CFD Modeling of Mesophilic Sludge Digester

Franz Jacobsen[*1], Eugene de Villiers[2], Francisco Campos[3], Paolo Geremia[4]

[1-4]ENGYS AUST Pty Ltd Brisbane Australia, ENGYS Ltd London UK,ENGYS Ltd London UK,ENGYSSrl, Trieste, Italia

[*1]f.jacobsen@engys.com;[2]e.devilliers.engys.com;[3]f.compos@engys.com; [4]p.geremia@engys.com

***Abstract*-The used of Computational Fluid Dynamics (CFD) modeling techniques for the design or upgrade of sludge digesters has significant potential for cost savings; however, validation of modelling techniques is essential for widespread acceptance of this methodology. Queensland Urban Utilities (QUU) is planning to upgrade its four primary mesophilic anaerobic digester tanks atOxley Creek Water Recycling Plant (WRP) in Brisbane, Australia. A CFD model was used to study the existing arrangements of the sludge digesters to determine the predicted effective volumes in their present configuration and energy inputs. Modifications to the digester tank were proposed to improve the effective mixing volume. A validation of the CFDmodeling software for non-Newtonian sludge digesters was conducted using physically measured data obtained from a study based on a similar sized sludge digester located in California.**

Keywords-Computational Fluid Dynamics; CFD; Waste Treatment;Sewerage; Mesophilic;OpenFOAM;HELYX; ENGYS

I. INTRODUCTION

QUU is planning to upgrade four of the primary digester tanks at Oxley Creek Water Recycling Plant (WRP), located in Brisbane Australia. CFDmodeling was conducted to determine mixing compressor capacity requirements. The purposes of this study are listed as follows:

- To determine if the existing mixing compressors can provide sufficient mixing energy to achieve 90% effectively mixed volume for Tanks 1 and 2 (which are identical), based on the current typical viscosity curve for digester sludge.
- To determine if the existing mixing compressors are able to provide sufficient mixing energy to achieve 90% effectively mixed volume for Tanks 3 and 4 (which are identical).
- To determine the optimum mixing compressor capacity for each type of arrangement, to achieve 90% effective mixing.

The key study deliverable was to determine the predicted effective volumes for digesters 1 and 2 and digesters 3 and 4 under their present configuration and energy inputs.

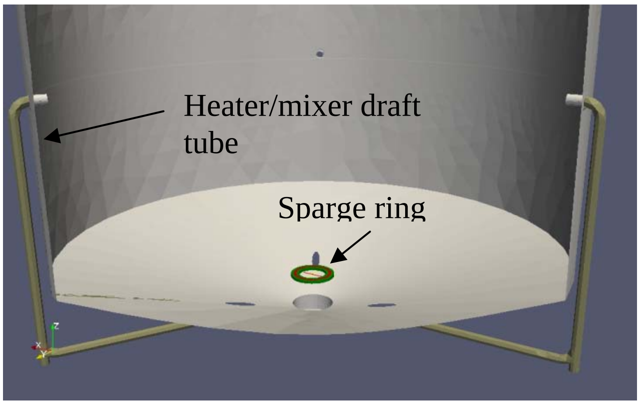
CFD modeling is an effective cost saving method for the optimization of sludge digesters. However, it requires careful consideration and experienced judgment to be utilized meaningfully. Some form of model validation and calibration against physically measured data is essential. Lithium ion trace testing data has recently been made available for the sludge digesters which are the subject of this paper; however, the data was made available too late for inclusion in this paper and is the subject of ongoing work. Alternatively, a validation case is presented at the end of this paper, based on physically measured data obtained from a previous paper[3]. The validation model was based on a similar sized sludge digester located in California, USA.

A. General Arrangement

The four digesters at Oxley Creek WRP each have a diameter of 18.3 meters. Top operating level to the base of the wall is 9.84 meters. The digester floor has a 15 degree sloping floor to a central 1.8m diameter sump. Digesters 1 and 2 have a central unconfined sparge and three 12” heater/mixer draft tubes. A sample of the CFD model geometry is shown inFig. 1(a)in whichthe central sparge ring and two of the heater/mixer draft tubes are visible.

Digesters 3 and 4 have three 500mm vertical draft tubes, equally spaced 5.45 meters from the center of the tank as shown in Fig. 1(b).Each bottom end of the draft tubes has conical fluting. Mixing gas is released through a sparge ring at the base of the draft tubes. The single heater circuit (not shown) consists of 45m of 150mm pipe.

DOI: 10.5963/JWRHE0402006



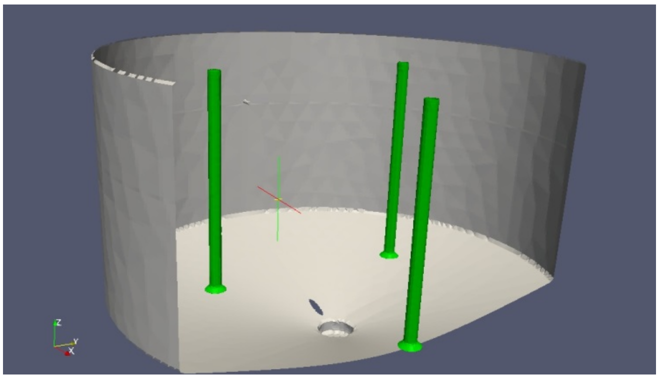


Fig.1(a) Cut-away view of existing model geometry of digesters 1-2; (b) General view of digesters 3 and 4

The sludge heater draws sludge from the center of the tank, just above the sump and discharges roughly at mid-level in the tank.

