CFD modeling of jet mixing to study the mixing performance and sludge Prevention in crude oil tanks

Kavitha Nagarasampatti Palani [a*], Naveen kumar Saravanakumar[a]

[a]Department of Chemical Engineering, Sri Venkateswara College of Engineering, Sriperumbudur, Kancheepuram, India.

Correspondence: Assistant Professor, Department of Chemical Engineering, Sri Venkateswara College of Engineering, Sriperumbudur, Kancheepuram, India.

Email id: npkavitha@svce.ac.in, Ph: 9789950621

Abstract

Computational fluid dynamics (CFD) model was developed using FLUENT 18.1to study the hydrodynamics in jet mixer for the prevention of sludge in a large storage tank containing crude oil. Eulerian approach was used to predict the flow behavior of sludge and k-e model also used to analyze the turbulence in the storage tank. The parameters such as velocity, time, nozzle head in single and time, nozzle head angle position in multi jet were analyzed. The velocity at $30ms^{-1}$, the mixing pattern was appreciably good and the profile obtained was fully developed at the bottom. The profile developed in multi jet with various angle position also verified. RANS formulation for numerical analysis also verified in which the amount at all points were not exceeding 300 which was appreciable in modeling.

Keywords: Jet mixing, Eulerian model, Fluent, k-e model, Hydrodynamics

Introduction

Mixing is the most general unit operations available in chemical industries. Jet mixers are nowadays used instead of impellers because of its high velocity circulation without any moving parts, less energy and maintenance cost, occurrence of less dead zones in larger tanks, higher turbulence and shear rate, vortex motion, contact surface reduction and less in mixing time (Lotfi al., 2017, Toru et al., 2012). Jet mixers are nowadays used in heavy oil storage tanks to avoid the sludge formation.In jet mixers, circulatory pattern was formed leads to cause the oil and sludge movement from the bottom to the middle resulting in mixing of two fluids (Paul et al., 2004). There are so many parameters influencing the mixing in jet mixers like mixing time, fluid density, diameter of the tank, diameter, position, nozzle diameter, geometry &velocity of jet and Reynolds number (LotfiNeyestanaket al., 2017, Raja et al., 2007).

Researchers have proposed various methods with various parameters to study the multiphase flow in storage tanks. Numerous studies have been conducted to correlate the mixing time, diameterof tank, jet velocity. On contrary, mixing time was dependant on lower Reynolds number and independent of higher Reynolds number (Fosset, 1951, Fox and Gex, 1956). And also, mixing time was reduced in stronger jet mounted in entry of the system (Coldrey et al.,1978). Meanwhile, dissipitation rate of energy governed the mixing time in a expanse long away from the nozzle (Grenville and Tilton, 1996). The orientation of the jet influenced the flow pattern and mixing time was dependant on diameter of nozzle and angle of inclination (Patwardhan, 2002). The mixing time is an essential parameter to predict accurately for an excellent design and process operation of large storage tanks. But, it needs some detailed information about flow properties and hydrodynamics. In such cases, flow velocity, jet parameters and mixing optimizations are hardly to attain in 2D and 3D.patterns.

Computation Fluid Dynamics (CFD) is a forceful tool to predict all the mentioned contrary parameters precisely for an effective design of storage tanks. And also, it is a supplement to the lab scale data with an extensive methods to get high resolution in mixing time and fluctuations in turbulent and laminar flow. So, our objective is to study the two-phase system and modeling of submerged turbulent jet mixing in two phase with two dimensional (2D) and three dimensional (3D) system with single and four nozzles as shown in figure 1 by CFD Eulerian approach using FLUENT 18.1 to predict the mixing time in crude oil storage tank.

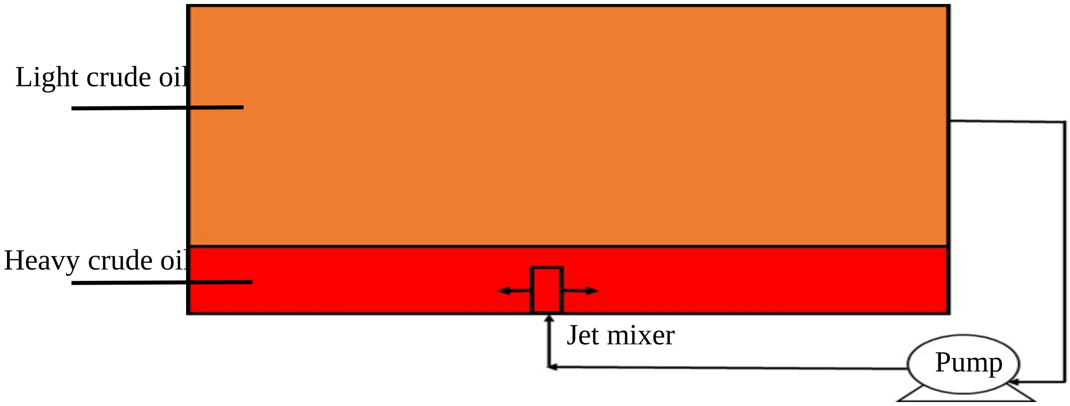

Figure 1. Schematic representation of crude oil storage tank

2. CFD Modeling

The geometry and mesh generated in ANSYS is shown in figure 2. The developed system is an oil storage tank with dimensions of 114 m × 7 m which is floated as a ceiling and contains sludge with one meter height from tank bottom and the rest is crude oil. The nozzle, placed at 80 cm from tank bottom with the diameter of 100 mm, and receives crude oil from upper layer and jets it into the sludge (LoftiNeyestanak et al., 2017). The assumptions made for modeling was incompressible flow, 2-dimensional flow, uniform dispersion and isothermalconditions.

