

THE USE OF COMPUTATIONAL FLUID DYNAMICS IN THE OPTOMISATION OF SLUDGE PROCESS

D. Harrison , M.Dawson***, S.Rafique*** and T.Firmansyah****

* *Monsal, Oak House, Ransom Wood Park, Southwell Road West, Mansfield, NG21 0HJ*

*** *BHR Group Limited, The Fluid Engineering Centre, Cranfield, Beds, MK430AJ*

ABSTRACT

The paper outlines a project to review the anaerobic digestion plant No. 1 for the Orange County Sanitation District, California. The study, investigating all aspects of the plant, thickening, pumping, anaerobic digester heating and mixing and gas distribution, has drawn on fundamental rheological analysis of sludges along with scale and computational mixing models to predict the effect on the digesters of increasing feed dry solids from 4% to 9%.

Traditionally in the US, waste water plants operate with feed sludge dry solids of <4% and therefore 2% to 2.5% digested sludges. Digested sludge at this concentration will tend to approximate Newtonian fluids very similar to water, making both pumping and mixing relatively easy. However recent trends towards mechanically and chemically thickened sludges (up to 14% dry solids) have highlighted the difficult nature of the material, a Bingham plastic with a high yield stress and shear thinning properties. In turn this has led to problems with pumping and digester mixing, resulting in undersize pumps, foaming and short circuiting.

The paper discusses the engineering feasibility study carried out for Plant No. 1 within Orange County Sanitation District and in particular the mixing investigation. Two novel techniques for testing and utilizing site specific rheology data to produce hydraulic and pump evaluations are presented, and the use of a scale model and computational fluid dynamics are reviewed for the assessment of digester mixing.

As part of the hydraulic and pumping evaluation study carried out at Plant No. 1, rheological testing of 3 sludge streams was carried out. Samples were taken and then thickened to give a range of properties from 2% to 9% dry solids. This site specific rheology allows accurate investigation of pumps and pipe head losses to be calculated.

A laboratory based study was undertaken to assess the effect of increasing dry solids loading on the anaerobic digester mixing. A 15:1 scale physical model of the digesters was constructed by and various configurations of the existing pump recirculation mixing system were analysed. The model was used to assess the effect on feed dispersion, active volume and blend time with increasing dry solids.

As dry solids within the digester increase, the rheological properties of the sludge become more viscous, reducing the effectiveness of any given mixing system. The paper presents the effect of 2%, 4% and 6% digested solids levels, with a range of mixing energies and pump recirculation configurations on the system.

From this work, a computational fluid dynamics model was constructed. The results of testing within the scale physical model were then used to calibrate the CFD grid. Digested sludge behaves as a non Newtonian shear thinning fluid. Current state of the art for CFD

does not allow prediction of turbulent flow within a digester due to the non-Newtonian nature of the sludge. Calibration of the CFD model allowed a simpler laminar flow approach to be used to predicting the effect of modifications to the existing mixing system.

The paper goes on to explain why it is crucial to use site specific rheological data when conducting basic design for thicker sludge's and discusses what can go wrong if the properties of sludge's are not fully understood.

KEY WORDS

Digester Refurbishment, Rheology, Mixing, Pumping, CFD, Scale Modelling

INTRODUCTION

District Plant No. 1, is located in Fountain Valley, California and is currently treating an influent flow of 330,000m³/day. Following screening and primary settlement with Ferric chloride the primary sludge is pumped directly to the digesters at a dry solids concentration of about 5%. Surplus Activated Sludge (SAS) is thickened by Dissolved Air Floatation (DAF) to 4% and pumped directly to the digesters.

There are a total of 2 x 3000m³ and 8 x 8000m³ mesophilic anaerobic digesters (MAD) treating a total feed of approximately 2,600m³/day primary and SAS. Average digester temperature is 35°C and average digester retention time 27 days with a minimum retention of 18 days.

In preparation for an expected increase in solids loading to Plant No 1, an internal feasibility study was carried out to develop a future digestion process train for the next 20 years. The study concluded that in order to meet the minimum of 15 days retention time set by the US EPA for a Class B biosolid, either additional digesters would be required or digester feed would need to be thickened.