II. METHODOLOGY/ PROCESS

A literature review was undertaken to gain an appreciation of the flow characteristics in the draft tubes, also referred to as airlifts or gas pistons in various texts. To confirm the selection of the appropriate CFD solver, it was necessary to determine the flow regime as described in Fig.2. The outlined method of using a low regime map for vertical gas-liquid flow was employed to classify the flow regime in the draft tube [5]; it was found to fall within the bubble flow region, which confirms the section of the solver.

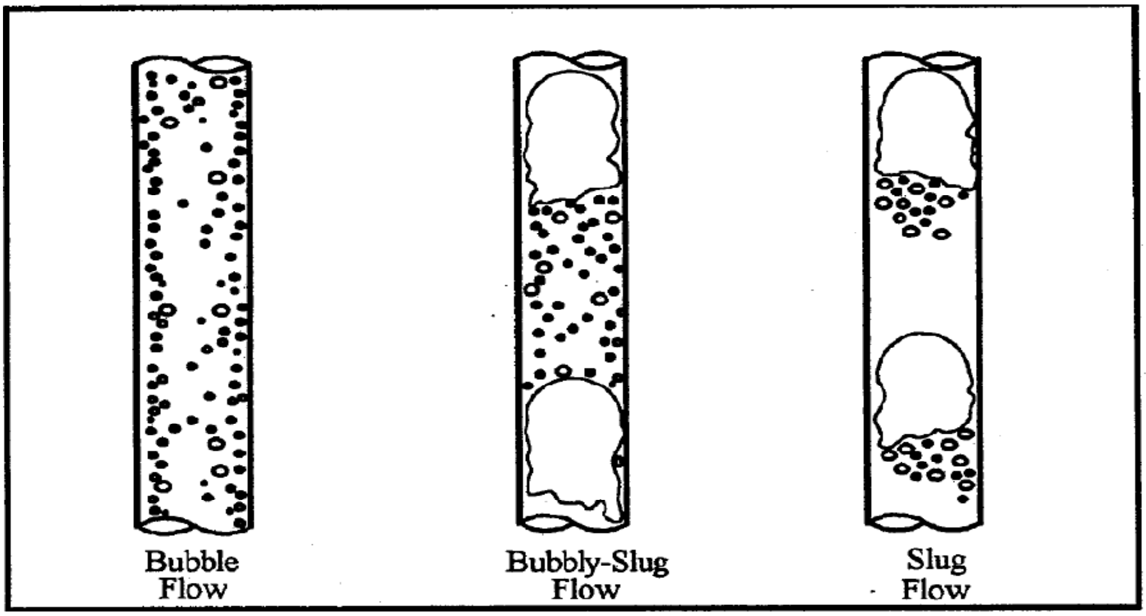


Fig.2Two-phase flow regimes in airlift pumps as air input increase [4]

A. Rheology

The viscosity of the sludge adopted for the CFDmodeling is based on site-specific rheological evaluation. The rheological evaluationwasconducted using samples from digester 1. The fluid properties adopted for this study correspond to the results of sludge samples tested at 37°C, 5% solids. The result of the rheological tests is shown in Fig.3. The CFD model utilizes a power law function to account for the non-Newtonian (shear rate thinning) viscosity of the digester sludge.

DOI: 10.5963/JWRHE0402006

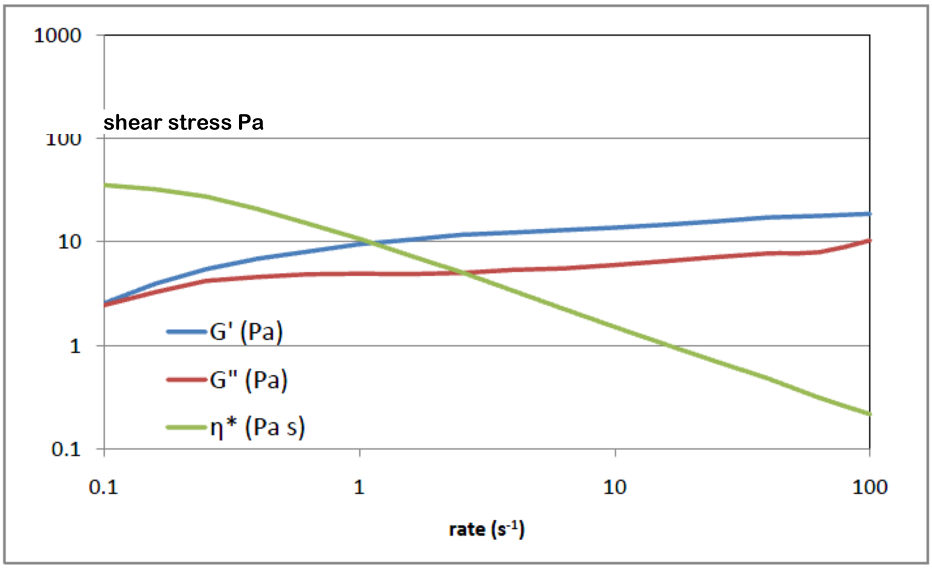


Fig. 3 Rheology test data at 37°, 5% solids.Viscosity vs. shear rate, from [2]

B. CFD Model

The numerical software used in this study wasHELYX, developed by ENGYS. HELYX is based on OpenFOAM [1], which is produced by OpenCFD Ltd. HELYX includes open source utilities and solvers that can simulate complex fluid flows involving chemical reactions, turbulence and heat transfer.

