

Using CFD Modelling as an Evaluation Tool in the Design of Jet Mixers

Vilas Sarsani¹ and Amit Bathija²

¹Wastewater Engineer, Mixing Systems, Inc., Dayton, OH. Email: vilas@mixing.com

²Vice President, Mixing Systems, Inc., Dayton, OH. Email: amit@mixing.com

ABSTRACT

Mixing is among the most important applications used by wastewater treatment facilities to achieve blending, solids suspension, and prevention of sludge accumulation. Jet mixing is a widely adopted mixing technology used in equalization tanks, anoxic basins, sludge storage tanks, and digester tanks. With increasing energy costs in recent years, it is very important to study and optimize the mixer design to improve mixing performance and minimize energy expense. This paper compares the mixing performance of two commonly used jet mixing technologies in wastewater tanks using a computational fluid dynamics (CFD) tool. Accordingly, three-dimensional CFD models are developed for these mixers and solved based on the Reynolds-averaged Navier-Stokes equations using the standard $k - \epsilon$ turbulence model. The simulation results are used to determine the mixing flow patterns, length of jet plume, dead volume, and mixing velocities achieved for each mixer. The results are also interpreted to determine the effect of total solids concentration on the mixing performance.

INTRODUCTION

Mixing is one of the key unit processes in wastewater treatment. It is used in different stages of wastewater treatment to achieve physical, chemical and biological uniformity, solids suspension and to prevent sludge accumulation. The mixing of large wastewater tanks can be achieved by using mechanical mixing, gas mixing, and recirculated pump mixing.

Within the category of recirculated pump mixing, there are two common but distinct methods of in-tank mixing. Traditional Jet Mixers are a commonly used pump mixing technology in wastewater treatment plants. In this jet mixing technology, the recirculated liquid is pumped through a double-nozzle configuration to achieve high-velocity plumes that entrain surrounding fluid and create a circulatory mixing pattern.

A Hydraulic Sludge Mixer is another type of pump mixing technology, and commonly used in sludge and digester applications. In Hydraulic sludge mixing technology, the recirculated liquid is pumped through a single nozzle at a high pressure/head and velocity in order to create a unique rotational pattern. Although both mixers work on the same operating principle, hydraulic sludge mixers use relatively lower flow, higher pressure/head, and fewer in-tank nozzles as compared to traditional jet mixers.

As mixing is an energy-intensive process, it is important to select the most-efficient mixing system to maximize performance and reduce plant operating costs. During the design phase, it is difficult to assess the effectiveness of nozzle placement without performing testing, which can be a very time-consuming and expensive process. With rapid development in technology, CFD modeling has proven to be a better and cheaper alternative method of evaluating mixing performance. Numerous CFD studies have been carried out and published over the years showing that the simulation results are in correlation with experimental results (Furman and Stegowski 2011; Rahimi and Parvareh 2005,2007; Wasewar and Sarathi 2008).

The primary objective of this paper is to compare the mixing characteristics of a wastewater tank equipped with a directional jet mixer and hydraulic sludge mixer. Resultant tank mixing has been compared using velocity analysis, vector plots, streamlines and particle trajectories. The paper also studies the effect of total solids concentration on the mixing performance.

CFD MODEL FORMULATION

The CFD simulation to evaluate the mixing in the wastewater tanks is accomplished using Finite Volume Method (FVM), which is most appropriate for complex geometries and mass conversation equations. The simulation is carried out by breaking the geometry into millions of unstructured volume elements and discretizing and solving the continuity and momentum equation on each control volume using the 2nd order discretization equations. The Navier-stokes equation used for incompressible flows (Liu and Zhang 2020) is

$$\text{Continuity Equation: } \nabla \cdot u = 0$$

$$\text{Momentum Equation: } \frac{\partial u}{\partial t} + u \cdot \nabla u = \frac{-1}{\rho} \nabla p + \nu \nabla^2 u$$

where

u = Velocity vector,

p = Pressure,

ρ = Density,

t = time, and

ν = Kinematic viscosity

In this paper, ANSYS version 2020 R2 is the commercial CFD package used for simulation. A 3D Computational Model is formulated to evaluate the mixing achieved inside a circular tank by the directional jet mixer and hydraulic sludge mixer. The various steps involved in formulation of this CFD model are described below:

System Geometry

All the 3D model geometry utilized in this paper was modeled using AutoCAD by Autodesk. In the present analysis, a circular tank with a diameter of 23.7 m (78 ft) and sidewall height of 9.1 m (30 ft) is studied. The maximum liquid level in this tank is designed for 8.5 m (28 ft). The maximum simulated liquid volume will be 3785 m³ (1 Million U.S Gallons). Two mixers which utilize a same 75 HP recirculation pump is simulated for the comparison. The geometry of the mixers used is as follows:

Directional Jet Mixer

One Directional Jet Mixer model DM2JM48 manufactured by Mixing Systems, Inc, is simulated inside the tank. The Directional Jet Mixer is a traditional jet mixer with all jet nozzles pointing in one direction. The simulated mixer consists of 48 Jet nozzles and is designed to create a circular mixing pattern. The above-mentioned jet mixer is powered using a 75 HP recirculation pump at a duty point of 2180 m³/h at 6.7-meter TDH. Fig 1(a) presents an illustration of the three-dimensional geometric model prepared for this analysis.

