The 2^{nd} International Seminar on Science and Technology
August 2^{nd} 2016, Postgraduate Program Institut Teknologi Sepuluh Nopember, Surabaya, Indonesia

Computational Fluid Dynamics Simulation of Clearance Effect in High Solid Loading Polydisperse Solid-Liquid Mixing

Iman Mukhaimin[1], Tantular Nurtono [1], and Sugeng Winardi[1]

Abstract*— A high solid loading concentration of solid-liquid mixing was investigated to observe the effect of ratio C, Clearance, and T, diameter tank, with C/T 0.33; C/T 0.25 and C/T 0.17 on local volume of hydrodynamic and spatial distribution of polydisperse solid suspension using CFD, Computational Fluid Dynamics. The 45º pitch blade turbine, diameter 0.5T, with down pumping flow simulation was used to remove solid particle from bottom of the tank. The tank is also equipped with four baffle with the size of 0.1T.. A solid-liquid mixing consists of five fractions of glass beads with equal proportion (X1=X2=X3=X4=X5=0.2X) have 40% wt total solid concentration with liquid fraction is aqueous solution of NaCl. The effect of ratio C/T at impeller speed 612 rpm create a flow pattern in the tank different. Effect ratio C / T also indicated the distribution on solid had a good uniformity index when N≥ Njs, just suspended speed. The highest uniformity was obtained on C/T 0.17. it also made difference power consumption on each geometry with C/T 0.17, 0.25, and 0.33 respectively are 251.18, 238.13, and 270.65watt.

***Keywords*— Polydisperse, simulation, stirred tank, solid-liquid mixing, pitch blade turbine**

I. INTRODUCTION

A solid-liquid mixing generally gives a large influence to quality of a product. The quality of a solid-liquid mixture is generally based on the homogeneity of the mixture. Homogeneity of the mixture can be achieved if mixed at the optimum experimental conditions. Optimum conditions of a mixture will be very different for each type of material mixing. It is very dependent on the distribution of solid in a solution. Therefore, it would be interesting to study not only the empirical approach but also a modeling approach.

Empirical approach always has the main problem obtaining data distribution of the result of mixing solid. The development of research continually made it easier to predict the distribution of the solid-liquid mixing. Starting from LDV method, Laser Doppler velocimetry [1] consider the local speed of dispersed and continuous phase, PGT, Pressure Gauge Technique [2], compare the drag coefficient approached by Pinelli et.al and Brucato et.al in turbulent flow until the novel method PEPT, Positron Emission Particle Tracking, [3].

PEPT is the novel method to obtain data distribution of solids in liquids. One of the parameter is uniformity index of suspense as conducted by experiment with PEPT that defines quality of mixing with uniformity index which increases concentration of mixture will producing a higher uniformity index. The spatial solid distribution is well predicted except near the base of the vessel and underneath the impeller where it is largely overestimated. However, predictions improve significantly with increasing solid concentration [3]. Uniformity index represents a value of quality of mixing in each cell volume. A higher volume obtained when distribution of each solid fraction has same concentration throughout the tank. Furthermore [4] with PEPT states that type and direction of the rotation of the impeller effects on the homogeneity of the solid particles in suspense continuous phase. The technique is still far from economical, it requires a large cost in operation. modeling approach.is needed to solve this problem.

The modeling approach are widely used in a study using CFD. Numerical approach based on CFD was needed to reach the approach turbulence, multiphase and impeller modeling [5]. This approachment minimized the limitation in experiments to predict the real-time simulation of 3D images as conducted by [6] that Reactor tank design parameters, types and dimensions, will influence on mixing profile. Ratio C/T is one of the main parameters. The effect of C/T will define a mechanism of fluid to lift particle up from the bottom of the tank and the effect of C/T will significantly influence on impeller speed used [7]. C/T ratio conduces the hydrodynamic of the tank to change. The smaller C/T ratio will need a bit power to lift all particles from the bottom of the tank and it will greatly affect the rotational impeller speed.,

In addition, there was a concern on the condition of blending operations containing rotary impeller speed and solid concentration. Rotational speed of the impeller becomes an important factor because the energy which is generated from the rotation of the impeller is needed to remove particles that are perfectly suspended [8]. Determining the rotational speed of the impeller will be directly proportional to both the concentration and diameter of solid particles [9]. The impeller speed also becomes parameter to estimate the particle settling velocity and drag coefficient in flow stirred tank [10].