Two solvers were used in the study. Two-phase solver bubble Foam was used to evaluate flow rates, in which air phase is pumped into the continuous sludge phase. A single-phase solver was used for the overall model, which is comprised of the entire tank. A number of separate models were developed utilizing the multiphase solver, as described in the results section. Models were developed for the draft tubes and sparge rings in digesters 3 and 4 and the heater-mixer pipes in digesters 1 and 2. The results of the two-phase models were used to establish boundary conditions for the overall single-phase model.

C. Numerical Approach

The HELYX solver used in this study is a two-phase solver based on the Euler-Euler two-fluid methodology, which is suitable to compute dispersed gas-liquid and liquid-liquid flows. In the Euler-Euler two-fluid approach, the phases are treated as interpenetrating continua, which are capable of exchanging properties such as momentum, energy and mass. The model undergoes the following assumptions:

- Phases are incompressible
- The dispersed phase particle diameter is constant
- The flow is isothermal
- Only momentum exchange is accounted for in the momentum transport equations.

HELYX also incorporates turbulence with the non-Newtonian two-phase solver. The turbulence model used is the Reynolds-Average Navier-Stokes (RANS) equation. A low Reynolds number model was used to match the low turbulent characteristics of the fluid.

Eulerian conservation equations are used to describe both phases in the two-fluid model. Each of the phases is treated as continuous and inter-penetrating, and is represented by averaged equations. The equations implemented in OpenFOAM solver are given below. The equations for the two fluid modelling approaches in OpenFOAM are implemented from Rusche (2002). The averaged interphase momentum transfer term accounts for the transfer of momentum between the two phases. The averaged momentum and continuity equations for each phase φ can be written as:

$$\frac{\partial \alpha_\varphi \overline{U}_\varphi}{\partial t} + \nabla . \left(\alpha_\varphi \overline{U}_\varphi \overline{U}_\varphi\right) + \nabla . \left(\alpha_\varphi \overline{R}_\varphi^{eff}\right) = \frac{\partial \alpha_\varphi \overline{U}_\varphi}{\partial t} + \nabla . \left(\alpha_\varphi \overline{U}_\varphi \overline{U}_\varphi\right) + \nabla . \left(\alpha_\varphi \overline{R}_\varphi^{eff}\right) \tag{1}$$

$$\frac{\partial \alpha_\varphi}{\partial t} + \nabla . \left(\overline{U}_\varphi \alpha_\varphi\right) = 0 \tag{2}$$

DOI: 10.5963/JWRHE0402006

where the subscript φdenotes the phase;αis the phase fraction;R_{φ}^{-eff} is the combinedReynolds number (turbulence) and viscous stress; andM_{φ} is the averaged interphase momentum transfer term.

D. CFD Computation Domain Characteristics

The mesh resolution was determined by grid size sensitivity analysis, and consists of an unstructured grid. The time step is controlled as a function of the adopted dimensionless Courant number value (0.5). The Courant number is defined as:

$$C=u\Delta t/\Delta \quad (3)$$

where u = the magnitude of velocity Δt = the time step and Δx= grid size interval.The geometry from a triangulated surface file was used to generate the computational mesh, which consists of an unstructured mix of polyhedral hexahedral cells. The spacing of the mesh is 25mm near the wall boundaries and approximately 100mm elsewhere in the region of the digester tank. The dimensionless value of y+ indicates the required resolution of the grid spacing in the boundary laminar layer:

$$y+ = \rho * u * y / \mu \quad (4)$$

whereρ = density, u = friction velocity, y = distance from wall, μ = dynamic viscosity. The first grid point is located at approximately y+ = 20-80 along walls; near-wall flows are resolved using the near-wall function.The mesh was generated using the mesh utility in theHELYX software package, which allows the user to define a range of parameters such as regions of refinement and the number of boundary layers.

Fig.3 3provides an overview of the mesh. The mesh consists of approximately 2,800,000 cells. The mesh quality was found to meet normal modeling guidelines for aspect ratios, skewness and orthogonality.