It was determined that the best economic solution was to modify the digester feed flow sheet, such that both primary and SAS would be co thickened by centrifuge prior to digestion. This centrifuge thickening will be phased and initially all of the primary sludge will be centrifuged together with 35% of the WAS. The remaining WAS will continue to be thickened using the existing DAF process. The percentage WAS going to centrifuge will increase incrementally to 65% and ultimately 100%.

This paper will report on the effect that increasing feed solids concentration will have on the existing mixing system.

RHEOLOGY STUDY

It has been known for many years that sewage sludges and digested sludges are non-Newtonian and that the viscosity of the fluid increases with dry solids concentration and that the increase is non linear, with 5% digested sludge being an order of magnitude more viscous than 2.5% digested sludge (Dawson, 2000). This has not been a major concern in the engineering of sewage sludge treatment plants until recently. The increasing trend to digest thicker sludges has led to the problem of pumping and mixing ever more difficult

fluids. This problem is amplified by the limitations of existing methods to predict sludge rheology, for 30 years the UK water industry has used WRc TR185 correlation (Frost), however developments in process technologies are now producing sludge such as polymer thickened SAS that are not covered by existing predictive equations.

Sludges often exhibit non-Newtonian behaviour where the shear rate and shear stress do not vary proportionally as for a Newtonian fluid such as water. Due to the different composition of sludges, their rheological behaviour varies between sludge types and between sites. The most common non-Newtonian behaviour found with sludges is known as 'shear-thinning', where viscosity reduces with increasing shear-rate and hence position within the tank or in proximity to the mixer. In addition, some sludges exhibit a yield stress which must be exceeded if the sludge is to flow at all.

The sludge viscosity is given by

$$\mu_a = \tau / \dot{\gamma} \quad \text{Equation 1}$$

where:

μ_a = apparent viscosity [Pa.s]
 $\dot{\gamma}$ = shear rate [1/s]
 τ = shear stress [Nm⁻²].

There are two mathematical models that are commonly used to characterise sludge rheology. These are the Herschel-Bulkley model (or Generalised Bingham Plastic):

$$\tau = \tau_y + k \dot{\gamma}^n \quad \text{Equation 2}$$

and the Power Law model:

$$\tau = k \dot{\gamma}^n \quad \text{Equation 3}$$

where:

τ_y = yield stress (N.m⁻²)
k = consistency coefficient (Pa.sⁿ)
n = consistency index (-).

As part of the initial engineering study samples were taken from Plant No 1 and analysed to determine values for τ_y , k & n. Samples were taken for the digested sludge and for the primary and SAS over 3 weeks to get as representative a sample as possible.

The primary and SAS were then blended to represent the various future conditions as the proportion of SAS being thickened by centrifuge increased. Three blends were analysed representing, all of the primary plus 35% SAS, all of the primary + 65% of SAS and finally all of both the primary and SAS. Furthermore the 3 blends were thickened so that sludge rheology could be measured at 3%, 6% and 9% dry solids to reflect the full range of sludge rheology that would be experienced.

The analysis for digested sludge was simpler as no blending was required. Solids concentrations of 2%, 4% and 6% were analysed to reflect the expected solids concentrations following digestion. This led to a large number of rheological sample to be analysed and due to the logistical difficulty of transporting all these samples to the UK the

majority of analysis was carried out in the USA. As a quality control exercise, a number of samples were split and sent to the UK for parallel analysis.

The measured rheology was used to calculate pump duties for primary and WAS transfer mains, digester heating recirculation pumps and for the digester mixing system. This approach allowed the team to accurately size pipework and optimise pump selection for the required duty, reducing the risk of either under design or the cost of over design.

MIXING STUDY

As part of the engineering study the current digester mixing system was evaluated using a combination of physical modelling and computational fluid dynamics (CFD) to assess the effect of increasing digester dry solids concentrations from 2% to 4% and 6% respectively.

It is becoming increasingly apparent that as digesters are pushed harder by reducing retention time and increasing dry solids loading traditional approaches to mixing digesters are beginning to experience difficulties, (Brade, Cumiskey, Harrison).