Monodisperse solid-liquid suspensions have been reported in a number of mixing studies, both numerically and experimentally, though the majority of them are focused on dilute suspensions which have little practical relevance. However, suspensions of particles of different diameters in turbulent mixing in stirred vessels have been reported in very few works. Studies have found that smaller particles distribute more uniformly than larger particles [11]. Particle used in the mixing process generally does not consist of one size, monodisperse, but rather consists of several size of particle diameter,

---

[1] Iman Mukhaimin, Tantular Nurtono, and Sugeng Winardi are with Departement of Chemical Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, 60001, Indonesia. Mukhaiminiman056@gmail.com; tantular@chem-eng.its.ac.id; swinardi@chem-eng.its.ac.id.

polydisperse Thus, needing approachment to predict the effect of particle size on particle and phase interaction. [12] with PEPT experiments and CFD simulations states that monodisperse particles with N=2Njs have higher uniformity index than N=NJS.

The accuracy of CFD in the mixing of suspensions of solid particles of different sizes, i.e. binary or poly disperse, especially in dense suspensions has not been sufficiently studied and, in particular, little information exists on flow hydrodynamics. Hence, a detailed validation using detailed 'pointwise' measurements of 3-D velocities as well as the distribution of the phases and phase components, especially under conditions of moderate to high solid loadings, is needed in order to assess the capability of the numerical models and CFD codes available in these complex flows and the influence of polydisperse on interaction and distribution in a mixture hasn't been explained clearly until conducted this study.

II. MATERIAL AND METHOD

This research simulated a system of polydisperse solid-liquid mixing as conducted by [4] experiment with PEPT. The mixing simulation were conducted in a fully-baffled flate-base perspex vessel of diameter T= 0.288 m., agitated by a down-pumping 6-blade 45° pitched-turbine (PBT) of diameter D=T/2, as illustrated in Figure 1, The height of suspension was set at H=T and three kind of impeller off-bottom clearance, C/T, were 0.17; 0.25; and 0.33.

The model used is a stirred tank fullgrid 3-D (360°) to get a flow pattern closed with the actual conditions. Simulations carried out by ANSYS FLUENT 15.1. Materials used in the simulation of solid-liquid mixing is defined in six phases. The suspending liquid was an aqueous solution of NaCl of density 1150 kg/m^3 and solid particles used were spherical glass beads of density 2485 kg/m^3. Five nearly-monomodal particle size fraction, (dp 1mm; 1.5 mm; 2 mm; 2.5 mm; 3 mm), were used to make polydisperse solid-liquid suspensions. The five-sized fractions were mixed in equal proportion with a total concentration,X, 40 wt%, i.e $X_1= X_2 =X_3 =X_4 =X_5=0.2\ X$. The Simulation was conducted with impeller speed at 612 rpm (C/T 0.17 = N>Njs , C/T 0.25 = N=Njs, and C/T 0.33 = N<Njs)

Geometry domain of Stirred tank is divided into two parts: moving and stationary zone. Multi-component modeling of solid phase and liquid phase of Eulerian-Eulerian multiphase model was used. The Multiple Reference Frames (MFR) method was used to take into account the interaction between the stationary baffles and the rotating impeller pitch blade turbine in conjunction with the well known mixture $k - \varepsilon$ turbulence in transient condition during 60s. The Volume of stirred tank was divided to 50 zone to determine spatial distribution of mass of each size of solid particle.

III. RESULTS AND DISCUSSION

CFD is widely used by many researchers to analyze the solid-liquid suspension system with visual, qualitative and quantitative. Although most CFD studies with numerical approach still focused on monodisperse systems. CFD can predict hydrodynamics and solid particle size distribution properly than others but for polydiperse system has not been studied comprehensively.

This study using 3D-images CFD observe hydrodynamic flow and distribution of solid particles in a stirred tank. The movement of particles in the mixing process is depended on the speed and vectors resulting from the mixing process. 6-blade 45° pitched-turbine (PBT) generate a pumping down flow that resulted from the tip of the impeller to the bottom of the tank. Thus this flow continue to keep the particles remain elevated from the bottom of the tank which will increase the value of uniformity index.