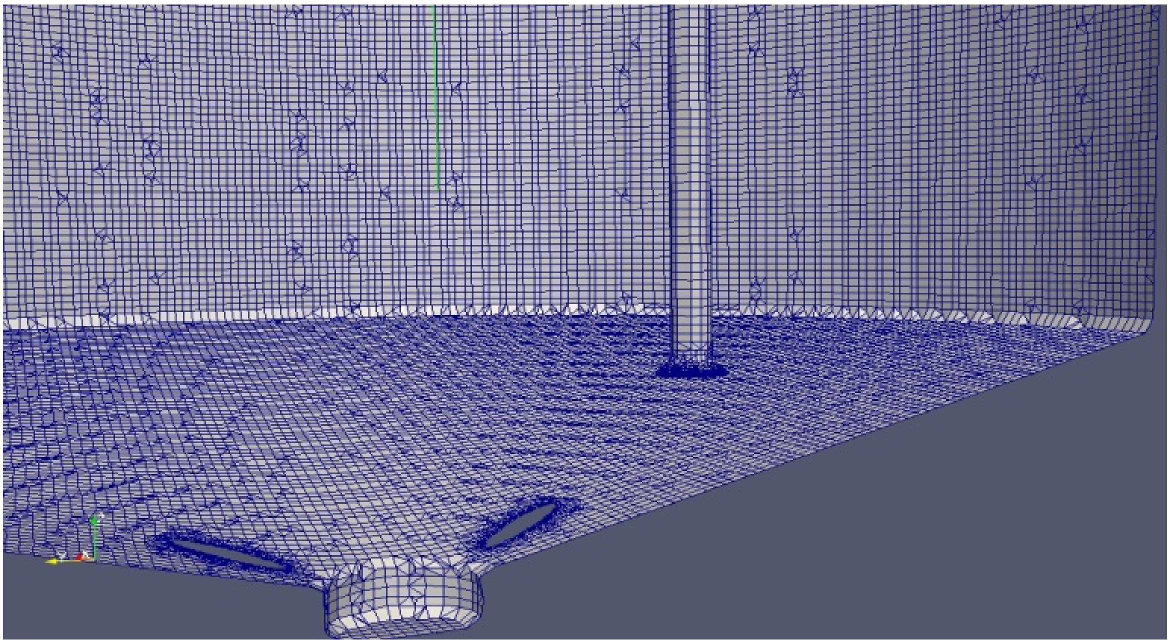
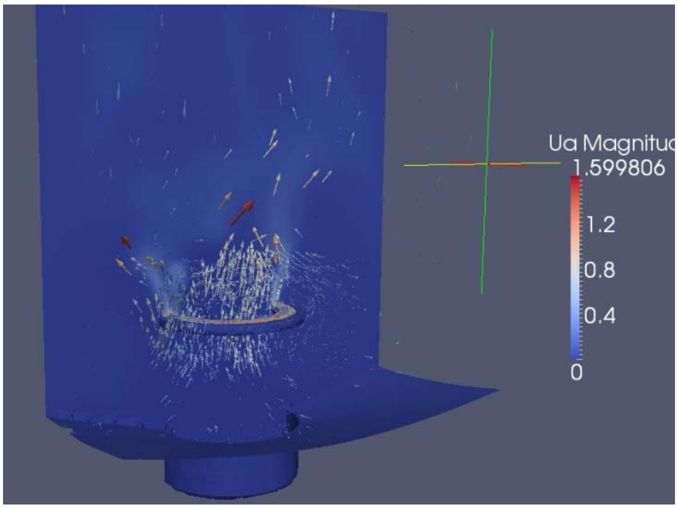


Fig.3Mesh sample, with draft tube visible in the background

III. RESULTS

Various models were developed using the two-phase solver as described below. The results of the two-phase solver models were used as the boundary flow conditions for the overall single-phase solver. This approach was adopted to reduce the computation modeling time.Fig. 5(a)shows the dispersed bubbles phase emanating from the central sparge used in digesters 3 and 4, which is indicated by the vectors. Fig. 5(b)similarly indicates the velocity of the bubbles (Ub). Fig.5heat mixer draft tube of digesters 1 and 2 in a two phase sub model (truncated for clarity). The air phase and dispersed bubble are shown in red, while the liquid phase is shown in blue.

DOI: 10.5963/JWRHE0402006



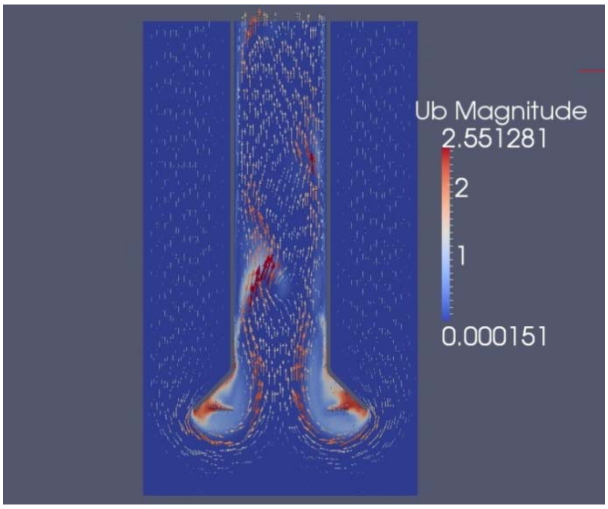


Fig.4(a) Two-phase model of digester 3 and 4 heat mixer draft tube; (b)two-phase model of digester 1 and 2 sparge ring model

