment box type structure where biological organisms break down and remove the contaminants within the sewage. The Cairns Council's Northern WWTP is an activated sludge compartmentalized bioreactor consists of the following zones.

- Four anoxic zones
- Four aerobic zones
- Three anaerobic zones

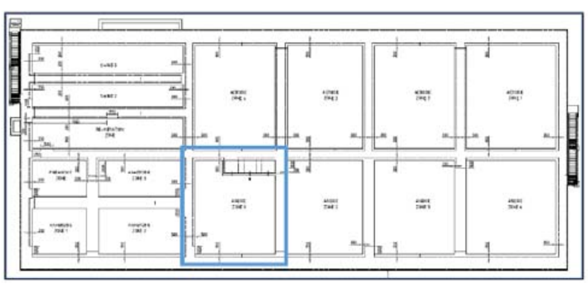

Fig 1. Top view northern WWTP

The WWTP bioreactor consists of several pumps and mixers which promote the fluid flow as shown in Table 1. The current configuration of the mixers allows for either continuous or intermittent operation. Currently the mixers are configured to run for 15 minutes before switching off for 15 minutes. This is repeated over a 24 hour period, seven days a week, all year round. This configuration was based on the original design of the plant when handed over to Council.

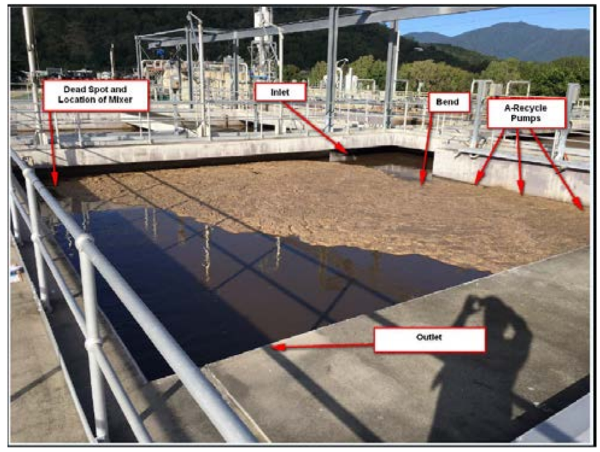

Fig. 2 Northern WWTP Anoxic Zone 1

Table1 MIXER CONFIGURATION PER ZONE IN BIOREACTOR

New Tag	New Comment	Motor Size	PLC Interface
Anoxic 1	A-recycle Pump 1	25	VSD
Anoxic 2	A-recycle Pump 2	25	VSD
Anoxic 3	A-recycle Pump 3	25	VSD
Pre Anoxic	Pre Anoxic Zone Mixer	7.6	DOL
Anaerobic 1	Anaerobic Zone 1 Mixer	7.6	DOL
Anaerobic 2	Anaerobic Zone 2 Mixer	7.6	DOL
Anaerobic 2	Anaerobic Zone 3 Mixer	7.6	DOL
Anoxic 1	Anoxic Zone 1 Mixer	7.6	DOL
Anoxic 2	Anoxic Zone 2 Mixer	7.6	DOL
Anoxic 3	Anoxic Zone 3 Mixer	7.6	DOL
Anoxic 4	Anoxic Zone 4/Swing 1 Mixer	7.6	DOL

The WWTP consists of a primary, secondary and tertiary treatment process. The majority of pollutants that can settle or float are removed by screens; this is the primary treatment process. The secondary treatment process occurs in the plants bioreactor. The treatment process simulates biological reactions similar to that which occur naturally in an aquatic environment [5]. The tertiary process involves further treatment such as membrane filtration before the effluent is discharged into a river catchment. The common process configuration for a bioreactor that includes an anoxic zone is shown below in Figure 3.

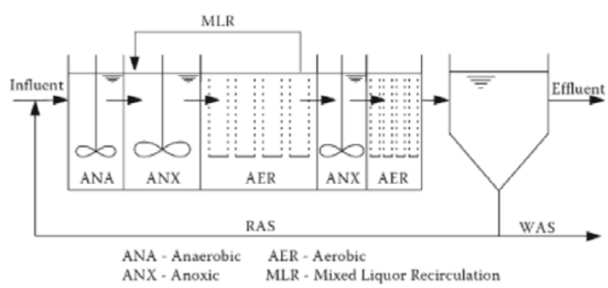

Fig. 3 Common process configuration for a bioreactor for biological nutrient removal [6].

