Seventh International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
9-11 December 2009

CFD STUDY OF SLURRY HOMOGENIZER

Manoj KANDAKURE[1*], Vivek VITANKAr[1], Biswajit BASU[1], R. SRIVATSAN[2] and H. NAGARAJ[2]

[1] Process Engineering & Sciences, Aditya Birla Science and Technology Company Ltd., Navi Mumbai, Maharashtra 410208, INDIA
[2] Grasilene Division, Grasim Industries Ltd., Harihar, Karnataka 581123, INDIA

[*] Corresponding author, E-mail address: Manoj.Kandakure@adityabirla.com

ABSTRACT

An impeller has been designed and optimised using Computational Fluid Dynamics (CFD) to enhance the mixing of the solid particles with the liquid solution to give the homogenised slurry. To arrive at the optimised design of the impeller, various single and dual impeller configurations with variation in impeller type, impeller blade angle and diameter and impeller location have been simulated. These impeller configurations were evaluated based on the performance parameters such as the solid particles suction generated due to impeller, mixing value, unmixed solid particles and power consumption by the impeller assembly.

NOMENCLATURE

C	Impeller clearance from the tank bottom [m]
D	Impeller diameter [m]
H	Liquid level inside the tank [m]
N	Impeller rotational speed [rpm]
P	Power consumption by the impeller [W]
T	Tank diameter [m]
W	Impeller blade width [m]
θ	Impeller blade angle
τ	torque on the impeller [N m]

INTRODUCTION

In the industrial processes, homogenised slurries with specified consistency are produced in stirred tanks by mixing solids with the liquid. Examples include polymerisation reactions, fermentation processes, waste water treatment, pulp processing and minerals processing. The stirred tank design with optimised location for particle injection, particle loading and their physio-chemical properties are the key parameters affecting the performance. The impeller type, speed, location have significant impact on power consumption and the consistency of the slurries (Bakker and Frijlink, 1988; Coker, 2001; Khazam and Kresta, 2008; Ozcan-Taskin and Wei, 2003).

In the present work, an impeller has been designed using CFD with optimised speed, diameter and location to mix the solid particles to give the homogenised slurry. The solid particles (or the solid slurry) are lighter than the liquid solution and the impeller has to draw these lighter solid particles towards the tank bottom and keep them in suspension for longer duration for better mass transfer and reaction conversion.

In this batch operation, initially the solid particles and the liquid solution are fed from the top of 12.5 m^3 tank with H/T ratio of 1.07. A pyramid shaped impeller located at the bottom of the tank, is used to mix the solid particles with the liquid solution. At the end of the batch (batch time of 10 min), the homogenised slurry is drained from the tank for further processing. Thus the liquid level inside the tank frequently changes during the feeding stage and draining stage. The tank is never completely drained in order to avoid the splashing and sloshing of the slurry due to partially submerged rotating impeller. Hence the impeller is always kept in the completely submerged mode. It was observed during the operation that the desired consistency of the slurry due to this pyramid shaped impeller is not achieved leading to the inconsistent product quality. The objective of the present work is to understand the performance of the pyramid shaped impeller and improve the performance of the tank by designing a suitable impeller to enhance the mixing of the solid particles with the liquid solution.

MODEL DESCRIPTION

CFD Modelling Approach

Steady state simulations were carried out using commercial Computational Fluid Dynamics (CFD) software. The whole system was divided into around 1.7 million unstructured tetrahedra elements for the existing case with the pyramid shaped impeller (Table 1). Figure 1a shows the mesh used for this simulation. It can be clearly seen that the fine mesh was used near the impeller blades and the coarse mesh was used in the tank. Fine mesh was used in the vicinity of the impeller in order to accurately predict the momentum transfer from the rotating impeller to the surrounding fluid inside the tank. Frozen rotor model, based on the Multiple Reference Frame (MRF) approach, was used to incorporate the mixing due to the rotation of the impeller. Despite the known limitations of the standard $k-\varepsilon$ turbulence model, the overall performance of this model for simulating flows in stirred reactors is adequate for many engineering applications (Ranade, 2002). Hence standard homogeneous $k-\varepsilon$ model with the scalable wall function was used to simulate the turbulence nature of the mixing.