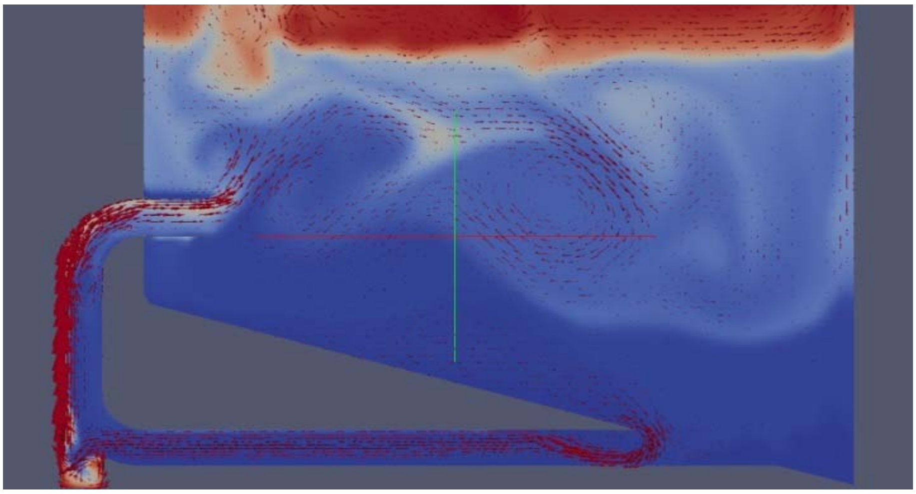


Fig.5Two-phase sub-model of digester 1 and 2 heat mixer draft tube (truncated for clarity)

The existing arrangement of digesters 1 and 2 was assessed using CFDmodeling tools, and was found to demonstrate mixing efficiency based on a proportion of tank velocity greater than 0.025m/s, or approximately 75%. The existing arrangement of digesters 3 and 4 was found to have a mixing efficiencyof 92% which meets the project criteria of 90%. Digesters 1 and 2 were modified by rotating the mixer nozzles at 45 degrees to the horizontal plane, as shown in Fig.10. The revised arrangement was found to have a mixing efficiency of 89%, which was deemed acceptable. A graphical representation of the effective mixing volume for digesters 1 and 2 is shown in Fig.6; digesters 1 and 2 are shown in Fig.8. A plot of the velocity profile shown in plan with velocity vectors is shown in Fig.7 for digesters 1 and 2 and in Fig.9 for digester 3 and 4.