The WWTP bioreactor consists of the following zones; Anaerobic, Anoxic and Aerobic. Each zone is designed to provide different environmental conditions to allow for the microorganisms within that zone the best opportunity to treat the influent. The concentration of biodegradable carbon and nitrogen is measured in Biological Oxygen Demand (BOD) and is a measure of dissolved oxygen required by microorganisms to convert the organic matter. Therefore excessive carbon and nitrogen in waterways will require additional dissolved oxygen to break down, and therefore potentially deprive fish and other aquatic organisms of oxygen and/or promoting the growth of algae [7]. The Queensland Department of Environment and Heritage Protection stipulate the BOD limits on the issued licenses for discharging wastewater in Queensland waterways, thus the limits can vary depending on the location and size of the treatment facility. However, a best practice environment management limit of 2mg/L for BOD is recommended [8].

The flow regime and mixing phenomena within an activated sludge bioreactor can be assessed experimentally through tracer techniques. However the size of full scale plants generally renders this unfeasible [9]. As such, Council is hesitant to modifying operations without undertaking a performance review. Computational Fluid Dynamic (CFD) is a tool to facilitate this review. To analyze the performance of the anoxic zone 1, CFD modelling was undertaken in this study to determine the fluid flow pattern. A 3D CFD model of anoxic zone 1 was developed to simulate the expected fluid flow. The CFD model results were validated by the physical data collected from the WWTP that is operated by Council and allow for a review of the fluid flow through the anoxic zone, therefore determining the optimal configuration for submersible mixers. The desired outcome of the study is to validate the current configuration of submersible mixers or to reduce the time of operation. Reduced mixer operating duration will contribute towards the decrease in operational costs by way of reducing the electricity consumption.

II. Waste Water Treatment Design

The major influence on the design of the WWTP is the desired pollutant removal rate, which can be determined mathematically. This however is based on assumed hydrodynamics of the bioreactor [10].

Traditionally, prior to modern technology, Engineers would have to undertake a repetitive and time consuming process of designing, modelling and validating bioreactor designs to ensure that the fluid flow behavior performed as expected. In addition, the mixer configuration and influence of this configuration could not be accurately modelled on a prototype designs. However as demonstrated by Brannock [10], the modelling of hydrodynamics effects of bioreactor configuration in large-scale situations can be undertaken thanks to the development of sophisticated CFD modelling.

A study undertaken to determine the influence of inlet structures or bioreactor geometry on fluid hydrodynamics using two dimensional CFD noted that; the inlet geometry had the biggest impact on flow behavior. Additionally two dimensional CFD models cannot successfully illustrate hydrodynamic behavior within a bioreactor [11]. As such a 3D CFD model would better illustrate the hydrodynamic behavior within a WWTP and the influence of bioreactor features, e.g. inlet and outlet geometry.

To overcome the assumption of hydrodynamic performance of a bioreactor and to maintain sludge concentration throughout the bioreactor, mixers could be installed in various positions to promote fluid flow [12]. CFD modelling can provided a reasonably accurate method for prediction of how the bioreactor features and mixing energy usage affects the hydrodynamics [13].

III. Minimum Vessel Velocities

As micro-organisms are used to react with nitrogen and phosphorus within the bioreactor zones, poor fluid flow may result in settlement and therefore decrease performance in the removal of biodegradable matter. To overcome the issue of settlement, submersible mixers may be used if the velocity within the zone is considered too low. As noted by Brannock [10] the minimum velocities required to overcome settlement has been extensively researched by academics and the wastewater treatment industry. Brannock [10] stated that a minimum velocity of 0.1 meter per second (m/s) would be adequate. However, it has been revised and suggested that 0.3 m/s is adequate as a minimum velocity to avoid settlement of micro-organisms.

IV. Specific Power Dissipation

The specific power dissipation is useful to determine if a zone is over powered by mechanical mixers thus contributing to waste of energy. The literature suggests [10] a target or range for the specific power dissipation and that the typical power requirements of 0.008 to 0.013 kW/m^3 for mechanical mixers within an anoxic zone. Furthermore this is supported by the recommendation from the Water Environment Federation which recommends that the power requirements for mixing as 0.01kW/m^3 [9]. To evaluate the performance of the submersible mixers, the specific power dissipation for the system can be determined using the following equation [14].

$$P_s = \frac{P_m}{V} \quad (1)$$

Where P_s is specific power dissipation in kW/m^3, P_m is mixer power in kW and V is the volume of the zone in m^3.

Another approach to determine the specific power dissipation is from the equation below that uses the dynamic viscosity of the fluid and the solid concentration [15].

$$P_s = 0.0582\mu^{0.3}C_{XS}^{0.298} \quad (2)$$

Where μ is the dynamic viscosity, Pa.s and C_{XS} is the solid concentration in W/m^3.