The solid loading for the present work was around 12%. Hence preliminary simulations were carried out by considering Lagrangian Particle Tracking multiphase model. However the results from the CFD simulations of various configurations showed that the solid particles fed from the top of the tank were revolving at the liquid surface and solid particles were not drawn towards the tank bottom. Thus the results were inconclusive in terms of the mixing achieved. Therefore Euler – Euler inhomogeneous multiphase model was chosen to model the multiphase liquid – solid mixing nature of the system. Both the phases were modelled as continuous phases. The two phases considered for the simulations are the solid slurry and the liquid solution. Both the phases were considered as the Newtonian fluids in order to model the fluid rheology. The mass transfer and the reactions taking place inside the tank are not modelled.

As it was discussed in the previous section, there is a frequent change in the liquid level during the feeding and draining stages of each batch. To incorporate this liquid level change and its impact on the mixing in the simulations, transient simulations need to be carried out. However a lot of computational time will be required to solve the transient simulation. Hence the steady state simulations were carried out. It was assumed that the tank is initially filled with solid particles and liquid solution as shown in Figure 1b. Liquid solution density is around 1200 kg/m^3 while the solid slurry density is around 500 kg/m^3. Thus the liquid solution, being the heavier component of these two phases occupies the bottom region while the solid slurry (consisting mainly of solids) occupies the top region. The volume occupied by the solid slurry is around 25% of the tank volume considered for the simulations. All the walls of the tank were modelled as no slip wall. Free slip wall boundary condition was used for the top liquid surface.

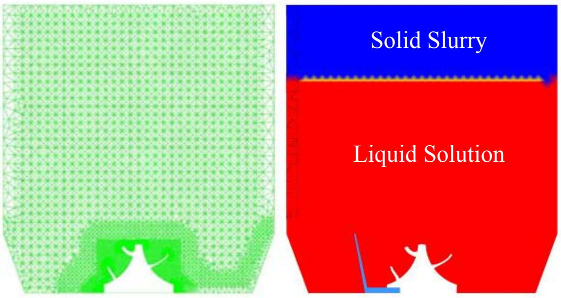


Figure 1: (a) Mesh generated for Case I (b) Initial volume fraction profile of Liquid solution for Case I.

Performance Parameters

The CFD results were evaluated qualitatively based on the velocity vector plots and the liquid volume fraction profiles and quantitatively based on the various performance parameters such as mixing value, unmixed solids, suction generated and the power consumption by the impeller. These performance parameters are discussed in detail in the following paragraphs.

Mixing value

As discussed in the earlier section, the solid slurry occupies 25% of the tank volume and liquid solution occupies the remaining 75% of the tank volume. Thus the complete homogeneous mixing of the solid slurry and the liquid solution inside the tank will have liquid volume fraction of 0.75 throughout the tank. For the present work, the mixing value is defined as the percentage of the tank volume having liquid volume fraction within the range of 0.65 – 0.85. Accordingly for the complete homogeneous mixing, this mixing value will be 100%; while the mixing value will be 0% for no mixing at all. Here it can be stated that impeller performance can be considered good when it achieves the mixing closer towards the ideal mixing.

Figure 2 shows the profile of the percentage of tank volume having corresponding liquid volume fraction for a typical simulation. The mixing value is calculated as the area under the curve falling inside the light shaded area (liquid volume fraction within the range of 0.65 – 0.85).