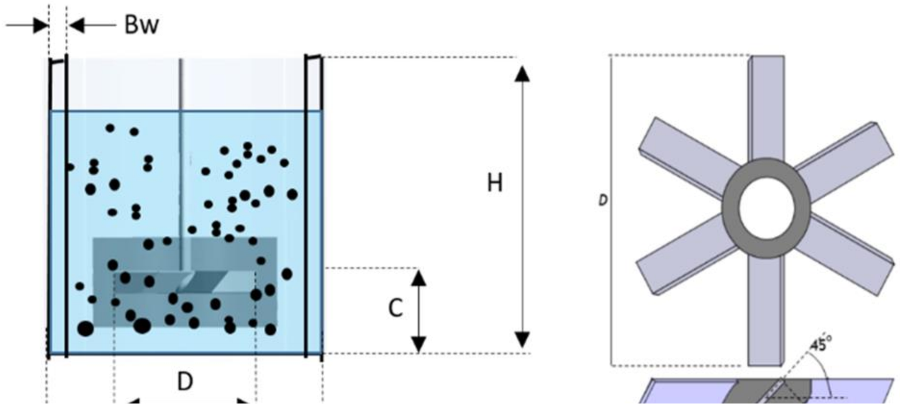
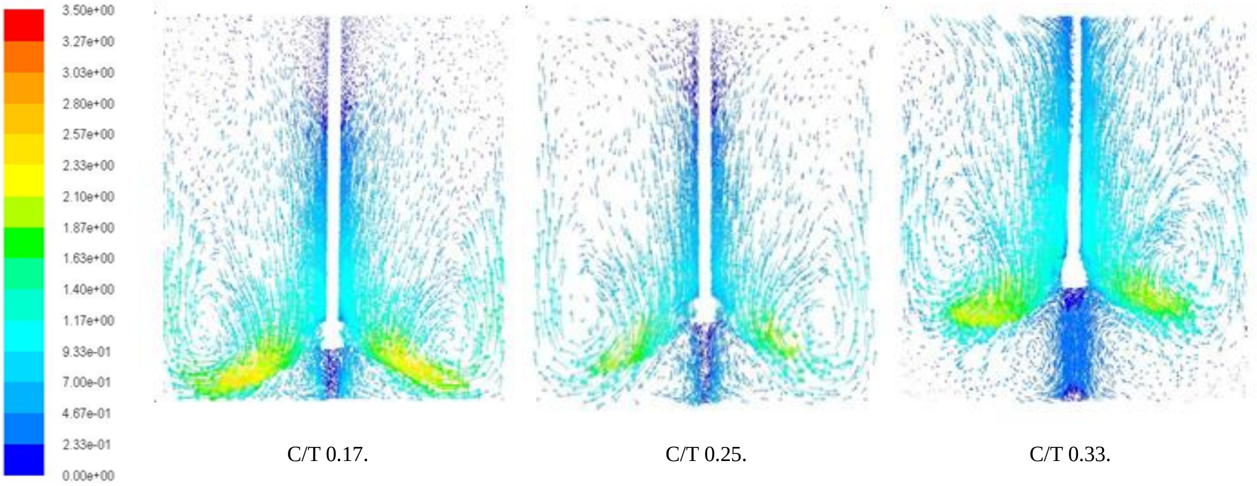


Figure. 1. Geometry of the tank and 45°



C/T 0.17. C/T 0.25. C/T 0.33.

Figure. 2. Distribution of Vector velocity with variation C/T ratio.

Just suspended speed is rotational impeller speed needed to lift solid particles from the bottom of the tank for 1s or 2s [12]. Differences of C/T on geometry of the tank make different flow profile in the tank. This will greatly affect the local velocity distribution of axial, radial and tangential. Each C/T has a dominant different zones. Data is taken on 0.53R and 0.9R radial position used to determine speed on tip of impeller end near wall of the tank.

Figure 2 also shows two macro vortex impeller on the bottom and the top of the impeller. In the section near the shaft seen many vector with high speed of 1.5 m/s so amount of solid particles will not be deposited. It is also common in the central part of axial flow vortex on impeller tip that makes solid distribution on the section would be less. The highest velocity profiles exist on the flow out of the bottom of the impeller. it is used to lift the solid particles from the bottom of the tank

The area near the base of the tank, the radial velocity component is dominant for any solid or liquid to push the particles from the bottom of the tank towards the near wall base of the tank. On 0.9R radial position, tangential velocity on all components look weak. This causes no movement at the base of the tank so a lot of particles will be deposited. To prevent that in this study used four baffles on the side wall of the tank, but it still can not be avoided particles deposited at the base of the tank.

In the beneath of the impeller are not affected by the main stream down pumping out flow of the impeller so appear vorteks areas which cause the particles spun and deposited in there. For hydrodynamic fluid with various of C/T has slightly different flow pattern. C/T 0.17 can reach a maximum speed in the area bottom of the tank that make particles easier to elevate from the bottom of the tank. vector velocity and flow rate C/T 0.33 reach the bottom of the tank very low, 0:07 m/s. It is certainly far different from the flow velocity at the C/T 0.17, 2.37 m/s and C / T 12.33, 1.63 m/s. In addition, the C/T 0.17 and C/T 0.25 minimize their small vortex in beneath the impeller which many causes particles trapped in the center of the vortex point.