The concentration of solids in the raw sewage would influence the fluid properties with respect to fluid density. For simplicity, it is assumed that the primary treatment process removes all solids and floating matter from the raw sewage. Therefore, the properties of raw sewage will assume to exhibit the same properties of water. During the collection of physical data from the treatment plant, the temperature of the sewage will be recorded and this shall indicate the assumed properties relating to density and dynamic viscosity.

V. A-Recycle Pump Configuration

The anoxic zone 1 has three 25kW VSD pumps which are controlled either automatically depending on the inflow into the plant, or manually depending on the needs of the facility.

VI. Methodology

A. Experimental

The major influence on the design of the WWTP is the desired pollutant removal rate, which can be determined mathematically. This however is based on assumed hydrodynamics of the bioreactor. As said, the WWTP is fully operational in the Cairns Regional Council. Therefore, only physical data were extracted from the plant for the validation of the simulation results. Data acquisition method have already been installed in the Cairns City Council, Queensland.

B. Collection of Physical Data

Preliminary data was collected from the treatment plant and was used to validate the CFD model. Velocity readings were collected using a Marsh McBirney Flo-mate velocity probe, all which were taken at approximately 0.5 meters off the wall and at a depth of 1 meter at the locations shown (Figure 3) which are: Middle of the inlet; Bend opposite the inlet; Adjacent to the outlet; Dead Spot opposite outlet. It is noted that limited access was available to the outlet due to the configuration of guardrails.

The data for the suspended solids was recorded using a meter supplied by Council. The probe was calibrated prior to use and therefore the results obtain can be considered accurate.

C. Determination of Hydraulic diameter, Reynolds number and Density

The anoxic zone of the WWTP has two inlets of different size. The inflow into the anoxic zone is illustrated by the blue arrows in Figure 4. Based on the as constructed drawings of the treatment plant, the inflow area is 12.32 m^2 total (Inlet Top = 9.52 m^2 and Inlet Bottom = 2.80 m^2). The hydraulic diameter for inlet 1 (2.72m x 3.5m) was calculated to be 3.06m and for inlet 2 (0.8m x 3.50m) was calculated to be 1.30 m. Furthermore, the inlet of the anoxic zone was reduced to a single inlet compared to the current configuration for simplicity of the CFD model.

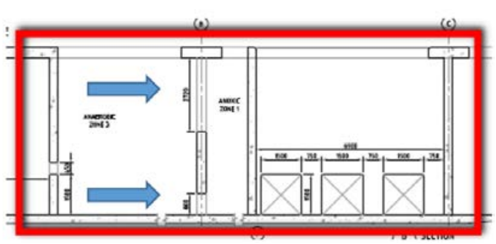

Fig. 4 Side view of the Anoxic zone inlets

Given that the fluid flow is not through a circular duct, the equation for Reynolds number must be treated differently. As such, the diameter needs to be considered as the hydraulic diameter [16]. Therefore, the corresponding Reynolds numbers calculated were 8709 for inlet 1 and 3705 for inlet 2. The flow through inlet 1 can be considered turbulent and through inlet 2 the flow can be considered transitional. However for simplicity, it was considered turbulent for the ANSYS Fluent model. Additional complexity would arise in development a CFD model that considered turbulent and transitional flow. As the inlet was reduced to a single inlet for simplicity of the CFD model, the revised hydraulic diameter for the single inlet is 3.5 meters. Therefore a revised Reynolds number for the inlet is 9984 and thus the flow is still turbulent.

Due to the specific natural of sewage and the difficulties in determining as exact value of density, it has been assumed that the density of sewage is similar to that referenced by Munson [16] for water at 29^0C, 996 kg/m^3. This assumption is based on majority of solids being removed by the primary treatment process at the WWTP. It is acknowledged that a change in the density of the fluid could alter the calculated Reynolds number and other flow properties, however for simplicity the properties of sewage was assumed as water at 29^0 C.

D. CFD Modelling and simulation

In this study, CFD modelling is the prime considerations for the prediction of flow behavior inside the bioreactor. To develop the initial model in ANSYS Fluent, the geometry of the structure must first be defined. The geometry for the anoxic zone model was developed using Design Modeler and then the geometry was imported into ANSYS Fluent for meshing. As previously stated, the CFD modelling and analysis was undertaken for anoxic zone-1 in the WWTP bioreactor that described in Figure 1 and Figure 2.