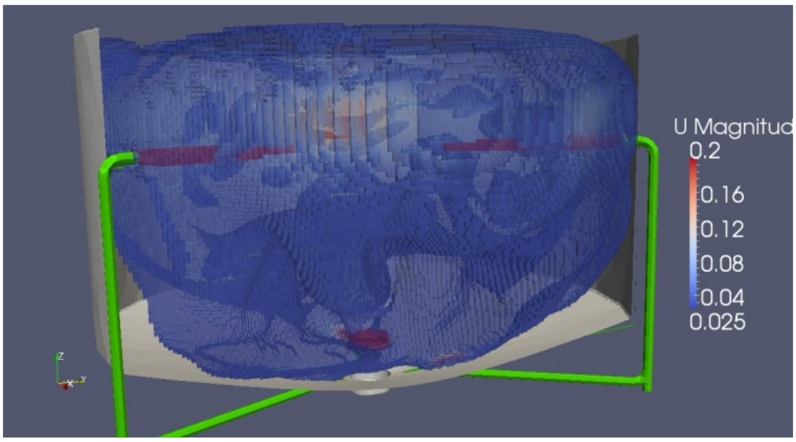


Fig.6 Digesters 1 and 2: active volume

DOI: 10.5963/JWRHE0402006

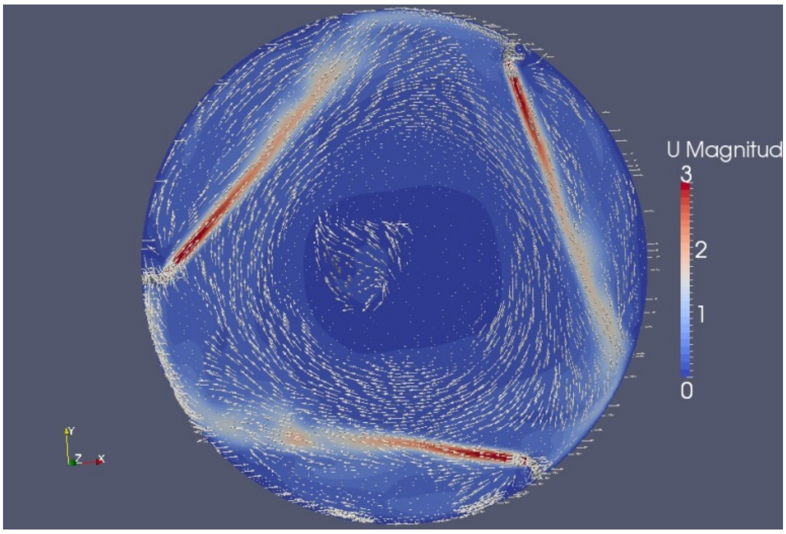


Fig.7Digesters 1 and 2: plan velocity profile

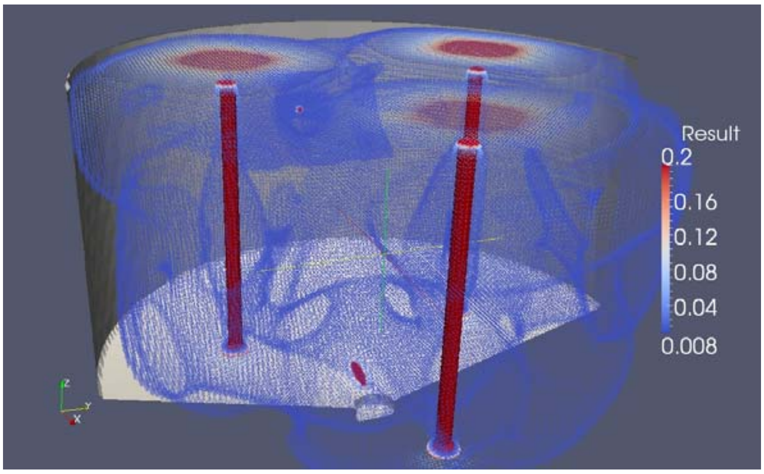


Fig.8 Digesters 3 and 4: active volume

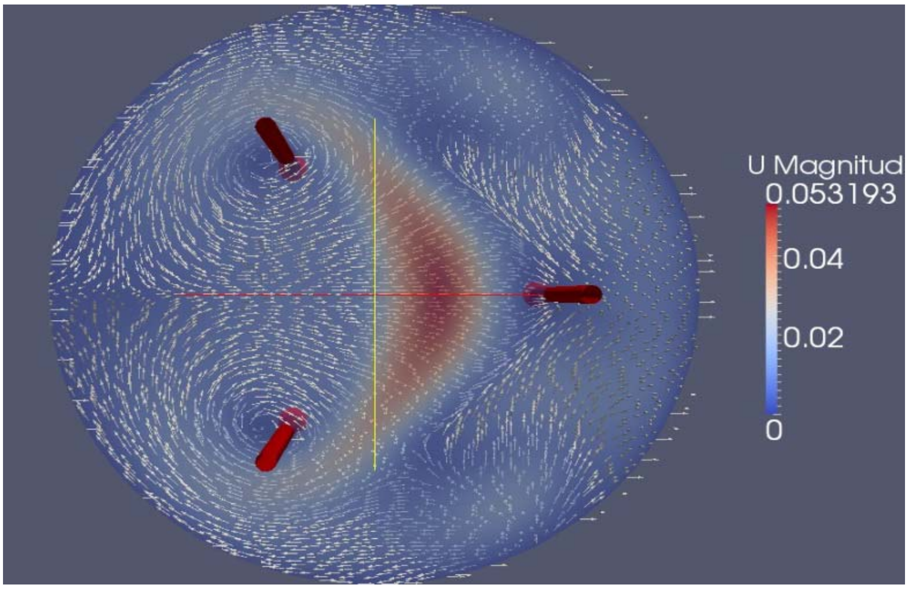


Fig.9Digesters 3 and 4: plan velocity profile

DOI: 10.5963/JWRHE0402006

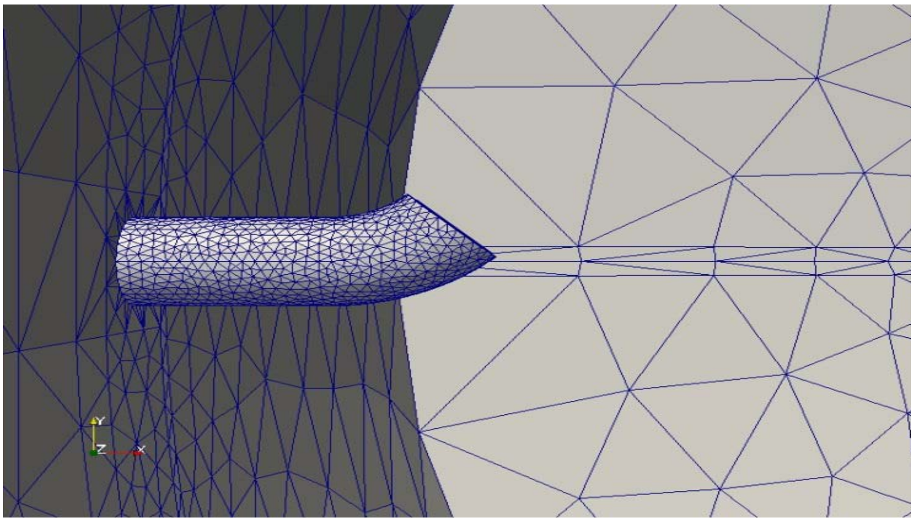


Fig.10Digesters 1 and 2: nozzle rotated at 45 degrees (development case)

A. Model Validation

A validation of the CFDmodeling software for non-Newtonian sludge digesters was conducted using physically measured data obtained from a paper published in 2006 (James J. Marx et al. 2006) [3]. The validation model was based on a similar sized sludge digester located in California, USA.

The lower active volume HELYX results are due to the 2006 model's inability to compute turbulence for its Euler-Euler two-fluid solver.Site-specific sludge rheology data was used for the study. The rheology data was incorporated into a scaled physical model of the digester mixing system, which was the basis for the calibration of a CFD model used to assess digester mixing. The model study was recreated in the present study to validate the CFD methodology.