All 10 digesters at Plant No. 1, are mixed by a pumped jet recirculation system. Digested sludge is drawn from two points within the digester, one located centrally at the surface and the other at the base of the digester cone. Duty / assist pumps then recirculate the digested sludge through six nozzles located around the circumference of the digester at three levels. Two nozzles are located each at the bottom, middle and top of the digester side wall. All nozzles are orientated to produce a rotational flow in the same direction.

This mixing system has operated reliably for the digesters providing good mixing and dispersion of feed. However the experience indicated that increasing the digester dry solids will reduce the effectiveness of this mixing. To assess this effect a CFD model of the digester was constructed and the mixing system simulated at 2%, 4% and 6% dry solids concentration.

CFD is a mathematical modelling technique that solves fundamental flow equations for each cell in a mesh, 1.6 million in this instance, to form a construct of the equipment being investigated. The flow field within the equipment is simulated and flow patterns, velocities, residence time distributions, additive blend times, pressure drops and other relevant parameters can be determined. Many single and some multi-phase flows can be modelled.

As has already been stated, sewage sludges exhibit non-Newtonian flow behaviour that is shear rate is not directly proportional to shear stress, and hence viscosity varies with shear rate and position in the equipment. In some cases sludges also exhibit a yield stress, a threshold shear stress has to be surpassed before the sludge will flow at all.

Currently, there is no established, validated CFD model for transitional or turbulent non-Newtonian flows, however laminar non-Newtonian flows can be modelled with confidence. This means that digesters, blend tanks or other equipment where flows are turbulent in much of the vessel must be modelled by assuming the sludge has Newtonian flow behaviour. The assumption of Newtonian behaviour for the purposes of CFD modelling of sludge flows can lead to serious errors.

Turbulence is essentially a chaotic process involving a wide range of length scales and timescales. It is therefore generally impractical to attempt simulation of the details of the

turbulent fluctuations. Instead, turbulence is included in CFD simulations by devising mathematical descriptions of its principal effects on flow. In essence, turbulence tends to reduce gradients of velocity and concentrations, making the flow appear to be more viscous and enhancing mixing. The way in which these viscosity and mixing effects are represented mathematically defines the “turbulence model” and there are a great many of them available for selection by the CFD analyst. The principal function of the turbulence model is to calculate local values of turbulent viscosity. This quantity is then used in the calculation of turbulent mixing.

Previous work (Dawson 2004) has shown that a reasonable approximation of non-Newtonian flows in jet mixed digesters can be obtained using non-Newtonian laminar CFD models which are fully developed and commercially available. However, due to the lack of work carried out in this field with sewage sludges, it was imperative that the CFD be calibrated.

In order to calibrate the CFD model a physical scale model of one of the digesters was constructed, using a stimulant to represent the digested sludge; the mixing system was modelled at 2%, 4% and 6% solids concentrations.

A 3-D Acoustic Doppler Velocimeter (ADV) was used for velocity measurements below the liquid surface in the physical model. It has three heads that emit 16 MHz acoustic signals which converge on a measuring volume of 0.09 cm³. The ADV can measure velocities as low as a few mm/s. Velocities in the x, y and z directions are output to a data acquisition computer. Readings of the velocity within the physical model were compared with the velocities calculated using CFD allowing accurate validation. In addition, dye was added to the physical model to trace the active volumes and flow patterns effects of the feed, heating recirculation and jet mixing system.

Good agreement was found between the physical model and the CFD model at transitional and turbulent jet Reynolds numbers using 2% and transitional jet Reynolds numbers using 4% and 6% digested sludges concentrations. However, poor agreement was found with turbulent jets in 4% and 6% sludges, in these cases the non-Newtonian laminar model over predicted the velocities within the digester. Work continues at BHR to improve the accuracy of the CFD using various approaches including turbulence models under development for non-Newtonian applications. The thick sludge, turbulent jet validation results from this study underline the difficulty in applying CFD to real life situations and reinforce the need for caution in using this technique.