The standard $K-\omega$ turbulence model was used in this study for solving turbulent flow. It is based on transport equations for the turbulence kinetic energy (K) and the specific rate of dissipation (ω). The standard $K-\omega$ transport equation was developed by Wilcox [17] which was formulated to better compute low-Reynolds number effects, compressibility and shear flow spreading [18]. The standard $K-\omega$ transport equation is suitable for complex boundary layer flows under adverse pressure gradient and separation [19].

E. Boundary conditions

The boundary conditions of each zone were defined and are shown in Figure 5 to ensure that they had been correctly interpreted by Fluent. Walls were defined as Walls and the inlet and A-Recycle pump velocity of 0.149 m/s and 0.832 m/s were entered for the respective inlets while the outlet was selected at a pressure-outlet of 0 Pa. The liquid temperature was considered as 29°C and the density of fluid was assumed of 996 kg/m^3.

F. Mesh Indepenance study

The mesh independence study had been conducted which is illustrated in Figure 6. As seen from figure 6, the results obtained were not influenced by the model's mesh size.

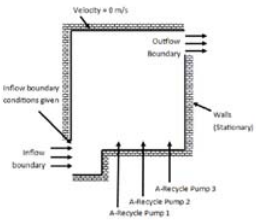

Fig. 5 Boundary Conditions of a zone.

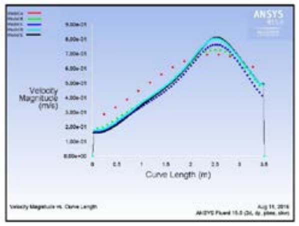

Fig. 6 Mesh independence study

VII. RESULTS AND DISCUSSION

A. 2D CFD results

The 2D model boundary conditions and geometry have neglected the influence of the 7.6 kW submersible mixer. The mixer has been neglected to evaluate the performance of the anoxic zone based on the flow from the A-Recycle pump, MRAS pump and the typical daily inflow into the plant. The plant daily inflow and the MRAS Pump flow rates have been summed together to give an Inlet velocity, for the preliminary data which is 0.149 m/s. Neglecting the mixer is illustrated the flow pattern expected within the anoxic zone and therefore establish whether submersible mixers are required.

Figure 7 and Figure 8 show the velocity and pressure profiles. As seen from these figures, the velocity is approximately 0.3 m/s and the static pressure is in the range of -0.14 to 0.24 Pascal. Furthermore, in the region where the static pressure is 2.0E-2 Pascal, the velocity of the fluid is considerably lower than 0.3 m/s which is consistent with the Bernoulli equation that highlights the relationship between a fluids velocity and the pressure for an incompressible flow [16].

As seen in Figure 9, the velocity vector for the 2D model suggests that some regions are experiencing good mixing however some regions within the anoxic zone are subjected to lower velocities than would be desired. The velocity vectors also illustrate the direction of flow within the anoxic zone and as shown in the bottom right region in Figure 20, the flow moves in a circular pattern between the outlet and the A-Recycle pump 1. This is caused from the entrainment that propagates from the A-Recycle pump. Whereas the adjacent region is experiencing considerably lower velocities which may result in stagnation. This phenomenon is a result of how the flow from the A-Recycle pump propagates.

It can be observed from figures 7-9 that within the anoxic zone 1 that short circuiting is occurring. This is happened where the flow moves directly from the inlet to the outlet and there is little interaction with the other fluid within the zone. If this is occurring, the mixing of fluids would be impacted and this may result in poor biological reactions. The region of the model in the top left corner is subjected to higher pressures than elsewhere in the zone and therefore subjected to lower velocities. From the velocity vectors, the region in the top left corner appears to circulate however at a velocity below that recommended by literature to prevent settlement. The physical implication of this would mean potential settling of micro-organism and un-desirable biological conditions.

Upon commencing the CFD simulation, it was observed that the jet stream from A-Recycle pump produced a swirling entrainment which is more noticeable on the right side. This is due to the higher static pressure on the left side of A-Recycle pump 1 due to the inflow from the inlet (Bottom left corner). The entrainment is a result of moving fluid entering a quiescent body of the same fluid, causing a velocity shear between the two fluids [20].