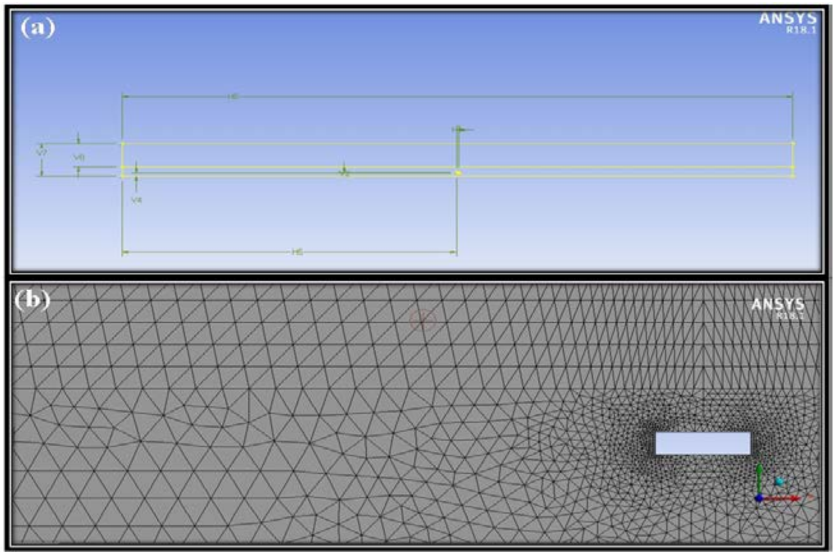

Figure 2. (a) Geometry and (b) Mesh generated by ANSYS

The jet mixing tank dimensions of 80 m × 7 m with four jet nozzles placed at the circumference of the base of the cylindrical tank which is placed equidistant from each other. Bottom part of the tank contains one-meter sludge and upper part is the six-meter crude oil where jet is placed inside the tank wall. The nozzle is placed at various height from tank bottom which has a diameter of 100 mm, and receives crude oil from upper layer and jets it into the sludge. The four nozzles were placed at opposite to each other. The jet position and crude oil properties are shown in the table 1 and 2.

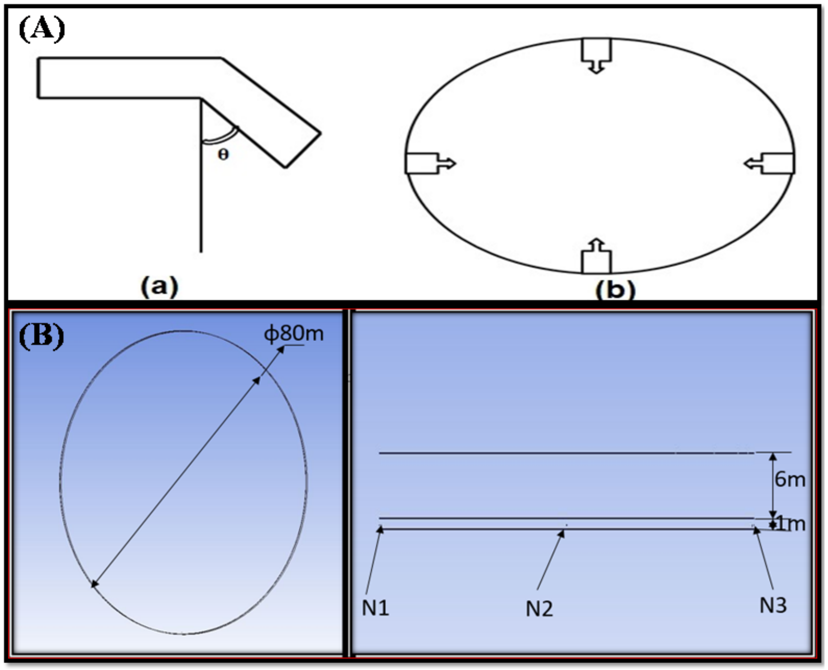

Figure 3. (A) Schematic representation of Jet angle with (a) 45^0 downwards (b) Elevation of four jet nozzle and (B) Geometry of the jet mixer in storage tank

Table 1.Position of jet nozzle at different height and jet angle

Nozzle height (m)	Angle		
0	0°	0° & 45°	45°
0.3			
0.5			
0.7			

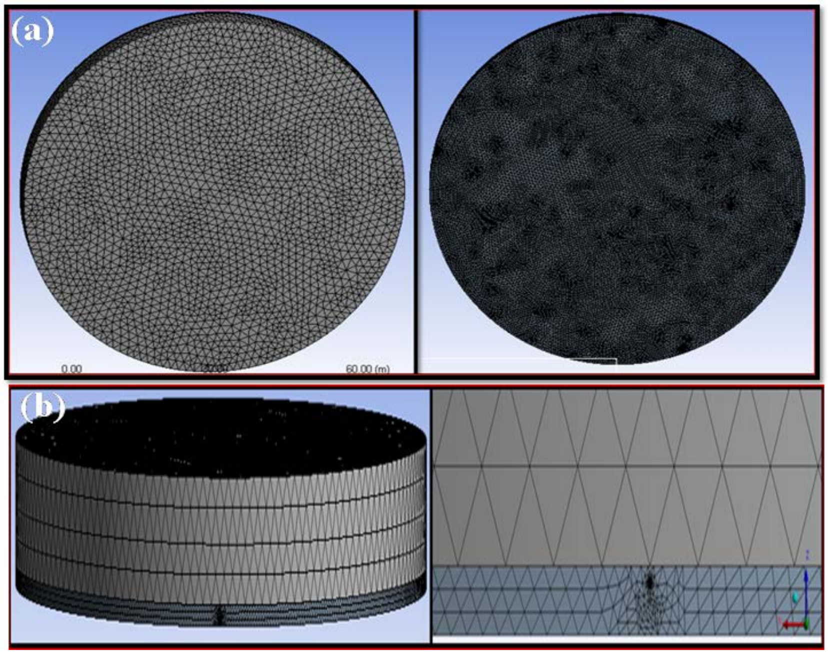

Figure 4. (a) Top and bottom part mesh of the jet mixer and (b) Side view of the jet mixer