Although some reduction in digester mixing effectiveness was expected with increased sludge concentration the scale of the effect was surprising. At 2% dry solids concentration the mixing was shown to be very effective with over 90% of the digester volume being actively mixed and the feed blend time being calculated as 1.7 hours .

At 4% digested solids concentrations the active volume was calculated as 45% with a blend time of 8.5 hours and at 6% dry solids concentration the active volume was greatly reduced to less than 28% and blend time was in excess of a day.

The CFD model is not just a tool to evaluate the effect of increasing viscosity on digester mixing performance. Once constructed and calibrated, the CFD model was also used to test modifications to the digester mixing system to improve performance at elevated dry solids concentrations. The CFD model allowed the team to test different flow rates, jet

configurations and sizes prior to proposing any modifications to the mixing system at Plant No. 1.

CONCLUSIONS

An engineering study carried out for Plant No. 1, Orange County Sanitation District, has demonstrated the benefits of analysing site specific rheology in optimising pump selection, and assessing the effects of sludge dry solids concentrations on heating systems and mixing systems.

The study, carried out by Monsal and BHR using both physical and CFD models, has also demonstrated, (for this site), that doubling the dry solids concentration within the digester will more than halve the active volume provided by the digester mixing system. This reduction in active volume, if not addressed will more than double the affective volatile solids loading in the digester and halve the retention time, with risk of process failure.

Calibration of a laminar, non-Newtonian CFD model against velocity measurements in a scale model digester with a shear-thinning, yield stress fluid demonstrated good agreement at low sludge concentrations or laminar/transitional jet Reynolds numbers. The overestimation of velocities using CFD with high sludge concentrations and turbulent jets underlines the need to further develop CFD models for these applications through accurate validation using measurements on real systems.

The work has also highlighted the increasing technical complexity and hence specialist knowledge required to engineer solutions for sludge treatment. Existing methods for predicting pressure losses in pumped sludge mains are no longer useful or accurate and the current state of the art for CFD cannot be reliably used to predict highly non-Newtonian thick sludge mixing. However, the approach taken has shown CFD can be a useful technique to employ in assessing digester mixing when applied and validated correctly.

REFERENCES

BRADY, C. AND CUMISKEY, A. 'Mixing, Pasteurisation and Hydrolysis – technologies relevant to the new legislation'. Sludge: Planning for the New Regulations, CIWEM, Derby, UK. 13 Mar. 2002.

CUMISKEY, A., HARRISON, D., DAWSON, M. AND JOLLY, M. 'Effective Design of Gas Mixing Systems for Thick Sludge Digestion – Two Case Studies' 6th European Biosolids and Organic Residuals Conference, Aqua-Enviro, Wakefield, UK. 11-14 Nov. 2001.

DAWSON, M. K., LEEFE, S.E., HARRISON, D., CUMISKEY A. 'Modelling of Flows in Sludge Mixing Equipment - The Use of Physical Models and Computational Fluid Dynamics (CFD) as Design and Evaluation Tools'. Sludge 12, Surrey University, Surrey, UK. 14th January 2004.

DAWSON, M., CHRISTODOULIDES, J., FAWCETT, N. AND BRADY, C. 'A Comparison of Mixing Systems in a Model Anaerobic Digester'. 5th European Biosolids and Organic Residuals Conference, Aqua-Enviro, Wakefield, UK. 19-20 Nov. 2000.

HARRISON, D., MAY, P., JONES, B. 'Enhancing Your Assets - Refurbishment of The Largest Digestion Facility in the UK' 2003 Residuals and Biosolids, Baltimore, Maryland USA. February 19-22, 2003.