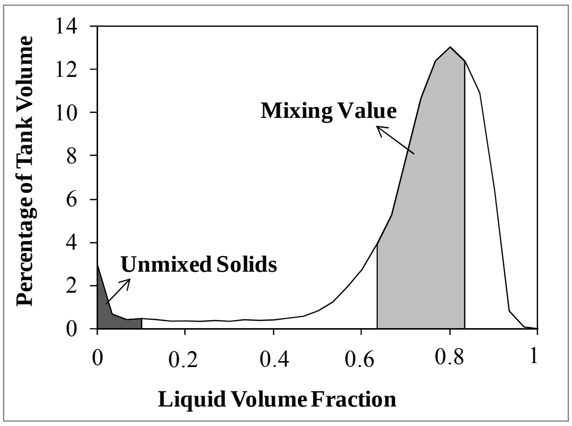


Figure 2: Typical liquid volume fraction values across the tank volume

Unmixed solids

Unmixed solids is defined as the percentage of the tank volume having liquid volume fraction smaller than 0.1 (in other words, the solid slurry volume fraction greater than 0.9). For this solid slurry, there is marginal or no mixing at all. As stated earlier, the solid slurry occupies 25 % of the tank volume. Thus the unmixed solids will be 25% for no mixing at all while the amount will be 0% for complete homogeneous mixing. It can be stated here that the impeller performance can be considered good if the unmixed solids is less. From Figure 2, the amount of the unmixed solids is calculated as the area under the curve falling inside the dark shaded area for a typical case.

Suction generated

Suction generated in terms of mass flow rate of solid slurry moving in downward direction at various axial locations is calculated from the CFD simulations. Figure 3 shows the variation of this suction generated with the normalised axial distance (normalisation done by the liquid level considered for the CFD simulations) from the tank bottom. From this Figure, total suction generated by impeller is calculated as the area under the curve of the suction generated at different axial distances from the tank bottom.

Thus it can be stated here that more the ability of the impeller to draw the solid slurry towards the tank bottom, better will be the impeller performance.

Power consumption by the impeller

The power consumption (P) is calculated as the product of torque (τ) on the impeller blades and the angular velocity ($2\pi N/60$ in rad/s) from the following equation.

$$P = (2\pi N/60)\ \tau \qquad (1)$$

Where, torque (τ) is obtained by integration of the pressure on the impeller blades from the CFD results.

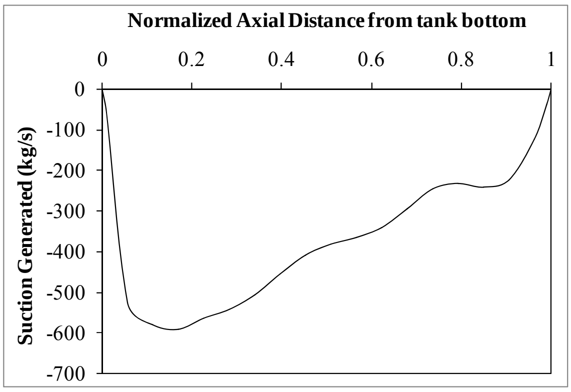


Figure 3: Typical variation of suction created with the normalised axial distance from the tank bottom

Due to the "specific application" considered in the present article, the quantitative validation of the simulation results was not possible. Qualitative validation for the base case configuration was done through the visual observations in the plant in terms of the solid slurry floating at the top region of the tank. In the present work, the relative comparison of the CFD predictions for variation in the impeller geometrical parameters and operating conditions were carried out. This relative comparison, in terms of above performance parameters discussed, will help in obtaining the optimum impeller configuration for better mixing.

RESULTS

Case I (Base case):

This is the base case where a bottom mounted pyramid shaped impeller is used to mix the solid slurry with the liquid solution inside the tank. This impeller is continuously rotating at the speed of 500 rpm.

Results from the CFD simulations for this case are shown in the Figure 4. Figure 4a shows the vector plots of the normalised velocities (normalised by the impeller tip velocity) while Figure 4b shows the liquid volume fraction profile obtained from the CFD simulations. It can be seen from Figure 4a that the pyramid shaped impeller creates an good amount of circulation in the bottom half region of the tank however the vectors in the top half region suggests less suction generated by this impeller in that region. Figure 4b clearly shows that only small amount of the solid slurry is being drawn towards the tank bottom by the impeller and most of the solid slurry remains in the top region of the tank. As stated earlier, ideally for completely homogenised mixture of solid slurry and liquid solution, the liquid volume fraction values should be around 0.75 throughout the tank region. From Figure 4b it can be clearly seen that the amount of liquid volume fraction near the homogenised condition is very small indicating the poor mixing achieved due to this pyramid shaped impeller. Thus it can be stated that the suction generated by the impeller is not sufficient enough to completely draw the solid slurry towards the tank bottom.