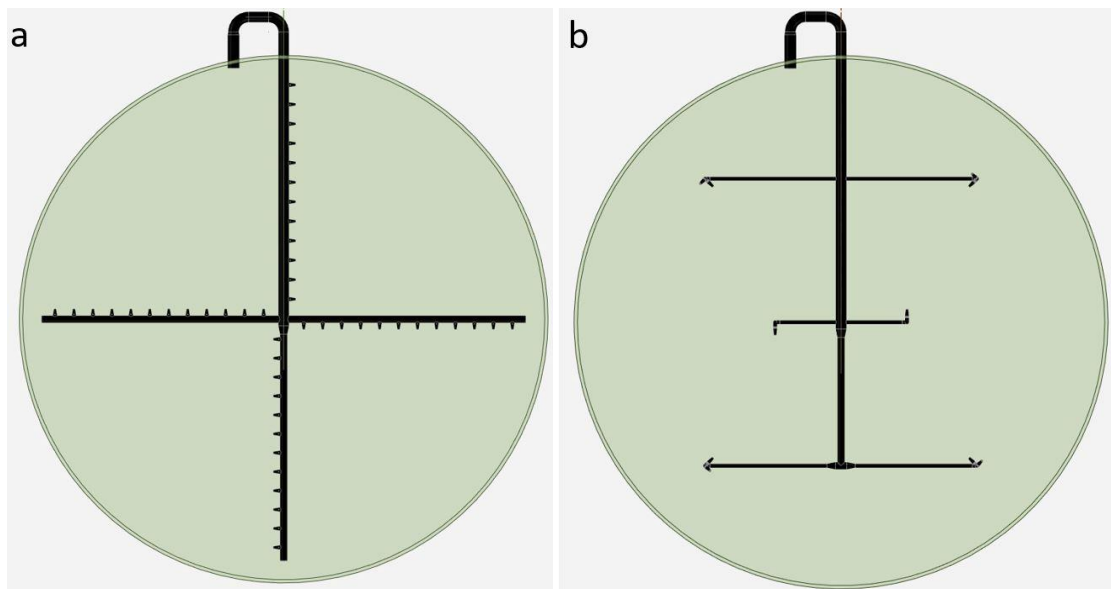


Figure 1. Plan View of Tank Geometry with (a) Directional Jet mixer (b) Hydraulic sludge mixer

Hydraulic Sludge Mixer

One Sludge Mixing system consisting of 4 dual-nozzle assemblies and 2 single-nozzle assemblies, manufactured by Mixing Systems, Inc, is simulated inside the tank. The dual-nozzle assemblies are located at a diameter of 75% of tank diameter and 2 single floor nozzles are located at a diameter of 30% of tank diameter to create a circular rotational pattern. Fig 1(b) presents an

illustration of the three-dimensional geometric model prepared for this analysis. The above-mentioned jet mixer is powered using a 75 HP recirculation pump at a duty point of 1136 m³/h at 12.2-meter TDH.

Meshing

Meshing is a critical step in the success of a CFD Model. The 3D Geometry is divided into a number of unstructured volume elements using the ANSYS Meshing tool. Various mesh settings such as inflation layers and maximum size limits were introduced to improve the mesh quality.

The geometry of the tank with the directional mixer is divided into 22,302,436 tetrahedral and hexahedral mesh elements while the geometry of tank with the hydraulic sludge mixer is divided into 18,746,852 tetrahedral and hexahedral mesh elements. The generated computational mesh is shown in Fig 2 (a) and Fig 2 (b).



Figure 2. Computational Mesh of 3D Model with (a) Directional Jet Mixer (b) Hydraulic Sludge Mixer

Solver Setup

A 3D, implicit, pressure-based, single-phase, steady-state solver algorithm was utilized for predicting the velocity contours and vectors at different horizontal and vertical levels across the tank. The SIMPLE pressure-velocity coupling method along with second-order discretization equations were used in this analysis. The under-relaxation factors for pressure, density, body forces, momentum and turbulent kinetic energy are set to 0.5, 1, 1, 0.5 and 0.5 respectively.

Fluid Properties

For simplicity, the fluid used in this paper is assumed to be incompressible and isothermal. The temperature of the fluid is also assumed to be constant at 35°C. The rheology of wastewater and sludge is complex, and it varies depending on the treatment process and the influent. For this

analysis, the manure/sludge is assumed to exhibit non-Newtonian pseudo-plastic fluid behavior as shown in the below equations (Eshtiaghi et al. 2013).

$$\eta = k \dot{\gamma}^{n-1}$$

$$\rho = 0.0367 TS^3 - 2.38 TS^2 + 14.6 TS + 1000$$

where

η = viscosity,

k = consistency coefficient,

$\dot{\gamma}$ = shear rate,

n = power-law index, and

TS = weight percentage of total solids in the sludge/manure.