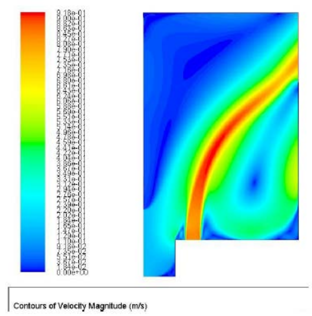

Fig. 7 Contours of velocity magnitudes

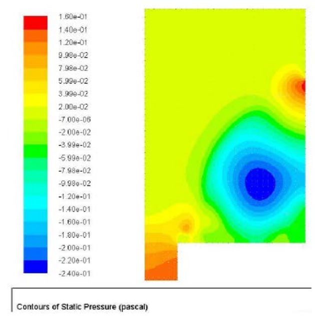

Fig. 8 Contours of static pressure

The results from the CFD model were compared with the plant data. The comparison of velocity values indicate that the CFD model simulation results were similar to those collected from the treatment plant with the CFD results showing slightly higher values than the physical data. However, the differences of the simulation data are within 3% to 10% of the physical plant data. This may be due to the limitation of 2D modelling, in that one component of the resultant velocity is neglected or as a result of the assumptions and simplification of the CFD model.

The flow sequences within the anoxic zone-1 also highlighted regions where dead spots may occur which are shown in Figure 10. They are more noticeable when the flow is fully developed. As seen, potential dead spots within the 2D model are behind and between the A-Recycle pump and right wall and at the inlet. The 3-D model showed better results to confirm if these phenomena would be likely to occur.

B. 3D CFD results

The results of the 3D model are similar to that produced by the 2D model. The flow from the high velocity A-Recycle pump was prominent in 2D modelling, furthermore the bend adjacent to the inlet was also observed to influence the flow behavior. The flow around the bend creates a small pocket of low velocity. These characteristic of the 2D model can be observed in the 3D model which can be seen from Figure 11.

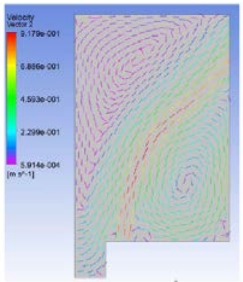

Fig. 9 Velocity vectors

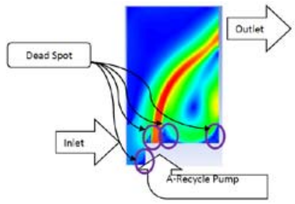

Fig. 10 2D model dead spots

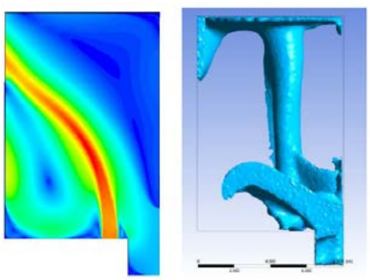

Fig.11 - Comparison of 2D and 3D flow pattern

The 3D model reflects the vertical clearance of 2.4 meters between the inlet and A-Recycle pump. As the A-Recycle pump is 2.4 meters below the top inlet, its curvature towards the outlet is not as prominent. Additionally, the flow into the anoxic zone from the top inlet is not influenced by the A-Recycle pump and therefore is able to continue along the face of the wall where the A-Recycle pumps are located. This flow pattern was not observed in the 2D model. The flow pattern around the bend is similar between the 2 simulations and therefore the similarity in results confirms the statement by Wood [11] that the inlet geometry has the biggest impact on flow behavior and that 2D CFD models cannot successfully illustrate hydrodynamic behavior within a bioreactor. As stated by Brannock [10] the minimum recommended velocity to prevent settlement of micro-organisms is 0.3 m/s.

The results from the CFD model neglecting the use of the 7.6kW submersible mixer indicate that much of the anoxic zone was subjected to velocities less than the required 0.3 m/s to prevent settlement. This is further illustrated from the streamline plots and velocity vector for the 3D model which is shown in Figure 12. From the streamline path plots, the expected flow path and the corresponding velocities can be observed. As such, the flow from the A-Recycle is prominent and is able to achieve the required 0.3m/s. Further highlighted are the regions within the zone that may experience low to no flow and this may indicate regions within the zone that are subjected to settlement. As such, the region to the left of the A-Recycle pump may be subjected to poor flow and this may result in un-desirable conditions. The 3D model results differ from the 2D model results.

The 2D model results in the region between the A-Recycle pump and outlet was achieving a velocity around 0.3 m/s, whereas the 3D model indicates this region would be subject to low velocity which is shown in Figure 12. Xie et al [21] confirmed that 3D modelling can reveal more details with respect to the flow behavior and can provide designers with a better understanding of how structure geometry influences fluid flow within a zone.

The velocity vectors in the anoxic zone provide additional details on the direction of the expected flow. It can be observed that in the dead spot location where the mixer is located, the direction of the flow is towards the surface and back towards the inlet. This is due to the flow from the A-Recycle pump 1 as it is reflected off the opposite wall. This is noticeable in the velocity vector plots for each interval below the surface which are shown in Figures 13-15. Whereas at 2 meters and 3 meters below the surface, in front of the A-Recycle pump 3, the direction of flow is downward. This as a result of the low velocity which ranges from 4.0E-3 m/s to 7.0E-3 m/s indicates that settlement may be occurring.