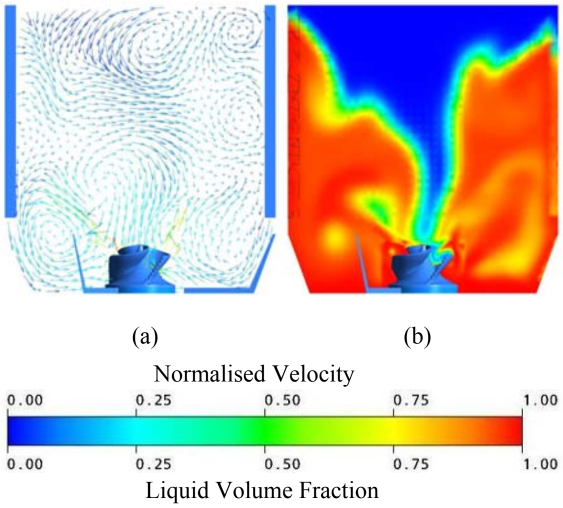


Figure 4: CFD results for the Case I (a) Normalised velocity vector plot (b) Liquid volume fraction profile

The quantitative evaluation of the case in terms of the various performance parameters reported in the previous section are tabulated in Table 2. From Table 2, the mixing value for this case is just 11.26% while the unmixed solids are 17.46%. Thus a very poor mixing is achieved due to this impeller. Total suction generated for this case is -97.1 kg/s and the power consumption calculated from the CFD simulations is 60.0 kW. It can be seen from the Figure 4 and Table 2 that there is a need to increase the suction generated by this impeller. The simulations with the increased impeller rotational speed were carried out. However marginal improvement in the performance was observed. The results of this case with the increased speed of rotation are not shown here.

Thus an improvement in the suction created by the impeller in order to draw more solid slurry towards the tank bottom and subsequent improvement in the mixing can be achieved by either introducing one more additional impeller in the existing system or by changing the design of the impeller. Various impeller configurations were simulated with variation in the impeller blade diameter, blade angle, impeller clearance, type and rotational speed. Impeller types studied were the pitched blade down-flow turbine (PBTD), curved blade down-flow turbine (CBTD) and the pyramid shaped impeller. However, only some of these simulations are selected for the discussion in this work and their details are listed in Table 1.

Case	Impeller Type	Tetrahedra elements in millions	D/T[b]	W/D[b]	C/T[b]	θ[b]	N (rpm)
I	Pyramid[a]	1.7	0.258	-	-	-	500
II	Pyramid + PBTD	2.5	0.250	0.30	0.33	30^0	500
III	Pyramid + CBTD	1.8	0.250	0.30	0.33	45^0	500
IV	CBTD	1.5	0.286	0.35	0.33	45^0	500
V	CBTD	1.5	0.286	0.35	0.17	45^0	500
VI	CBTD	1.5	0.286	0.35	0.17	45^0	350

Table 1: Various impeller configurations studied
([a] Pyramid shaped impeller is located exactly at the tank bottom; [b] Parameters are for the top impeller in the dual impeller configurations; PBTD: Pitched Blade Down-flow Turbine; CBTD: Curved Blade Down-flow Turbine)

Case	Total Suction generated (kg/s)	Mixing Value (%)	Unmixed Solids (%)	P (kW)
I	-97.1	11.26	17.46	60.0
II	-216.9	30.71	10.65	74.3
III	-270.6	77.66	3.45	118.5
IV	-343.6	68.28	2.96	150.0
V	-372.0	67.10	2.46	157.7
VI	-227.1	56.36	8.80	50.3

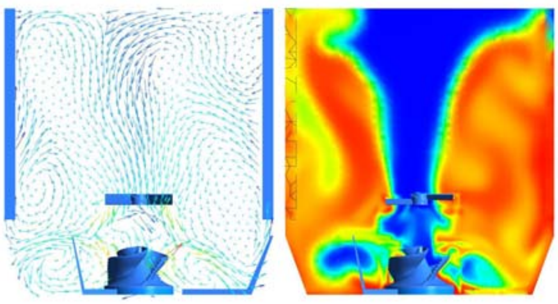
Table 2: Evaluation of various impeller systems studied based on the performance parameters.