The experimental data obtained for manure from Achkari-Begdouri (Achkari-Begdouri and Goodrich 1992) is shown in Table 1 and used in this analysis. In this paper, analysis is conducted for pure water ($TS=0\%$) which acts as a Newtonian Fluid and for other non-Newtonian sludge flows with concentrations of 2.5%, 5.4 %, 7.5%, 9.1% and 12.1% to study the effects of total solids on mixing performance.

Table 1. Fluid Properties of Liquid Sludge (Wu and Chen, 2007)

Total Solids, TS (%)	Density, ρ (kg/m ³)	Consistency Factor, k (Pa.s ⁿ)	Power Factor, n	Minimum Viscosity, η_{min} (Pa.s)	Maximum Viscosity, η_{max} (Pa.s)
0	998	0.001	1	-	-
2.5	1000.36	0.042	0.71	0.006	0.008
5.4	1000.78	0.192	0.562	0.01	0.03
7.5	1001.00	0.525	0.533	0.03	0.17
9.1	1001.31	1.052	0.467	0.07	0.29
12.1	1001.73	5.885	0.367	0.25	2.93

Turbulence Conditions

The standard $k - \varepsilon$ turbulence model based of Reynolds Average Navier-Stokes (RANS) equations is used to model the turbulent flows in the model (Wilcox 1998). The standard transport equation and constants are used for turbulence kinetic energy and the specific dissipation rate. (ANSYS 2020).

Initial and Boundary Conditions

The initial velocity and pressure of the fluid inside the tank is set at zero. The duty points of the recirculation pump used for both mixers are modeled using the boundary conditions. The solid

walls of the tank and mixers are modeled as a no-slip boundary condition with a roughness constant of 0.5 and standard wall functions are used for near-wall treatment.

Convergence Criteria

The convergence criteria for the analysis is set based on the RMS residuals of X, Y & Z momentum and mass balance to be less than 10^{-4} .

RESULTS AND DISCUSSIONS

Comparison of Mixing Systems

All simulations are run until the steady-state condition is reached and convergence criteria is met. For the purposes of comparison of both mixing systems, the simulation results from the sludge/manure with 5.4% TS concentration are utilized.

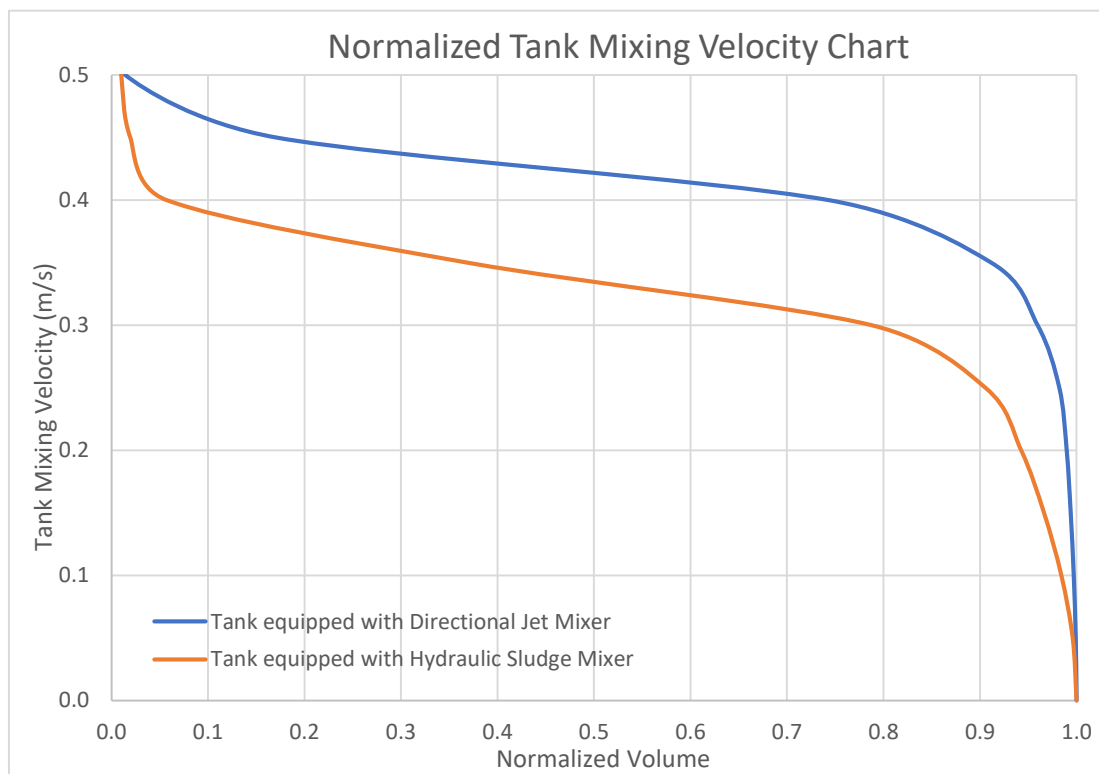


Figure 3. Normalized Mixing Velocity Chart for both systems with 5.4% Total Solids Concentration.