It is evident from figure 4 that irregular meshing is given for the sludge part and regular meshing is given for the oil part.The purpose of meshing is to separate the domain into a convenient number location for obtaining accurate results.Grid or Mesh is defined as smaller shapes formed after discretization of geometric domain. Mesh or Grid can be in 3- dimension and 2-dimension.The mesh of crude oil part is a regular triangle type and which have the lower geometrical flexibility but faster, and the mesh of nozzle and sludge part is an irregular triangle type and which have the higher geometrical flexibility but the methods were slower.

The equations chosen for determining the flow pattern in multiphase flow are

Continuity equation for phase q

$$\frac{d}{dt}\alpha_q\rho_q+\nabla\alpha_q\rho_q\vec{v}_{q}=\sum(\dot{m}_{pq}-\dot{m}_{qp})+S_q \qquad 1$$

where, the$\dot{m}_{pq}$ represents the mass transport phase p to q, $\vec{v}$ represents the phase velocity in ms^{-1}, α_q represents the volume of fluid in m^3, ρ_q is the phase density kg m^{-3} and S_q is the mass source.

Momentum equation for phase q

$$\frac{d}{dt}(\alpha_q\rho_q\vec{v}_q)+\nabla(\alpha_q\rho_q\vec{v}_q\vec{v}_q)=-\alpha_q\nabla P+\nabla\bar{\bar{\tau}}_q+\alpha_q\rho_q g+\sum_{p=1}^{n}(R_{pq}+\dot{m}_{pq}\vec{v}_{pq}-\dot{m}_{qp}\vec{v}_{qp})+(\vec{F}_q+\vec{F}_{lift,q}+\vec{F}_{vm,q}) \tag{2}$$

where, R_{pq} is the interphase between phase, P is the pressure (Pascal), F_q is the external force, $F_{lift,q}$ is the lift force in kg.m.s^{-2}, $\vec{F}_{vm,q}$ is the virtual mass force kg.m.s^{-2}

Table 2. CFD model input parameter

Input Parameters	Value
Inlet crude oil viscosity	9.75×10^2 kg m^{-1} s^{-1}
Crude oil density	8.58×10^2 kg m^{-3}
Heavy crude oil viscosity	5.0×10^2 kg m^{-1} s^{-1}
Heavy crude oil density	9.3×10^2 kg m^{-3}

3. Results and discussion

k-ε model has been used to find out the hydrodynamics of the fluid by first order upwind discretization and standard pressure momentum with pressure velocity algorithm. The inherent parameter considered was the jet parameters in jet flow distribution. So, CFD model was used to study the effect of velocity, time and nozzle head for single and four nozzles with variousorientations.

3.1Analysis with single jet

3.1.1 Effect of velocity on Mixing

The effect of time on jet velocity was investigated. Modeling was done for mixing profile in the first second with four velocities such as 5, 15,30 and 45 ms^{-1}.The profile was short in 5 ms^{-1}as shown in figure 5 (A) because the velocity is high near the nozzle so the dissipation of fluid jet into the tank contents results in diminish in velocity. The rate of penetration was moderate in 15 ms^{-1}as shown in figure 5 (B) which was due to the condition of no slip and force increased by the fluid leads to flow path sweeping. And also, the fluid was stagnant at the surface because of the lower penetration through the sludge lower layer. The velocity was further increased to 30ms^{-1} as shown in figure (c), there was significant changes were observed like increase in diameter, length of the penetration, radius of the jet and may reduce the sludge deposition (Randal et al., 2012). This increase in all the parameters obviously increased the mixing rate from the down layer to the top indicating that this velocity was optimum for mixing. The rate of mixing and depth of the penetration was gradually increased at 45ms^{-1}, but it was not much effective. This can be improved by increasing the mixing time (Sundarara and Selladurai, 2013.