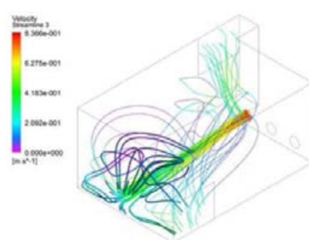

Fig. 12 3D Streamline plot isometric view

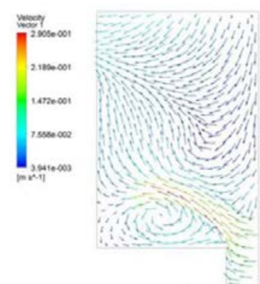

Fig. 13 Velocity vectors at 1m depth

Additionally, the narrowing of the inlet causes the velocity to increase around the bend, as it flows past the A-Recycle pump. The flow from the inlet then dissipates into the anoxic zone, the decrease in velocity results in the flow moving downwards. This is also due to the relationship between pressure and velocity. At the A-Recycle pump level of 0.75 meters above the bottom water level, the zone is subject to higher velocities and therefore a lower pressure. This contributes to the flow from the inlet to motion downward. As such, during operation of the A-Recycle pump 1 the regions in front of A-Recycle pump 3, may experience settling as a result of the low velocity and expected direction of flow. Similar results were observed by Xie at el [21] when comparing the velocity profile to the volume fraction of solids. The volume fraction of solids increased when the velocity decreased. Furthermore the accumulation of solids occurred when the velocities are relatively low. However, Xie at el [21] did not report on the corresponding velocity to cause accumulation.

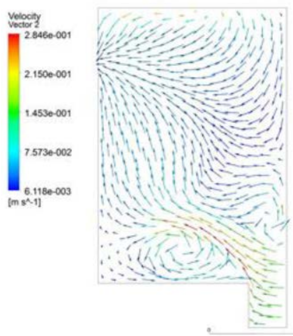

Fig. 14 Velocity vectors at 2m depth.

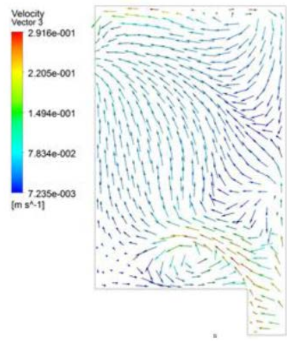

Fig. 15 Velocity vectors at 3m depth.

From the 2D model it was observed that the anoxic zone may be subject to short circuiting. Whereby the flow from the inlet and A-Recycle pump does not mix completely with the fluid within the zone. A limitation of 2D modelling is the neglecting of one component of the resultant velocity. The 3D model is able to consider this and therefore better illustrate the hydrodynamic performance. From the streamline path plot in Figure 12 and the velocity vectors in Figures 13-15, it is possible that short circuiting is occurring. As a result of the vertical clearance between the A-Recycle pump 1 and the inlet, the flow from the A-Recycle pump is able to reach the opposite side of the zone where is it deflected upwards and back along the surface. This limits the swirl entrainment observed in the 2D model between the outlet and A-Recycle pump. Therefore causing the region in front of the A-Recycle pump 3 to be subject to lower velocities, while the region within the submersible mixers produce an upward flow. The combination of these phenomenon assists to produce short circuiting within the anoxic zone. Figure 16 also shows this flow pattern from the side view velocity vector in the Y-Z plane in front of the A-Recycle Pump 3.

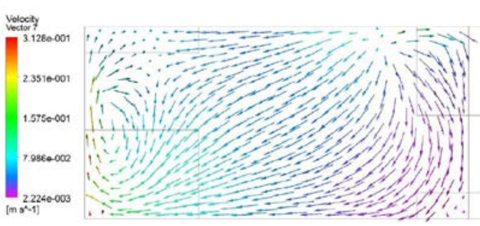

Fig. 16 Velocity Vector Z-Y plane in front of A-Recycle pump 3

The results from the CFD model simulation have been compared to the plant data. The physical data collection recorded the minimum and maximum velocity readings observed and for the comparison of results the average values have been used. Furthermore, the minimum and maximum values within the approximate location in the CFD model have also been recorded with the average values used. The difference between the simulation data and the physical data indicate that the CFD model produced valid results in comparison with the plant data.