Mixing Velocity Analysis

In order to uniformly mix the tank and prevent the sludge settling, the mixing velocity inside the tank should be much greater than the settling velocity of sludge/particles.

The average mixing velocity of complete fluid volume inside the tank with the directional jet mixer is found to be around 0.42 m/s. The average mixing velocity of complete fluid volume inside the tank with the hydraulic sludge mixer is found to be around 0.33 m/s. Figure 3 shows the normalized mixing velocity inside the tank. This shows that the at-least 90% of the tank volume will be greater than 0.35 m/s for the jet mixer and it will be greater than 0.25 m/s for the hydraulic sludge mixer.

Analysis of Velocity Contours

Figure 4 shows the velocity contour plot along two imaginary horizontal planes placed at the upper surface/maximum liquid level and 0.5 m from the tank bottom. Figure 5 shows the velocity contour plot along two imaginary vertical planes placed across the tank. This shows that both systems are able to mix the tank with effective mixing velocities but the velocity distribution in the center of the tank is better in the case of the directional jet mixer.

Analysis of Vector Plots

Figure 6 shows the velocity vector plots of two imaginary horizontal planes placed at the upper surface/maximum liquid level and at the tank bottom. This shows that the uniform stirring motion will be created inside the tank with good mixing velocities in both cases.

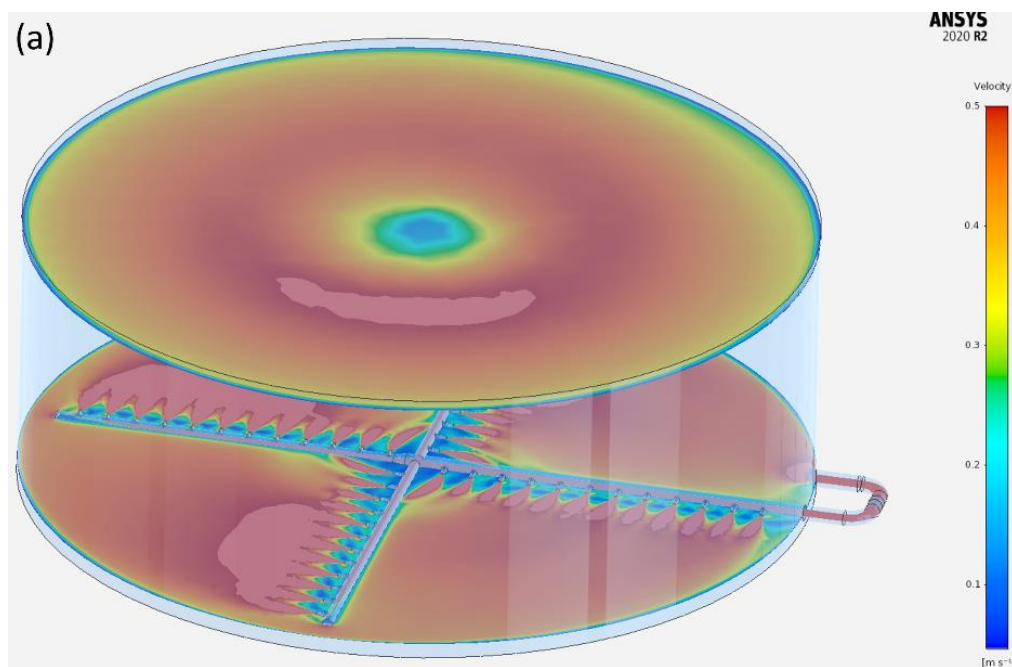


Figure 4 (a). Velocity Contour Plots of a two horizontal planes at upper surface and at 0.5 m above the floor inside a Tank (with 5.4% TS) with Jet Mixer

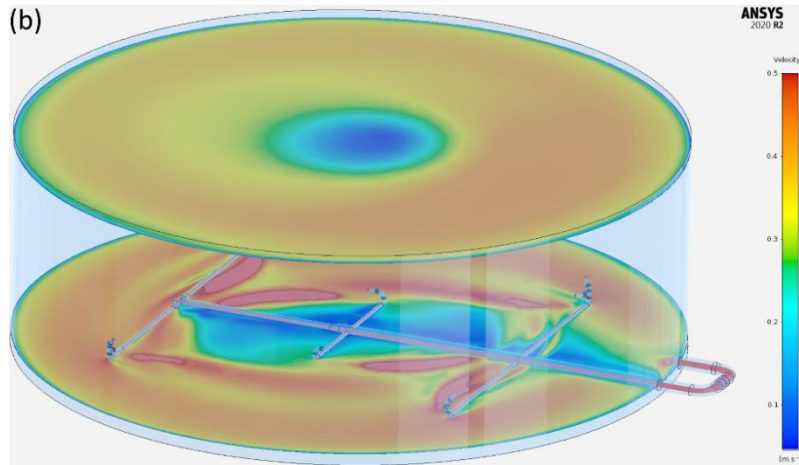


Figure 4 (b). Velocity Contour Plots of a two horizontal planes at upper surface and at 0.5 m above the floor inside a Tank (with 5.4% TS) with Sludge Mixer