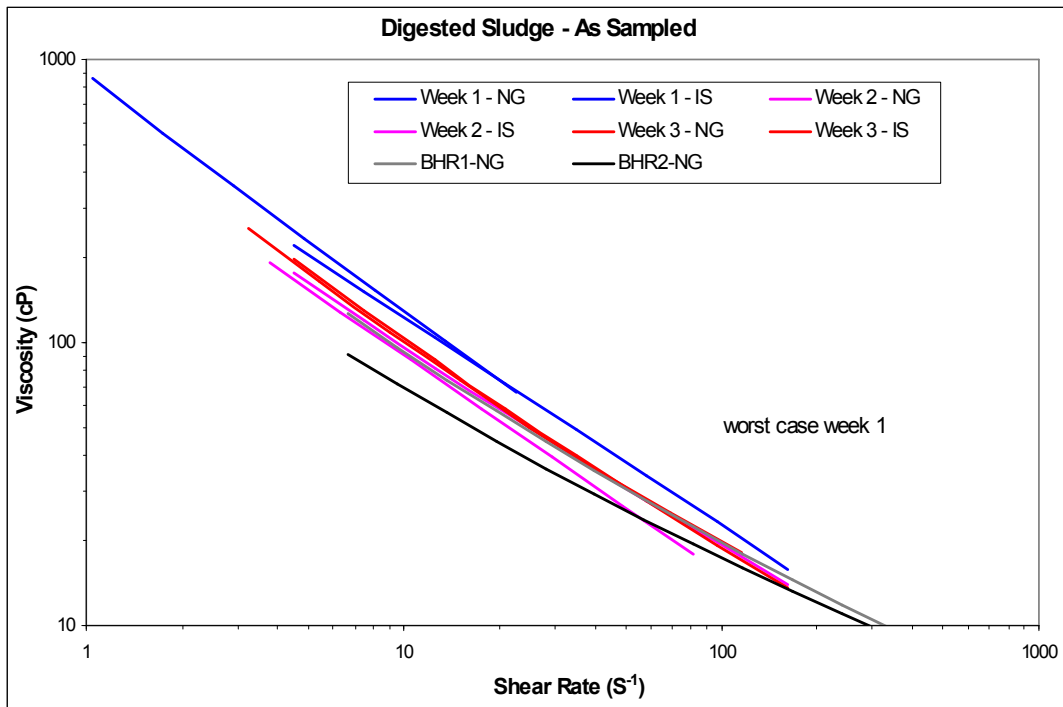
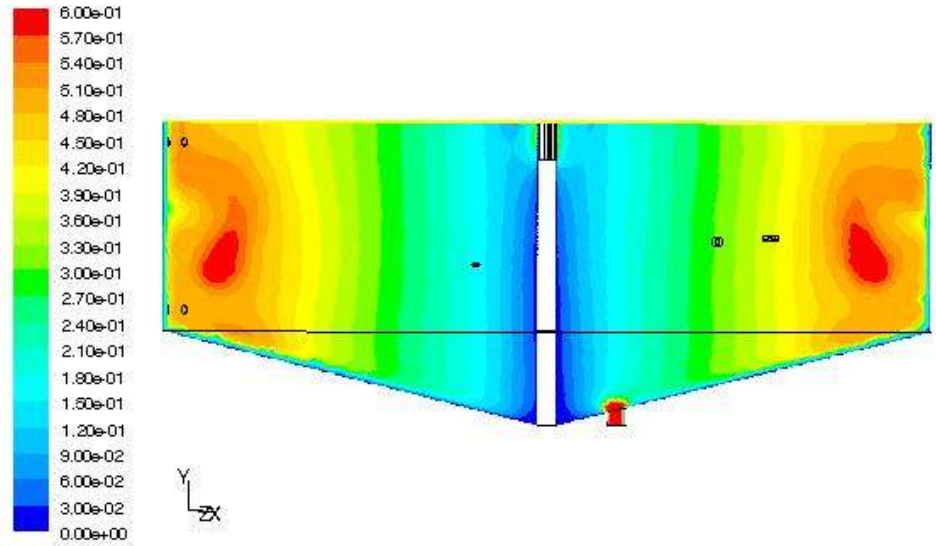