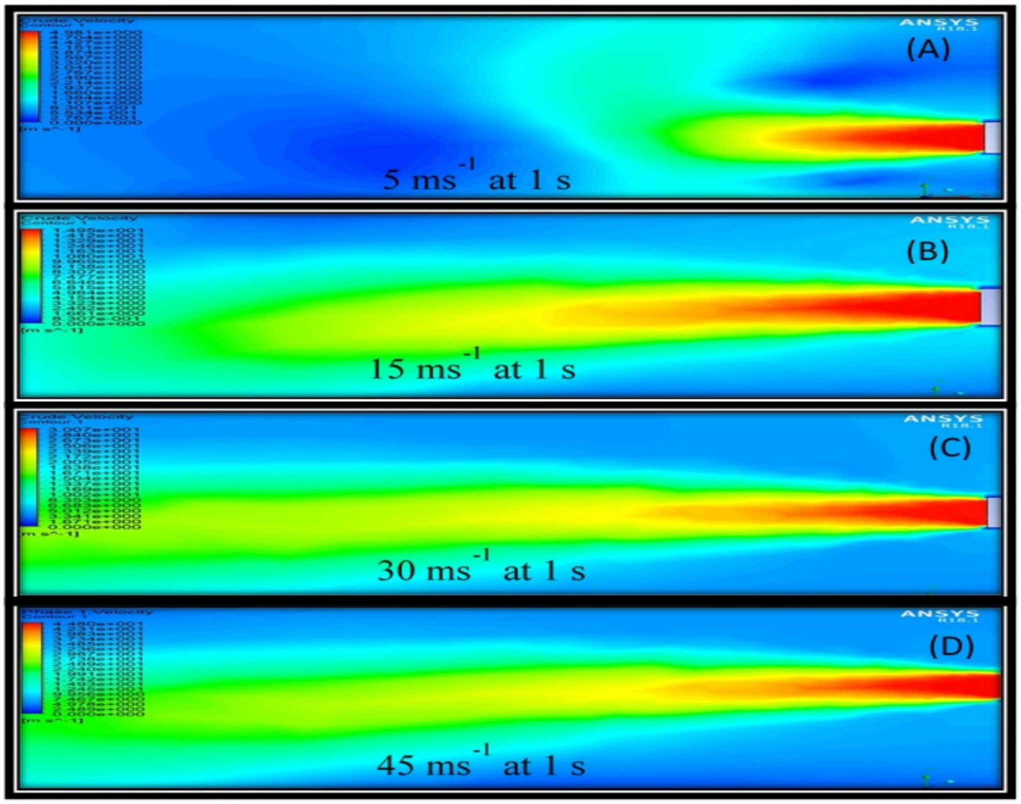

Figure 5. Velocity distribution profile of crude oil in 1s

3.1.2 Effect of time on mixing

The velocity profile was enhanced with increased time from 1 to 9 s. As the time increased, the disturbance was created the interface and proceeds to remove the stratified layer. The velocity rate increased show that increased in penetration of liquid with high circulation and rotational flow which leads to increase the mixing quality. The velocity profile is obvious in figure 6 (B & C), the penetration length of the jet increased with reduced width over time. As the time increased from 5 to 9s, the oil in the interface is forced back down the wall results in entrainment (Kevin and Jeffrey, 2010). Hence, 5s was an optimum time for mixing.

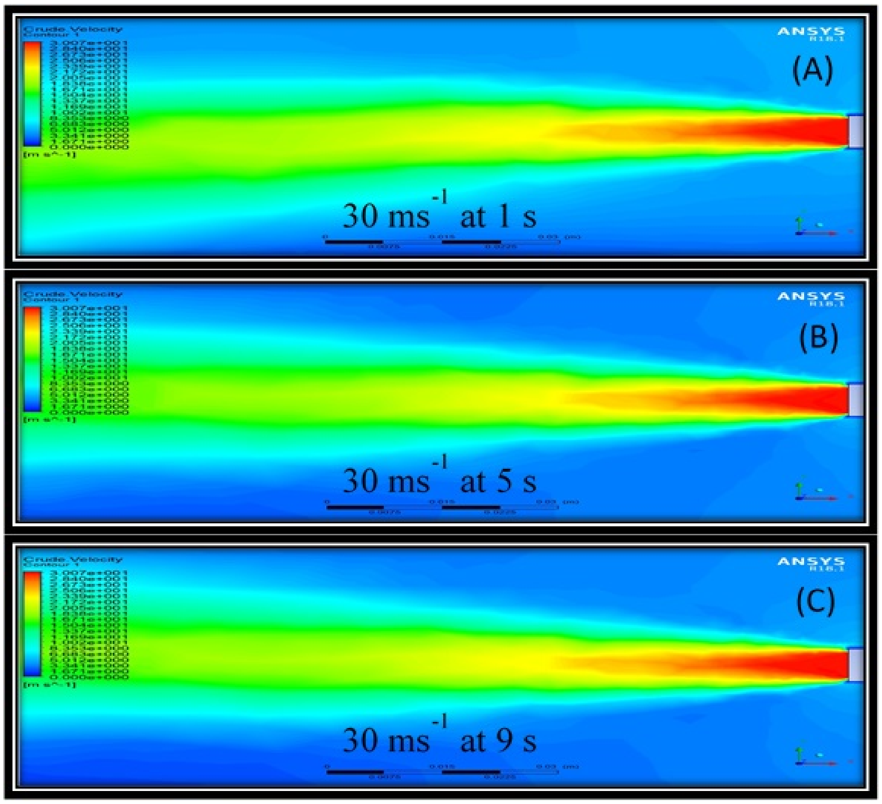

Figure 6 Jet velocity profile time (A) 1s, (B) 5s and (C) 9s

3.1.3 Effect of nozzle head on mixing

The mixing profiledistribution for the sludge at 1 m height with mixing time of 5 s was observed at velocity profile of 30ms^{-1} as shown in figure 7. The mixing was effective at 60cm as shown in figure 7 (A), the sludge in the bottom was easily mixed rather than at the top due to due to the higher-pressure head and opposing/resisting force when compared to the lift force exerted by the jet. And also, the jet dispersed the fluid radially outward the bottom with reduce in velocity (Patwardhan and Gaikward, 2003).The height was further increased to 80 cm, the mixing was moderate with good profile because of decrease in axial velocity by the surrounding liquid entrainment. The sludge distribution profile for 5s with a jet nozzle height of 100cm in 7(C), the mixing of sludge is not an effective because the bottom layers remains undispersed and stagnant. As the velocity increases, the resistance to the opposing force also increases and good mixing rate was attained. Hence, the sludge height of 1 m and the nozzle height of 80 cm was optimum for good mixing (Paul et al., 2004).