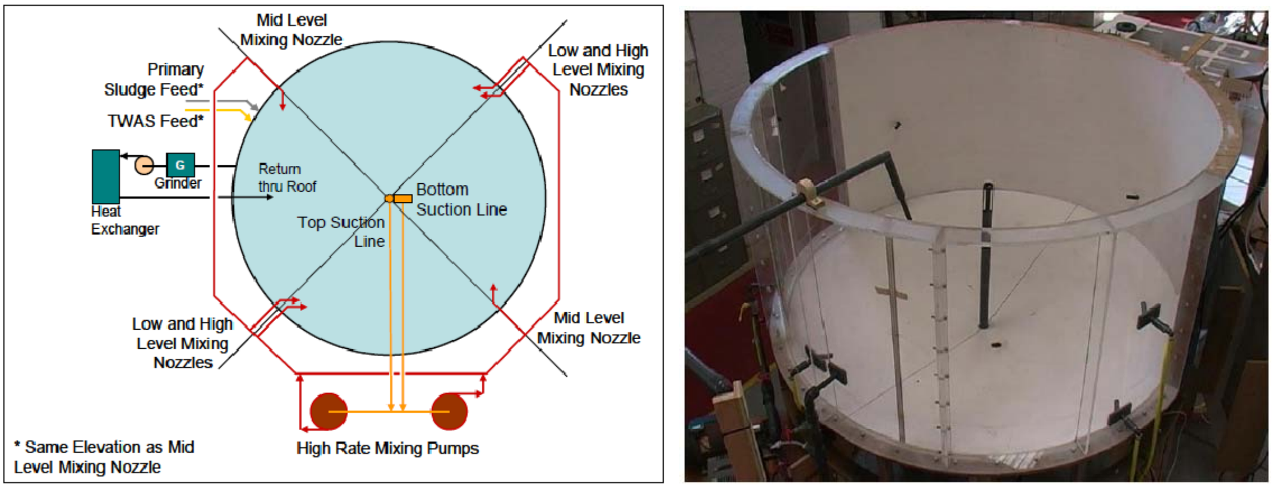


Fig.11(a) Physical model schematic layout; (b) photo of scaled physical model

The layout of the model and rheology input data is shown in Fig.11(a) and (b), and in Fig. 13.

DOI: 10.5963/JWRHE0402006

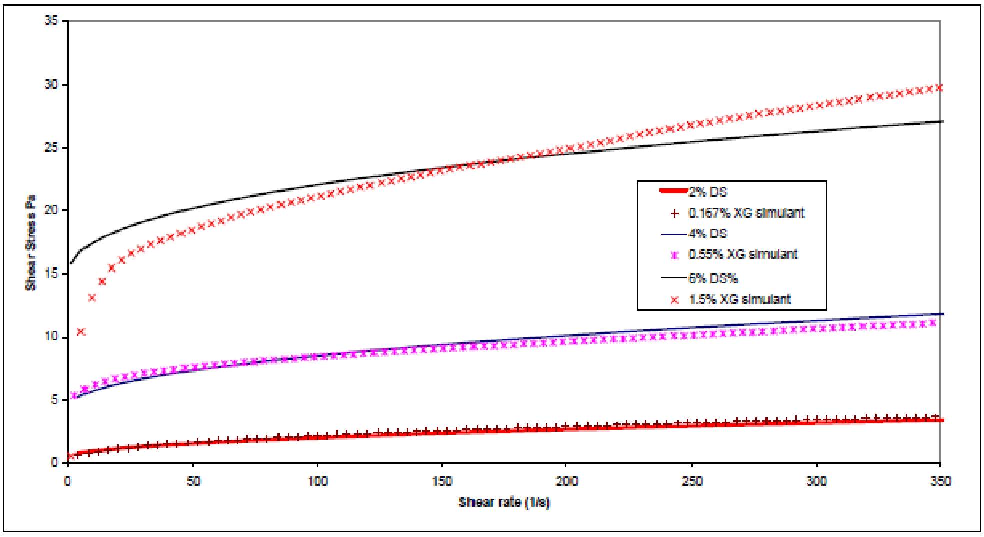


Fig.12Comparison of simulant and sludge rheology

B. *Validation Results*

Calibration of a laminar, non-Newtonian CFD model against velocity measurements in a scale model digester demonstrated good agreement at low sludge concentrations or laminar/transitional jet Reynolds numbers. The overestimation of velocities using CFD with high sludge concentrations was due to the lack of a turbulence solver.

A comparison of results shows excellent correlation between the 2006 and 2010 CFD studies as shown in Table 1. The lower active volume HELYXresults are due to the 2005 model's inability to compute turbulence.

TABLE 1COMPARISON OF RESULTS

Case	Total flow rate (gpm)	Active volume % (V >0.1m/s)	
		2006 Fluent study results	HELYX results
1	9500	62	58
2	12000	72	65

A comparison of plots with excellent correlation is shown in Fig.14andFig.15.The apparent differences in flow regions between the 2006 study and the present study in Fig. 15 are due to the color graduation scale. Fig. 14 shows close agreement in the active volume plots.