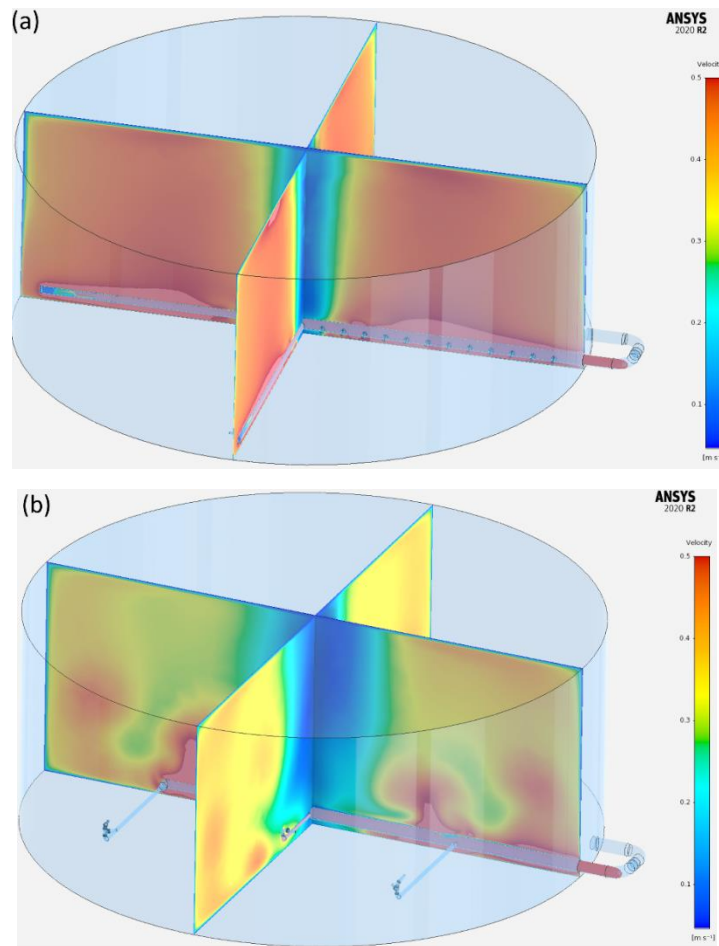


Figure 5. Velocity Contour Plots of a vertical planes across the center of tank (with 5.4% TS) inside a (a) Tank with Jet Mixer (b) Tank with Sludge Mixer

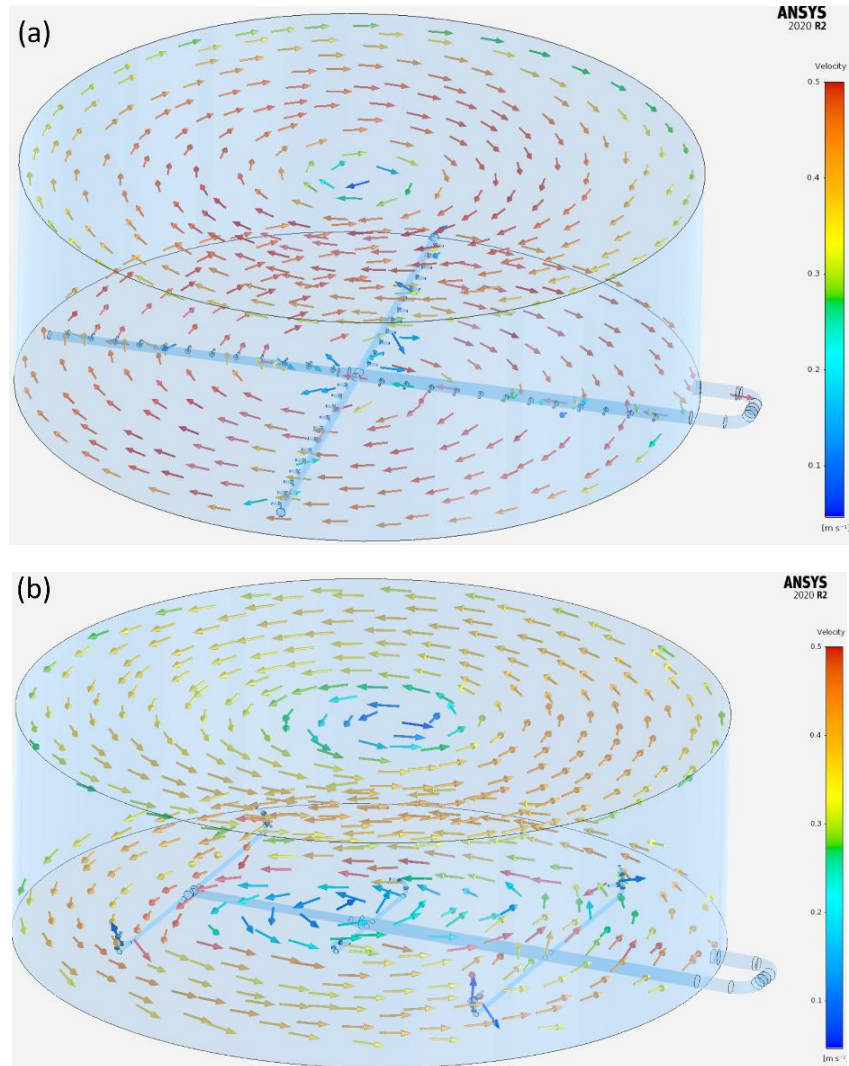


Figure 6. Velocity Vector Plots of a horizontal planes at upper surface and tank bottom inside a tank (with 5.4% TS) with (a) Jet Mixer (b) Sludge Mixer