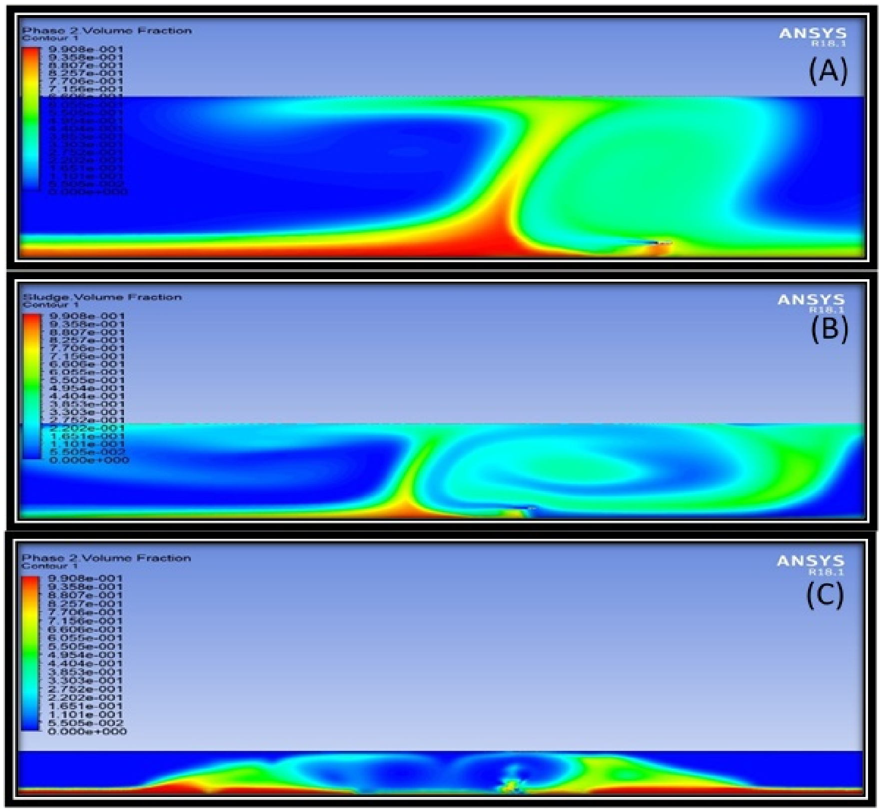

Figure7. Mixing profile distribution at jet nozzle height of (A) 60, (B) 80 and (C) 100cm

3.2 Analysis with multijet

The multi-jet performance was observed by optimizing such variables jet location, jet angle and time. Increase in time at constant velocity, causes the increase in jet path length and penetration length results in a higher mixing rate and better de-sludge process. The mixing effect is determined by taking average values of volume fraction of sludge at different location in the axis plane. The overall average volume fraction (OAVF) of the tank was in the range of 0.804 – 0.751

at 30s. The OAVF value was increased at the nozzle height of 0.5 and 0.7 than 0.3 due to the improper mixing results in desludge process. However, the OAVF of overall tank was in the range of 0.804-0.751 is overrated value for the mixing process. So, the procedure was continued for 60, 90 and 150s.

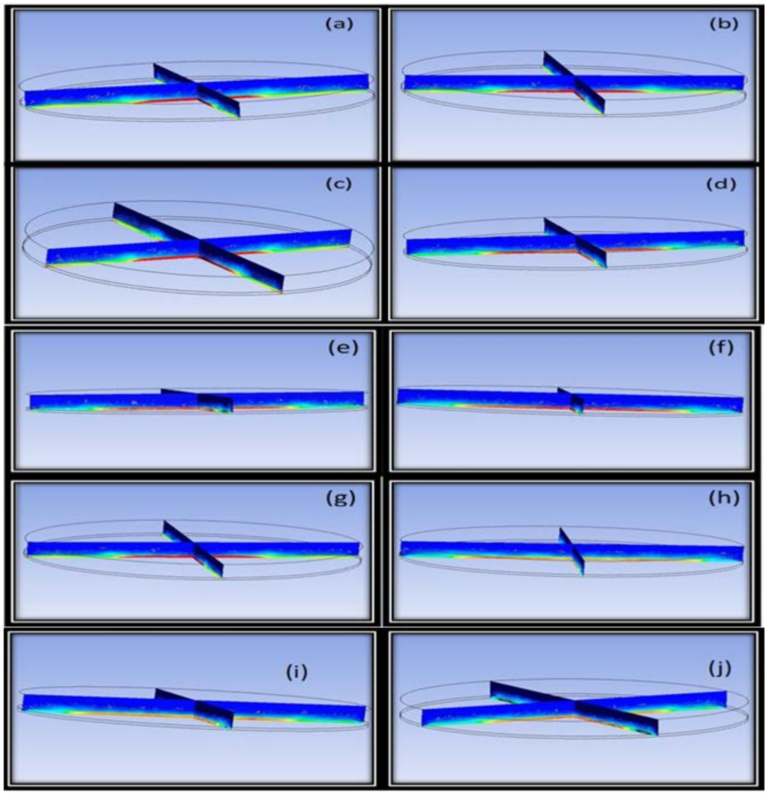

Figure 8.Sludge mixing distribution profile at 30s (A) jet height of 0.3 m, nozzles 0° , (B) jet height at 0.5 m nozzles 0° (C) jet height at 0.7 m nozzles 0° (D) jet height at 0.3 m with 2 nozzles 0° and 2 nozzles 45° declined jet. (e) jet height at 0.5 m with 2 nozzles 0° and 2 nozzles 45° declined jet., (f) jet height at 0.7 m with 2 nozzles 0° and 2 nozzles 45° declined jet. (g) jet height at 0.3 m Four nozzles with 45^0 angle downwards (h) jet height at 0.5 m four nozzles with 45^0 angle downwards. (i) jet height at 0.7 m four 45° declined jet., (j) jet height at 0 m four nozzles with 45^0 angle upwards.