Case II:

Case II is the modification to the case I wherein a pitched blade down-flow turbine (PBTD) is utilised in addition to the pyramid shaped impeller to improve the suction. PBTD impeller is placed at 1/3rd location from the tank bottom. Blade angle of 30^0 was selected as it generates more axial flow compared to the impeller with higher blade angles (Carpenter, 1997). The rotational speed of both the impellers is kept at 500 rpm.

The CFD results for Case II are shown in the Figure 5. Figure 5a shows the vector plot of the normalised velocities while Figure 5b shows the liquid volume fraction profile over a plane along the axis of the tank. From Figure 5a, it can be clearly seen that this dual impeller system is able to create more suction (more downward velocity vectors) in the top half region and better circulation in the bottom half region of the tank. Due to the strong circulation loops near the tank bottom on the both sides of the impeller, the solid slurry drawn by these impellers remain near the tank bottom within these circulation loops as can be seen in Figure 5b. Also from Figure 5a, the flow profile developed by the pyramid shaped impeller is getting suppressed due to the flow profile of the PBTD impeller.

From Table 2, the mixing value for this case II is 30.71% and the unmixed solids are 10.65%. This shows a better mixing compared to the base case (Case I) due to the better total suction generated (-216.9 kg/s) compared to the base case value of -97.1 kg/s. However the desired mixing was not achieved by this impeller configuration.



(a) (b)

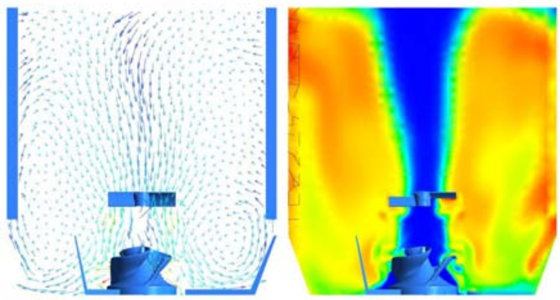
Figure 5: CFD results for the Case II (a) Normalised velocity vector plot (b) Liquid volume fraction profile (Legends are same as in the Figure 4)

Case III:

In literature (Kumaresan, 2006), it has been stated that the Hydrofoil (HF) impeller generates more axial flow suction and hence better mixing than pitched blade down-flow turbine (PBTD). Hence in order to increase the draw-down of the solid slurry, a curved blade down-flow turbine, which has a similar design as that of hydrofoil impeller, is used in the present work.

The geometry of this case III is similar to that of the Case II except that the PBTD impeller with blade angle of 30^0 in Case II is replaced by the Curved blade down-flow turbine (CBTD) with blade angle of 45^0. Since in the previous case (Case II), the flow profiles generated by the pyramid shaped impeller were getting suppressed by the PBTD impeller with blade angle of 30^0, the blade angle for this case was changed to 45^0.

Figure 6 shows the CFD results for this case. It can be seen from Figure 6a that more velocity vectors are in the downward direction in top as well as the bottom half of the tank. Thus there is a formation of two bigger recirculation loops resulting into a better suction generated inside the tank. However the flow profiles generated due to the pyramid impeller are completely getting suppressed due to the flow profiles generated by the CBTD impeller. Liquid volume fraction profile shown in Figure 6b suggests better mixing of solid slurry with the liquid solution as the liquid volume fraction is around 0.75 (value for complete homogeneous mixing) in the major portion of the tank.



(a) (b)

Figure 6: CFD results for the Case III (a) Normalised velocity vector plot (b) Liquid volume fraction profile (Legends are same as in the Figure 4)

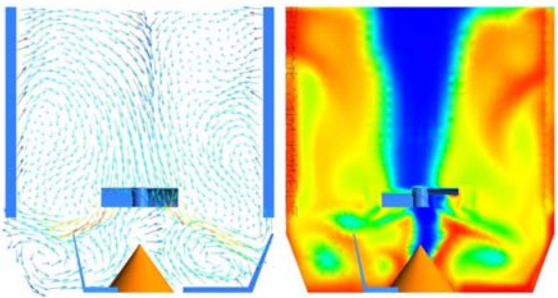
From Table 2, it can be seen that the mixing value for this case III is 77.66% and the unmixed solids are 3.45%. This shows much better mixing in this case compared to the previous cases, Case I and II. Total suction generated in this case (-270.6 kg/s) is also better than that for the previous two cases. However the power consumption by this dual impeller system is 118.5 kW which is almost double compared to that for the base case (Case I).