Path line Plots

Figure 7 shows the 90-minute particle path-line plot of a single particle released from the tank bottom. This shows that the particle will be resuspended and mixed along the vertical axis of the tank with a stirring motion and effective mixing velocity in both cases.

Length of Jet Plumes

Figure 8 shows the iso-surface plot of all 0.65 m/s velocity surfaces inside the tank. This shows that high-velocity jet plumes are released out of the nozzles resulting in mixing inside the tank. It also shows that the jet plumes released out of the hydraulic sludge mixer is longer than directional jet mixer because of its higher pressure/operating head.

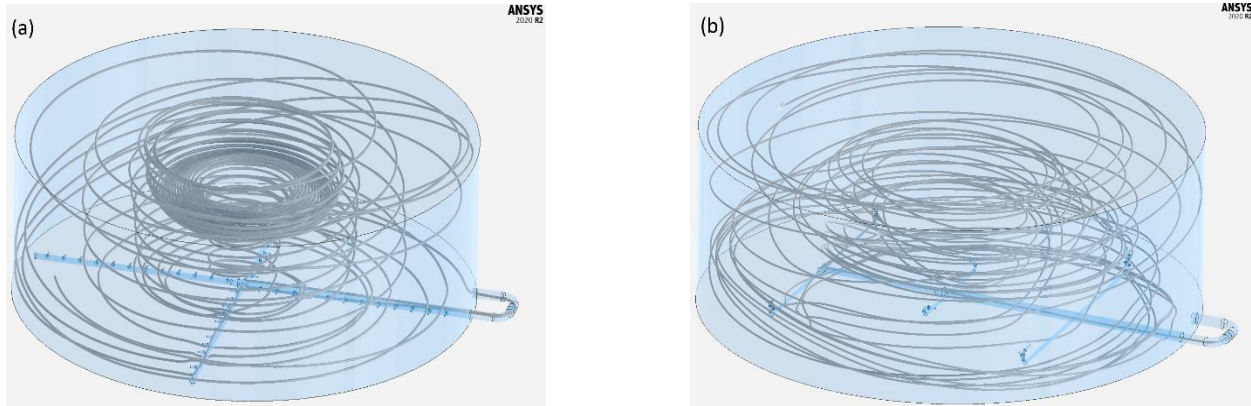


Figure 7. 90-minute Particle Path line Plot of a single particle released from the tank bottom inside a tank (with 5.4% TS) with (a) Jet Mixer (b) Sludge Mixer

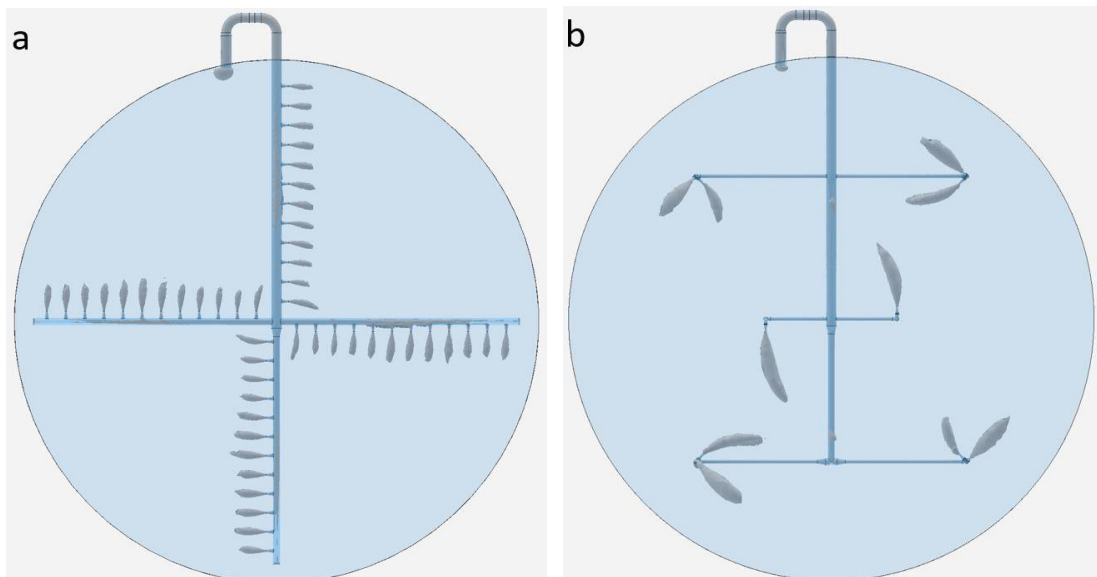


Figure 8. Iso-surface Plot of all 0.65 m/sec surfaces inside a tank (with 5.4% TS) showing High-Velocity Jet plumes created by (a) Jet Mixer (b) Sludge Mixer