3.2.1 Effect of multi nozzle position at 90 s

The OAVF inside the tank was observed in the range of 0.715-0.635, when the time was increased from 30 to 60s. It was quite higher. So, the time was further increased to 90s, the OAVF value was decreased to 0.571 by the effect of velocity and time.

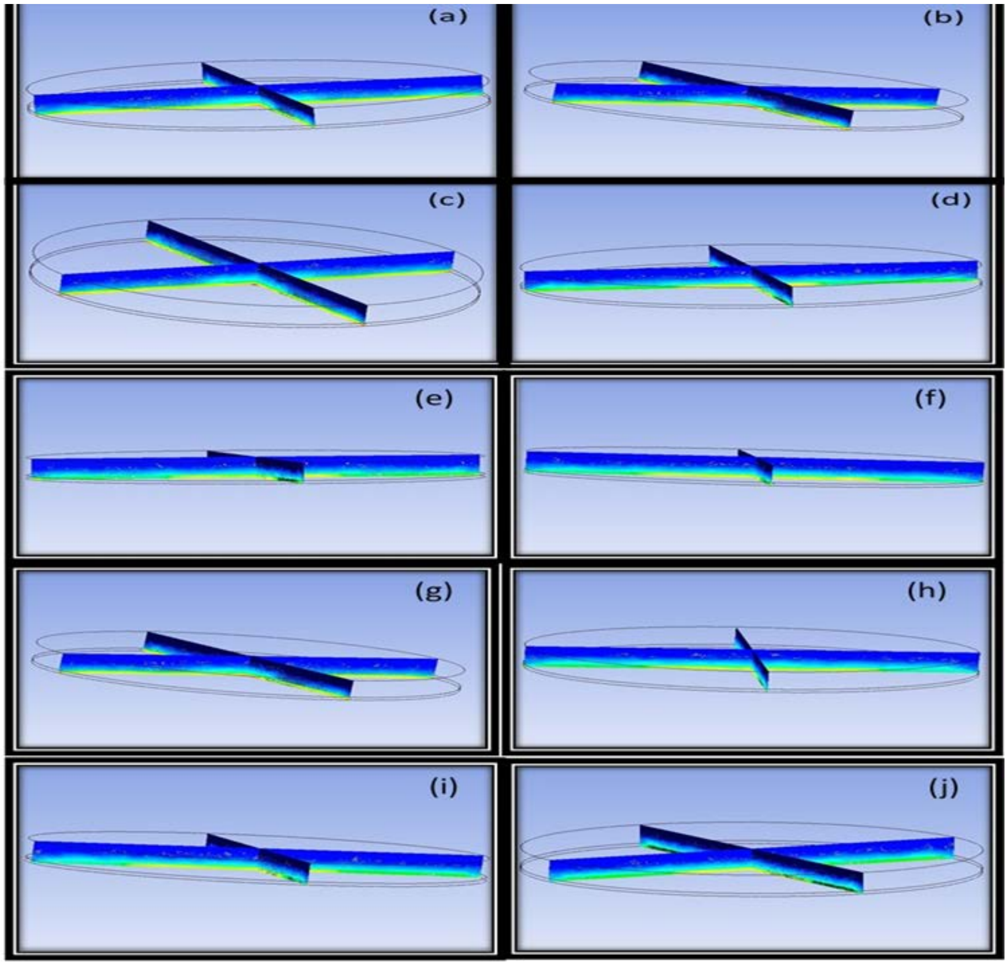

Figure 9. .Sludge mixing distribution profile at 90s (A) jet height of 0.3 m, nozzles 0° , (B) jet height at 0.5 m nozzles 0° (C) jet height at 0.7 m nozzles 0° (D) jet height at 0.3 m with 2 nozzles 0° and 2 nozzles 45° declined jet. (e) jet height at 0.5 m with 2 nozzles 0° and 2 nozzles 45° declined jet., (f) jet height at 0.7 m with 2 nozzles 0° and 2 nozzles 45° declined jet. (g) jet height at 0.3 m Four nozzles with 45^0 angle downwards (h) jet height at 0.5 m four nozzles with 45^0 angle downwards. (i) jet height at 0.7 m four 45° declined jet., (j) jet height at 0 m four nozzles with 45^0 angle upwards.