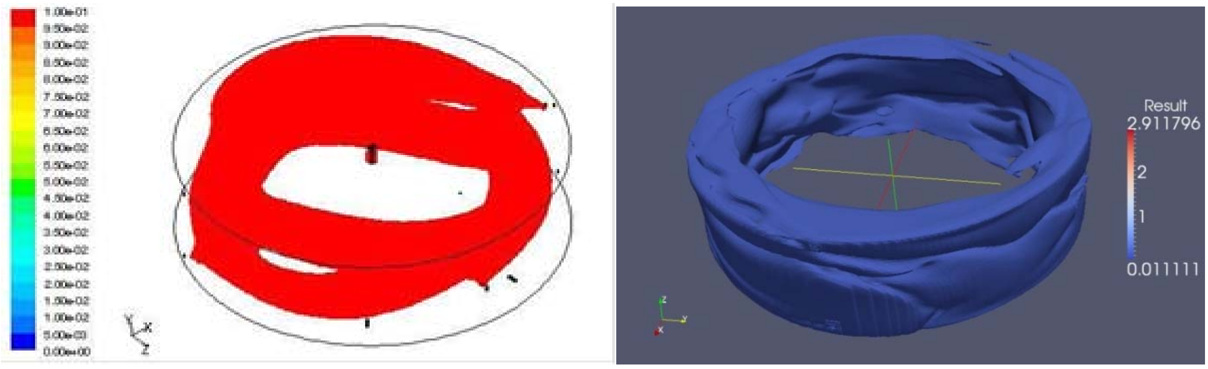


Fig. 14(a) Active wolume of CFDmodel dry solids, Fluent 2006; (b) HELYX 2010

DOI: 10.5963/JWRHE0402006

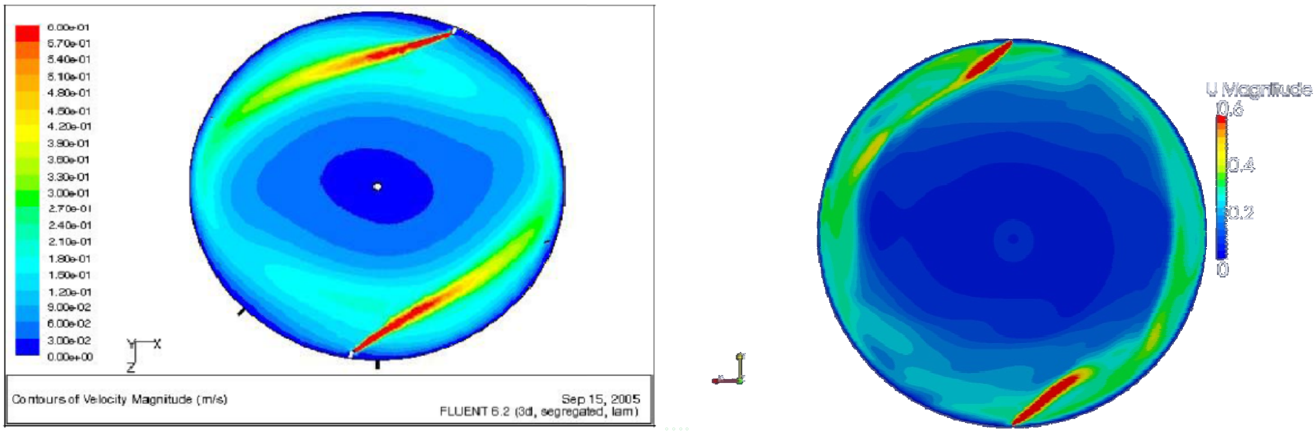


Fig. 15(a) Horizontal section through CFD model at 4% dry solids, Fluent 2006; (b) HELYX2010

IV. CONCLUSION

A CFD model was used to study the existing arrangements of the sludge digesters at Oxley Creek WRP. The CFD model was able to determine the effective mixing volume of the sludge digesters. Modifications were made to the CFD models, which were adopted by QUU, to improve mixing volumes as required. A validation of the CFDmodeling software for non-Newtonian sludge digesters was conducted using physically measured data obtained from Marx et al. (2006) [3]. The validation model was based on a similar sludge digester located in California, USA. This study demonstrates the practicality and robustness of using CFD models for sludge digesters, and also demonstrates the potential cost benefits to sludge digester operators.

ACKNOWLEDGMENT

I would to thank Queensland Urban Utilities for their kind permission to submit this paper, and especially the kind support from HaraldKemmetmuller. I also owe special gratitude to Eugene de Villiers for his assistance in various aspects of setting up the CFD models.

REFERENCES

[1] OpenCFD, "The Open Source CFD Toolbox User Guide, OpenCFD,"2009.

[2] T Nicholson, "Reological evaluation on Oxley Creek Digester," UniQuest, 2010.

[3] James J. Marx et al, "Applying Rheological Techniques to upgrade Anaerobic Digerters and Handel High solids Concentrations,"Monsal Limited Mansfield, England, 2006.

[4] Reinemann, D.J. and Timmons, M.B., "Prediction of oxygen transfer and total dissolved gas pressure in airlift pumping," Journal of Aquacultural Engineering, 8, 29-46, 1989.

[5] Patel, "CFD Simulation of Two-phase and Three-phase Flows in Internal-loop Airlift Reactors," LAPPEENRANTA UNIVERSITY OF TECHNOLOGY, Faculty of Technology, Department of Mathematics and Physics, 2010.

[6] HenrikRusche,"ComputationalFluidDynamicsofDispersedTwo-PhaseFlowsatHighPhaseFractions," Thesis, Dpt. Of Mech.Eng., Imperial Collage, London, 2002.

DOI: 10.5963/JWRHE0402006