Effect of Total Solids on Mixing Performance

Figure 9 and Figure 10 shows the normalized mixing velocity charts of tanks with the jet mixer and hydraulic sludge mixer with varying solids concentration. Table 2 shows the average mixing velocity inside the tank with varying solids concentration. This shows that the mixing performance and velocity are reduced as total solids concentration increases. This simulation model does not factor in the increase in the pump power that may be necessary to offset the effects of higher viscosity and specific gravity of sludge.

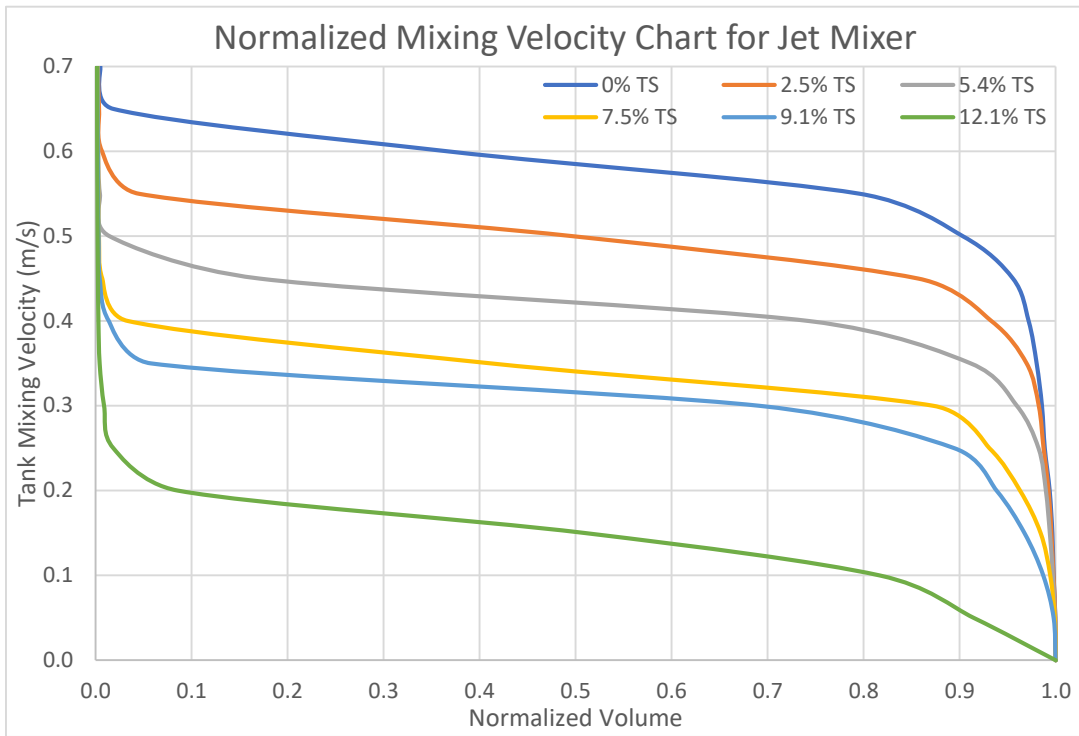


Figure 9. Normalized Tank Mixing Velocity Chart for Directional Jet Mixer with varying solids concentration.