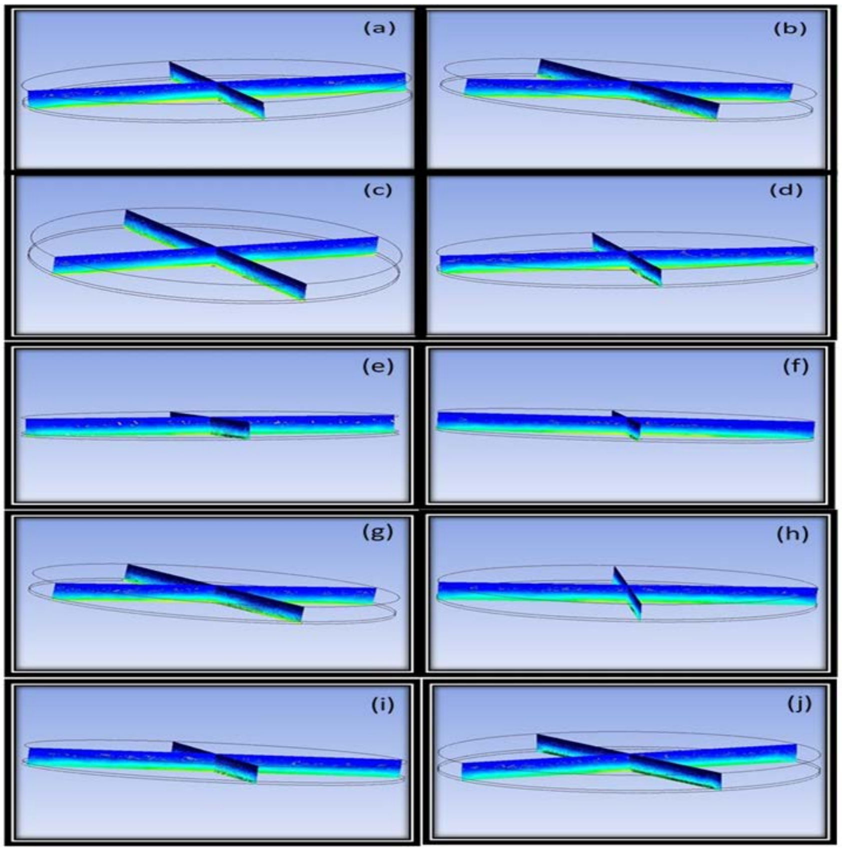

Figure 10. Sludge mixing distribution profile at 150s (A) jet height of 0.3 m, nozzles 0° , (B) jet height at 0.5 m nozzles 0° (C) jet height at 0.7 m nozzles 0° (D) jet height at 0.3 m with 2 nozzles 0° and 2 nozzles 45° declined jet. (e) jet height at 0.5 m with 2 nozzles 0° and 2 nozzles 45° declined jet., (f) jet height at 0.7 m with 2 nozzles 0° and 2 nozzles 45° declined jet. (g) jet height at 0.3 m four nozzles with 45^0 angle downwards (h) jet height at 0.5 m four nozzles with 45^0 angle downwards. (i) jet height at 0.7 m four 45° declined jet., (j) jet height at 0 m Four nozzles with 45^0 angle upwards.

The mixing time plotted against the jet injected at a 20° angle with five different angle of twisting.Normally both the jets are kept at 20° with 0° twisting itwill be shown in the fig 2 with the angle of twistingincreases, time required for mixing increases up to 45°twisting, surprisingly the mixing time get decreased athigher angles, and at 90° twisting gave a shortest mixingtime compared with the other four angle of the twisting.This procedure was repeated for three different Reynolds numbers. Reason for change in mixing time is due tochange in the flow pattern of the fluid in the tank. At 90° twisting the flow pattern was looking like a circular flowdue to this maximum jet length has been obtained whencompared to the other twisting degrees (Abdullah et al., 2017).

3.3 Effect of time on OAVF

The effect of time with respect to nozzle position is an important factor the OAVF was reduced when the time was increased from 30 to 150s, in which 150s of 0.3 m height nozzle has achieved the least OAVF value. The values obtained for OAVF with respect to time various nozzle height is shown in table.

Table 3 Effect of time and nozzle height on OAVF

Time (s)	**Height (m)**	**Angle (°)**		
		0	**0 & 45**	**45**
30	0.3	0.804	0.734	0.788
	0.5	0.825	0.783	0.803
	0.7	**0.834**	0.797	0.811
	0	-	-	0.751
60	0.3	0.715	0.668	0.67
	0.5	0.729	0.658	0.671
	0.7	0.741	0.697	0.684
	0	-	-	0.635
90	0.3	0.621	0.589	0.564
	0.5	0.617	0.583	0.589
	0.7	0.639	0.571	0.593
	0	-	-	0.591
120	0.3	0.552	0.528	0.536
	0.5	0.562	0.538	0.513
	0.7	0.584	0.528	0.523
	0	-	-	0.534
150	0.3	**0.471**	0.488	0.491

0.5	0.478	0.484	0.48
0.7	0.483	0.497	0.484
0	-	-	0.485

3.4 Evaluation and Verification of the Results

The accuracy of the results was analyzed and dependant on the turbulence models in all the numerical using RANS formulation. The wall y^+ data is a non-dimensional number similar to local Reynolds number, determining whether the influences in the wall adjacent cells are laminar or turbulent.

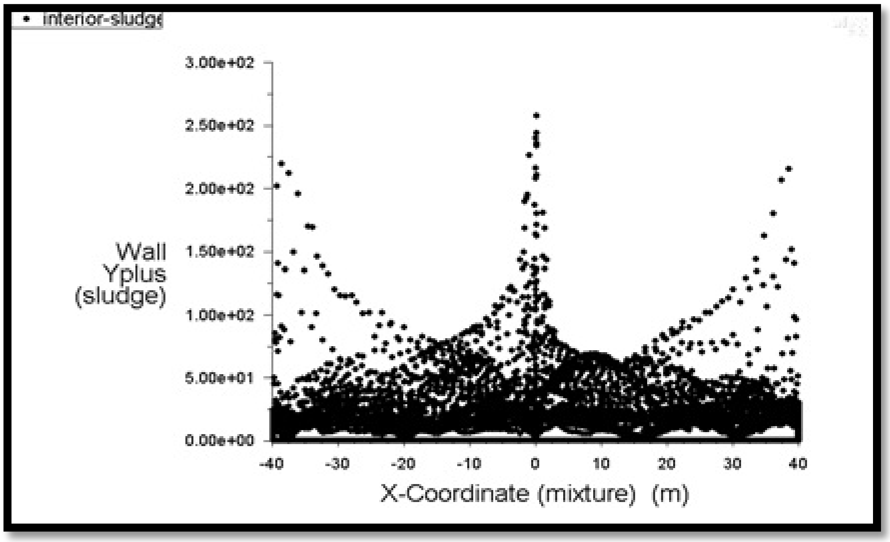

Figure 11. ***y*+ diagram of down wall**

y+ data shows sludge at the bottom and particle showing proper mixing distribution which were assessed to confirm the results. Less acceptable quantities of *y*+ 250 (< 300) indicates the sludge at the wall and at the bottom of the tank. The less in % deviation shows the good approximation of results obtained by modeling at time 150s in single jet position of 0.471 and multijet position of 0 and 45^0 as shown in table 4.