Case IV:

As can be seen from the results of the previous case (III), the flow profiles of the pyramid impeller was getting completely suppressed due to the flow profiles generated by CBTD impeller. Hence in this Case (IV), the bottom pyramid shaped impeller is removed and a cone is placed at that location which will help in developing the flow profile due to CBTD impeller. The presence of cone at the bottom also minimizes the size of the dead zones at the tank bottom. The diameter and blade width of CBTD impeller in this case (IV) is increased from the impeller values for the Case III (Table 1).

Figure 7 shows the CFD results for this case (IV). From the vector plot in Figure 7a, the formation of four circulation loops can be clearly seen. Two circulation loops are near to the tank bottom and these are smaller in size compared to the two circulation loops in the tank top region. As can be seen from Figure 7b, the percentage of the tank volume which has the liquid volume fraction near to 0.75 is also high compared to the Case I (Figure 4b).

From Table 2, the mixing value for this case IV is 68.28% while the unmixed solids are 2.96%. Total suction generated in this case (-343.6 kg/s) is much more than that for the previously discussed three cases (Case I, II and III). From Table 2, it can be seen from performance parameters for case III and case IV that there is no direct relation between the total suction generated and the mixing value. This is due to the fact that the Case III is the dual impeller system and Case IV is the single impeller system. When compared with the base case (Case I), the mixing achieved and the total suction generated are higher in case IV while the unmixed solids are less in case IV. Thus the case IV has better mixing compared to the base case. However the power consumption in this case IV is 150 kW which is almost 2.5 times of the power consumption in the base case (Case I).



(a) (b)

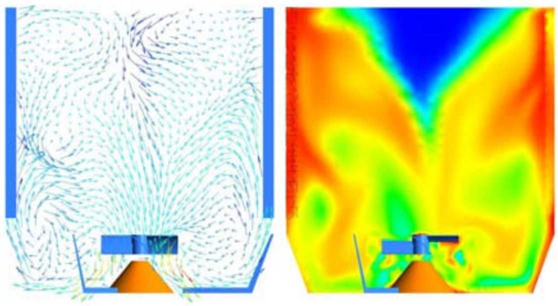
Figure 7: CFD results for the Case IV (a) Normalised velocity vector plot (b) Liquid volume fraction profile (Legends are same as in the Figure 4)

As mentioned earlier in the previous section, the impeller needs to be kept in the fully submerged condition in order to avoid the splashing of the solid slurry against the tank wall during frequent feeding stage and draining stage of the tank. Thus during each batch, the liquid level needs to be maintained more than the impeller location. Thus the effective production in terms of the amount of the homogeneous slurry that can be drained in every batch gets reduced in the cases II, III and IV compared to the base case (Case I) due to the higher values of the impeller clearance in the cases II, III and IV compared to case I. This reduction in the effective production per batch (considering the same batch time) will affect the overall plant production. In order to have the same production as in case I, the CBTD impeller in the case IV needs to be lowered and the performance needs to be evaluated.

Case V:

This case V is the modification to the Case IV wherein C/T for the CBTD impeller is reduced from 0.33 to 0.17. The bottom cone is again utilised to improve the flow profile developed due to the impeller and to reduce the dead zones. By placing the impeller near the tank bottom, the effective production in terms of the amount of the homogeneous slurry that can be drained in every batch will be higher than that for the case IV and will be similar to that for the case I.

CFD results for this case V are shown in the Figure 8. It can be clearly seen from the Figure 8a that the bottom cone is helping the flow profile developed by CBTD impeller and there is better mixing compared to the that in case I. The amount of unmixed solid slurry (blue region in Figure 8b) is less compared to that in the case I and the percentage of the tank volume having the liquid volume fraction near to 0.75 in this case is more than that in the case I. This indicates that the mixing in the case V is better than that in the case I.