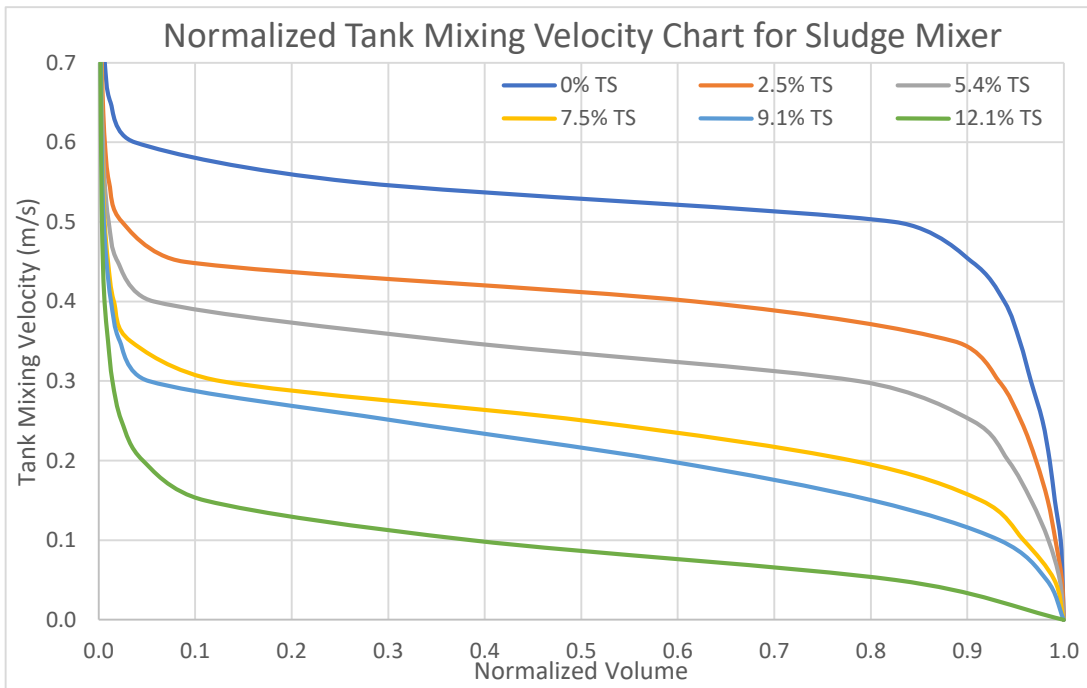


Figure 10. Normalized Tank Mixing Velocity Chart for Hydraulic Sludge Mixer with varying solids concentration.

Downloaded from ascelibrary.org by Vilas Sarsani on 06/14/21. Copyright ASCE. For personal use only; all rights reserved.

Table 2. Average Mixing Velocity with varying solids concentration

Total Solids, TS (%)	Average Velocity inside the Tank with Jet Mixer (m/s)	Average Velocity inside the Tank with Sludge Mixer (m/s)
0	0.58	0.52
2.5	0.49	0.4
5.4	0.42	0.33
7.5	0.34	0.22
9.1	0.31	0.2
12.1	0.14	0.08

CONCLUSION

This paper compares the mixing performances of a commercially available directional jet mixer and hydraulic sludge mixer. An advanced 3D Computational Fluid Dynamics (CFD) model is formulated based on wastewater properties and utilized as an evaluation tool. Based on the simulation results, the following conclusions are made:

1. Based on the normalized velocity plots, it is observed that both mixing systems are capable of effectively mixing the tank with good mixing velocities.
2. Based on the velocity contours, it is observed that the directional jet mixer is more effective in velocity distribution due to the presence of more jet nozzles.
3. Based on the velocity vector and path line plots, it is observed that both mixing systems are capable of resuspending the solids and create a stirring mixing pattern inside the tank.
4. Based on the normalized mixing chart (Fig 9 and Fig 10), the mixing velocities achieved inside the tank are high. So, while mixing with the lower concentrations of sludge, the pump speed can be reduced to save the energy costs.
5. Based on the normalized mixing chart for varying solids concentration (Fig 9 and Fig 10), although both mixing systems are capable of mixing 12.1% TS sludge, there might be some cases of settling. Additional measures such as adding more nozzles and increasing pump pressure can be done to improve the performance.
6. Based on the above results, the overall mixing performance of directional jet mixer is better than hydraulic sludge mixer for the higher TS concentrations due to the presence of more jet nozzles.
7. The above formulated CFD model can be further applied to different tank geometries and other jet mixers to improve the mixing performance and adapt the design accordingly.

REFERENCES

- Achkari Begdouri, A., and Goodrich, P. R. (1992). "Rheological properties of moroccan dairy cattle manure." *Bioresource Technology*, 40(2), 149-156.

ANSYS-Fluent Inc. *ANSYS Fluent 2020 R2 User Manual*.

Eshtiaghi, N., Markis, F., Yap, S. D., Baudez, J. C., and Slatter, P. (2013). "Rheological characterisation of municipal sludge: a review." *Water Research.*, 47(15), 5493-5510.

Furman, L., and Stegowski, Z. (2011). "CFD models of jet mixing and their validation by tracer experiments." *Chemical Engineering and Processing: Process Intensification*, 50(3), 300–304.

Liu, X., and Zhang, J. (2020). *Computational Fluid Dynamics: Applications in Water, Wastewater, and Stormwater Treatment*. American Society of Civil Engineers, Reston, Virginia, 28.

Wu, B., and Chen, S. (2007). "CFD simulation of non-Newtonian fluid flow in anaerobic digesters." *Biotechnology and Bioengineering*, 99(3), 700–711.

Rahimi, M., and Parvareh, A. (2005). "Experimental and CFD investigation on mixing by a jet in a semi-industrial stirred tank". *Chemical Engineering Journal*, 115(1-2), 85-92.

Rahimi, M., and Parvareh, A. (2007). "CFD study on mixing by coupled jet-impeller mixers in a large crude oil storage tank". *Computers & Chemical Engineering*, 31(7), 737-744.

Wasewar, K., and Sarathi, J. (2008). "CFD Modelling and Simulation of Jet Mixed Tanks". *Engineering Applications of Computational Fluid Mechanics*, 2(2), 155-171.

Wu, B., and Chen, S. (2007). "CFD simulation of non-Newtonian fluid flow in anaerobic digesters." *Biotechnology and Bioengineering*, 99(3), 700–711.