Table 4. Calculation of % Deviation

Time (s)	Volume Fraction at 0.3 m Jet Height		% Deviation
	Nozzle angle 0	Nozzle angle 0& 45	
30	0.804	0.734	8
60	0.715	0.668	6
90	0.621	0.589	5
120	0.552	0.528	4
150	0.471	0.488	**3**

4. Conclusion

A comprehensive 2-D and 3-D model for jet mixing was developed and studied using ANSYS software. In this work the sludge prevention in a crude-oil storage tank of 1 m sludge height was analysed. The effect of floating jet velocity in sludge prevention was studied using Euler-Euler method for the two-dimensional and three-dimensional CFD model crude-oil storage tank and the turbulence of the mixing flow was described using k-Epsilon model. It was observed that the increase in velocity changes the mixing pattern and improves the mixing of sludge with the crude oil. The effect of jet velocity, jet nozzle height, jet angle, number of jet and time on sludge prevention was analysed, optimum parameters were obtained from each model observations. It was predicted that effective mixing is attained when the jet velocity is at 30 m s^{-1}, jet nozzle height is 0.3 m and jet angle at 0^o. From the results it was inferred that the optimum time for efficient mixing was 150 s. The results were also validated using y^+ data in which the y^+value is below 300 is acceptable for the model. When comparing the number of jets used in the system the power required could be analysed to predict the efficiency factor.

Ethics approval and consent to participate

No animal or human material was not used

Consent for publication

There is no conflict of interest

Availability of data and materials

The corresponding author declare that the data supporting the findings of this study are available within the paper and any raw data files be needed in another format they are available from the corresponding author upon reasonable request.

Competing interests

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgements

The authors acknowledge the Electrochemical Engineering Laboratory, Department of Chemical Engineering, AC Tech Campus, Anna University for their support.

Author Contribution

Kavitha N P & Naveen kumar S: Writing – original draft, preparation and supervision.

Reference

Abdullah.S. M., Ahmed A., Ibrahim. A.,ThirumalaiKumar, R., 2017. Investigation of mixing time using jet mixer. Indian J. of Sci. Res. 292 -298.

Coldrey, P. W.,1978. Jet mixing. I. Chem. E. Course, Univ. of Bradford, England.

Fosset, H., 1951. The action of free jets in mixing of fluids. Tran. of the Inst. of Chem. Engg. 29**,** 322–332.

Fox, E. A., Gex, V. E., 1956. Single-Phase blending of liquids. AIChE Journal 2, 539–544, doi: https://doi.org/10.1002/aic.690020422.

Grenville, R.K., Tilton, J.N., 1996. A New Theory Improves the Correlation of Blend Time Data from Turbulent Jet Mixed Vessels. Chemical Engineering Research and Design 74,.3, 390-396.

Kevin, B., Jeffrey, M., 2010. Computational Fluid Dynamics (CFD) Simulations of Jet Mixing in Tanks of Different Scales. Nasa/Tm, 2010-216749.

LotfiNeyestanak, A. A., Tarybakhsh, M. R., Daneshmand, S., 2017, Study on the Effect of Jet Velocity on Mixing Performance and Sludge Prevention in Large- -Scale Crude-Oil Tanks by the CFD Technique. Kem. Ind 66 (5-6), 229−239.

Patwardhan, A. W., 2002. CFD modeling of jet mixed tanks. Chem.Eng. Science **57,** 1307–1318, doi: https://doi.org/10.1016/S0009-2509(02)00049-0.

Patwardhan A. W., Gaikwad S. G., 2003. Mixing in tanks agitated by jets. Chem. Eg. Res. Des. **81,** 211 - 220.

Paul, *E. L.*, Atiemo-Obeng, V. A., Kresta, S. M., *2004*. Handbook ofindustrial mixing science and practice. John Wiley & Sons. doi: https://doi.org/10.1002/0471451452.

Raja, T., Kalaichelvi, P., Anantharaman, N., 2007. Development of CFD model for optimum mixing in jet mixed tanks. Journal of Scientific & Industrial Research 66, 522-527.

Randal, W. Samstag, 1., Edward, A., Wicklein, 1., Roderick, D., Reardon, R. J., Leetch, R., 2012. Field and CFD Analysis of Jet Aeration and Mixing. https: //pubs. acs. Org /doi /abs /10.1021/ie0496683?journalCode=iecred

Sundarara, S., Selladurai, V., 2013. Flow and MixingPattern of Transverse turbulent jet in venture jet Mixer, Arabian Journal for Science and Engineering 38, 12, 3563–3573.

Toru, N., Tomoyuki, M,. Tsukasa, S., Hisashi, 2012. Removal of oiltank bottom sludge by novel biosurfactant. J. Jap.Petro. Inst. **55,** 138–141, doi: https://doi.org/10.1627/ jpi. 55.138.