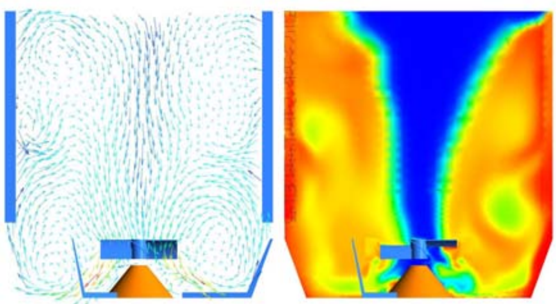
(a) (b)

Figure 8: CFD results for the Case V (a) Normalised velocity vector plot (b) Liquid volume fraction profile (Legends are same as in the Figure 4)

From Table 2, it can be seen that the mixing value for this case V is 67.1% while the unmixed solids are 2.46%. Total suction generated for this case is -372.0 kg/s. Thus the mixing achieved and the suction generated inside the tank is more than that for the base case (Case I). However impeller power consumption in Case V is 157 kW while the impeller power consumption in Case I is 60 kW. Thus the better mixing is achieved due to higher power consumption. In order to implement the impeller configuration in case V in the plant, the existing motor needs to be replaced and this would increase the implementation cost. Thus there is a need to achieve better mixing inside the tank for the same amount of power consumption as that in the base case.

Case VI:

As seen in the previous case V, there is a need to achieve the improvement in the mixing keeping the same power consumption by the impeller. This was achieved by reducing the impeller rotational speed from 500 rpm in Case V to 350 rpm for this case VI and keeping all the other geometrical and operational parameters constant.



(a) (b)

Figure 9: CFD results for the Case VI (a) Normalised velocity vector plot (b) Liquid volume fraction profile (Legends are same as in the Figure 4)

The CFD results for this case VI are shown in the Figure 9. Formation of circulation loops can be clearly seen in the Figure 9a. The velocity pattern developed in this Case VI is much better than that in the Case I (Figure 4a). The percentage of tank volume having liquid volume fraction near to 0.75 for this case (Figure 9b) is more than that for the Case 1 (Figure 4b). This is evident from the Table 2 as the mixing value for the Case VI is 56.36% and the unmixed solids are 8.80%. Total Suction generated is -227 kg/s. The impeller power consumption for this case is 50.3 kW which is lower than that for the Case I. Thus a better mixing is achieved in the case VI with lower power consumption and having the similar effective production compared to that in the case I.

A suitable impeller configuration can be selected from the various cases discussed above. For the selection of the impeller configuration, various implementation criteria such as ease of implementation, implementation cost, power consumption, etc can be used apart from the performance criteria of these impeller configurations. The trials in the plant will help in validating and testing the performance of the selected impeller configuration.

CONCLUSION

Various impeller configurations were simulated and their performances were evaluated qualitatively in terms of the velocity vector plots and liquid volume fraction profiles and quantitatively in terms of the mixing values, unmixed solids, the total suction generated due to the impeller and the impeller power consumption. All the implementation issues such as the power consumption, the effective production, implementation cost and ease of installation can be considered while selecting the suitable impeller configuration.

REFERENCES

BAKKER, A. and FRIJLINK, J.J., (1988), "The drawdown and dispersion of floating solids in aerated and unaerated stirred vessels", *Chem. Eng. Res. Dis.*, **67**, 208-210.

CARPENTER, K.J., (1997), Agitation, In: Sharratt P.N. (ed) Handbook of batch process design. Blackie Academic and Professional, London, UK, Chapter 4, 107-152

COKER, A. K., (2001), Modeling of chemical kinetics and reactor design, Gulf Professional Publishing, Boston.

KHAZAM, O. and KRESTA, S.M., (2008), "Mechanisms of solids drawdown in stirred tanks", *Can. J. Chem. Eng.*, **86** (4), 622-634.

KUMARESAN, T. and JOSHI, J.B., (2006), "Effect of impeller design on the flow pattern and mixing in stirred tanks", Chem. Eng. J. **115**(3), 173-193.

OZCAN-TASKIN, G. and Wei, H., (2003), "The effect of impeller-to-tank diameter ratio on draw down of solids", *Chem. Eng. Sci.*, **58**, 2011-2022.

RANADE, V.V., (2002), Computational Flow Modelling for Chemical Reactor Engineering, Academic Press, New York.