Figure 1 – Example of Measured Rheology

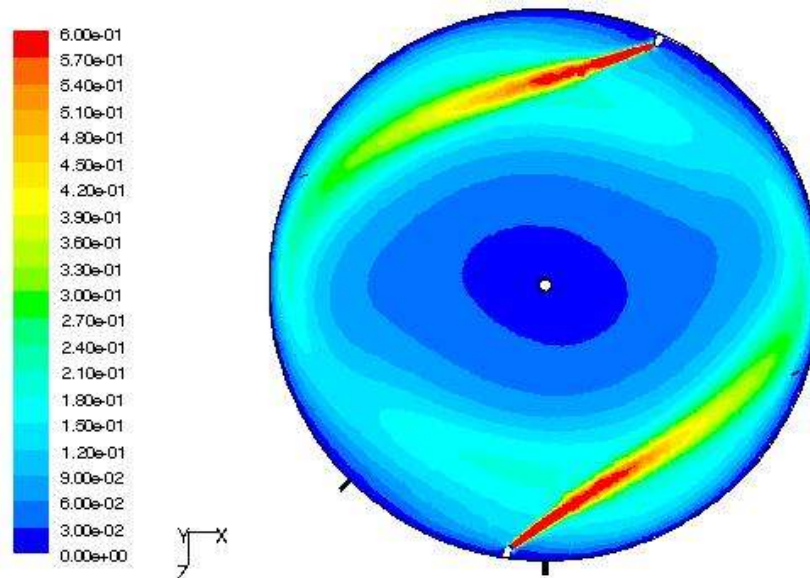


Figure 2 – Physical Model



Contours of Velocity Magnitude (m/s) Aug 25, 2005
FLUENT 6.2 (3d, segregated, lam)

Figure 3 – Vertical Section Through CFD Model at 2% Dry Solids



Contours of Velocity Magnitude (m/s) Sep 15, 2005
FLUENT 6.2 (3d, segregated, lam)

Figure 4 – Horizontal Section Through CFD Model at 4% Dry Solids

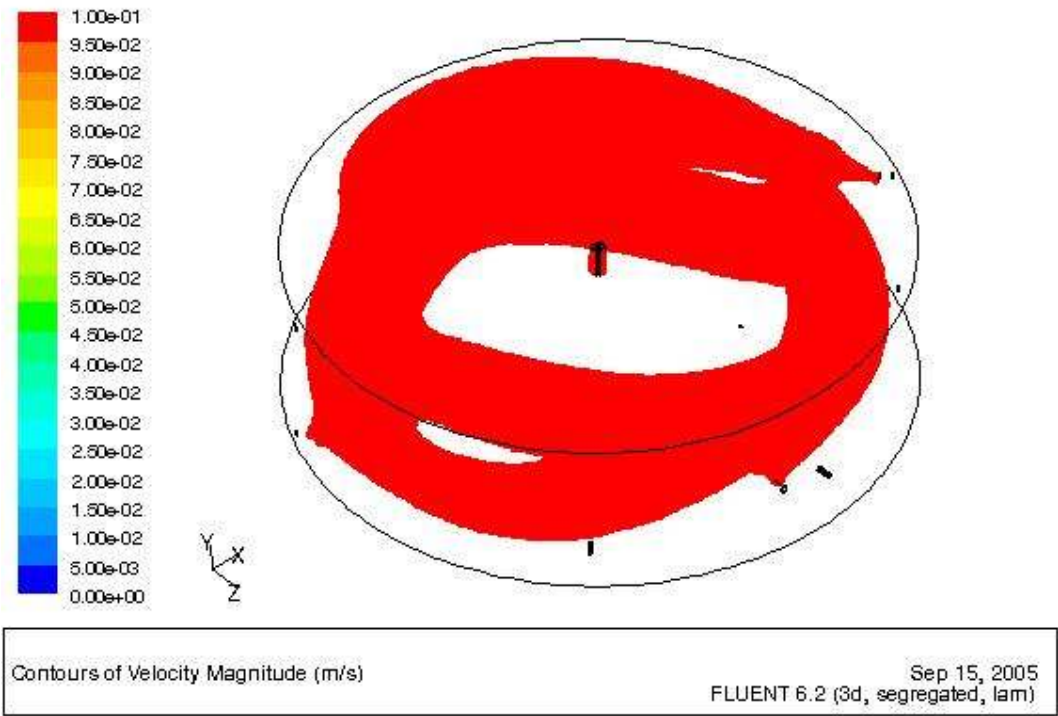


Figure 5 – Active Volume of CFD Model at 6% Dry Solids