C. *Suspended solids*

The concentration of suspended solids can be utilized to determine if a zone is wasting energy from mechanical mixers. Literature recommends a range of 8 to 13 W/m^3. The extensive data collected from the plant confirms that the anoxic zone suspended solids ranged from 2.65 g/L up to 3.41 g/L. Utilizing the equation for specific power dissipation proposed by Grady and Lim [15], considering the temperature of the liquid as 29^0 C, the corresponding dynamic viscosity is 8.15E-4 Pa.s. The density of dry sludge was assumed to be 1450 kg/m^3 as stated by Brannock [10]. Therefore converting of the suspended solids samples from g/L to kg/m^3 for the specific power dissipation calculation is given by; Inlet = 1.96 kg/m^3. Therefore considering the concentration at the surface for the inlet, the specific power dissipation for the anoxic zone is given by: Ps = 8.4 W/m^3. Based on the equation for specific power dissipation in equations (1) and (2) at section IV, the results indicate that the anoxic zone is marginally above the lower limit for the recommended specific power dissipation. The results for specific power dissipation range from 8.2 W/m^3 to 8.9 W/m^3. Furthermore, the results from the four locations indicate that settlement may be occurring. The concentration of solids increased as the depth increases. As such, considering the dead spot location, at the surface the suspended solids recorded was 3.19 g/L and at a depth of 3 meter was 3.41 g/L. This is a 6.5% increase in solid concentration over a 3 meter depth. As seen from Figure 17, the extrapolated results of this trend indicate that at 5.88 meters below the surface, the expected solids concentration would be 3.6 g/L which corresponds to 9 W/m^3. Based on the extrapolated results at a depth of 5.88 meters below the surface, the concentration of solids in grams per liter would be expected at the following locations: Inlet (3.23g/L), Bend (3.51 g/L), Dead Spot (3.95 g/L) and Outlet (3.68 g/L). Therefore the difference between the surface concentration of solids and the depth of 5.88 meters would be as follows: Inlet (12%), Bend (21%), Dead Spot (32%) and Outlet (13%).

Fig. 17 Extrapolated results of suspended solids

VIII. Conclusion

A comprehensive CFD (2D and 3D) model of an activated sludge process bioreactor anoxic zone 1 was developed to evaluate the hydrodynamic performance and influence of the inflow through the various inlets. The results of the study highlighted the limitation of 2D modelling in that it is not able to simulate the true hydrodynamic performance. This was observed when comparing the simulation results between the 2D and 3D models. The results of the 2D model slightly differed from that of the 3D model due to neglecting one component of the flow field (e.g. velocity, pressure). These findings align with the statement by Wood [11] that 2D models cannot illustrate accurately the hydrodynamic performance. Therefore the application of submersible mixers would be required. The CFD results for the 3D model indicated that without the submersible mixer, much of the anoxic zone is less than the required velocity stated by Brannock [10] of 0.3 m/s to prevent settlement. The results from the CFD simulation indicated that the anoxic zone may be subject to short circuiting and this is illustrated from the streamline plots and velocity vectors within the zone. The implication of this would results in un-desirable biological conditions as there may be increased retention time of micro-organisms. Furthermore settling of micro-organisms may occur within the zone as a result of low velocities. Regions within the anoxic zone were observed to experience low velocities and furthermore, some of these regions displayed flow in the downward direction, further contributing to settlement. The results of the suspended solid concentration while considering the fluid as water indicated that the zone is marginally above the recommended range of 8 W/m^3 and 13 W/m^3. However further investigation of the influent properties would assist to re-calibrate the CFD model and specific power dissipation. By incorporating into the CFD simulation, oxygen mass transfer, carbon oxidation, nitrification and denitrification it would allow for a better understanding of the influence of hydrodynamic performance and therefore the optimal configuration for submersible mixers. Further investigation should be undertaken of the plants performance as the bioreactor typically operates under low flow conditions. The physical data from the plant indicated that much of the anoxic zone is subject to velocities below 0.3 m/s. The reason for this may be due to the bioreactor configuration which has the 4 anoxic zones in series. This may result in the bioreactor preforming the required biological process over the four zones and therefore a lower velocity is acceptable. This is also considering that the concentration of suspended solids was also low, at 2.8 g/L to 3.4 g/L.

The study was not able to determine the optimal configuration of submersible mixers within the bioreactor. This was not able to be achieved as the CFD model considered the properties of fluid as water, which resulted in higher velocities compared to the physical data. Anoxic zone 1 may be able to operate at a low velocity due to the inflow from the a-recycle pumps, however this may not be the case for anoxic zone 2. Therefore a further study is warned to simulate all the anoxic zones to determine the optimal configuration of submersible mixers.

ACKOWLEGEMENTS

Authors greatly acknowledge the Cairns City Council for their experimental data. The authors also acknowledge i-Lab-Australia for their financial support for the free registration of this conference.

REFERENCES

[1] R. Hriez,. M. Latifi, and N. Roche, "Optimal design and operation of ac.tivated sludge processes: State-of-the-art", Chemical Engineering Journal, 281, pp.900-920, Elsevier, 2015.

[2] G. Tchobanoglous. and F.L.Burton, "Wastewater engineering. Management," 7, pp.1-4, 1991.

[3] L. O. Kolarik, A. J. Priestley, "Modern Techniques in Water and Waste Water Treatment," CSIRO Publishing, Melbourne, Australia, 1995.

[4] Parliament of Australia, Parliamentary Library Briefing Book, Energy prices—the story behind rising costs, viewed online 28th February 2016 at, http://www.aph.gov.au/About_Parliament/Parliamentary_Departments/Parliamentary_Library/pub s/BriefingBook44p/EnergyPrices

[5] S. R. Qasim, "WasteWater Treatment Plants: Planning, Design, and Operation," 2nd Edition, Technomic Publishing Company, Lancaster, Pennsylvania, 1999.

[6] G. Daigger, C. L. Grady C. N. Love, C. Filipe, 2011, "Biological Wastewater Treatment," 3rd Edition, CRC Press, New York, New York., 2011.

[7] B. Oram, "Phosphate in Water: Total Phosphorus and Phosphate impact on Surface Water,: Water Research Centre, N.D., viewed online 31st March 2016 online at http://www.water-research.net/index.php/phosphate-in-water

[8] Environment & Heritage Protection (EHP), "Wastewater release into Queensland waterways," viewed 31st March 2016 online at https://www.ehp.qld.gov.au/assets/documents/regulation/pr-gl-wastewater-to-waters.pdf .

[9] A. Karpinska, and J. Bridgeman, "Review: CFD-aided modelling of activated sludge systems – A critical review," Water Research, 88, pp. 861-879, Elsevier, 2016.

[10] M. Brannock, 2003, "Computational Fluid Dynamics tools for the design of Mixed Anoxic Wastewater Treatment Vessels," August 2003, viewed 20th March 2016, online at http://espace.library.uq.edu.au/view/UQ:106525 .

[11] Wood, MG Howes, T Keller, J & Johns, MR 1997, Two dimensional Computational Fluid Dynamics models for waste stabilisation ponds, Water Research Vol. 32, Issue. 3, pp958-963.

[12] Randall, CW Barnard, JL & Stensel, HD 1992, Design and retrofit of wastewater treatment plants for biological nutrient removal, Technomic Publishing Company, Lancaster, Pennsylvania.

[13] M. Brannock, "Mixing characteristic of full scale membrane bioreactors: CFD modelling with experimental validation," Water Research, Vol. 44, Issue. 10, P.1., 2016.

[14] T. R. Cumby, "Slurry mixing with impellers: Theory and previous research," Journal of Agricultural Engineering Research, Vol 45, Issue 3, pp157-173 , 1990.

[15] C. Grady, and H. Lim, "Biological wastewater treatment: Theory and application," New York, Basel, Dekker, 1980.

[16] B. Munson, T. Okiishi, W. Huebsch, and A. Rothmayer, 2013, "Fluid Mechanics: SI Version, 7th edition, John Wiley & Sons, Singapore, 2013.

[17] D. C. Wilcox, "Turbulence Modeling for CFD, 3rd edition, DCW Industries, Inc., La Canada CA, 2006.

[18] M. Cable, " An evaluation of turbulence models for the numerical study of forced and Natural convective flow in Atria," A PhD Thesis, Queen's University, Ontario, Canada, 2009.

[19] ANSYS Inc, "Training Agenda: Introduction to ANSYS Fluent 15.0; Workshop material," 2013, viewed 7th August, 2016.

[20] S. Krishna,"Characterization of the flow-field in circular subsonic impinging jets: Investigation of surface pressure fluctuations and far-field noise emissions through pressure sensor measurements," MSc Thesis, Delft University of Technology, 2016.

[21] Xie, H. Yang, J. Hu, Y. Zhang, H. Yang, Y. Zhang, K. Zhu, X. Li, Y. Yang, C. 2014, "Simulation of flow field and sludge settling in a full-scale oxidation ditch by using a two-phase CFD Model," Chemical Engineering Science Vol. 109, pp296-305, Elsivear, 2014.