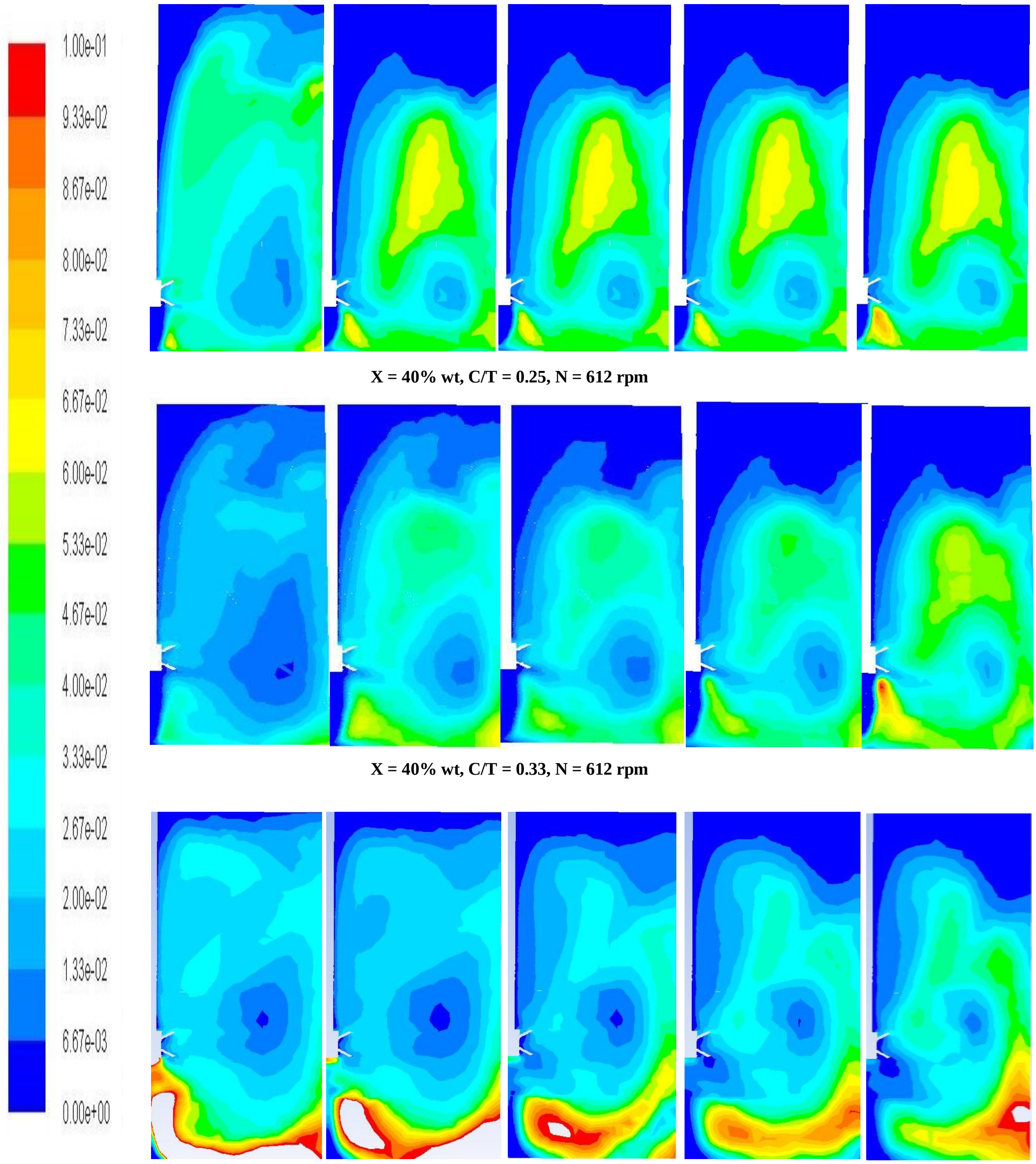
It cannot be avoided on C/T 0.33. Increased local velocity distribution is visible in the C/T 0.33. In 0.53 radial and axial positions 0.4-0.5, radial and tangential velocity become larger than the axial velocity. This is because the vortex at beneath of the impeller made axial velocity so weak thus down pumping flow on C/T 0.33 have wider angle than others. The weakening of the axial velocities undesirable because turbulence flow becomes weaker than others when it reaches the bottom of the tank and more particles deposited on the bottom of the tank

The ability to predict the spatial distribution is very important in modeling the suspension of solids in stirred tank. It impacts directly prediction of other phenomena such as mass and energy transfer. Theoretically when the rotational speed of the impeller is already at a speed of just suspended the particle will be lifted from the bottom of the tank and increase uniformity of the spatial distribution of solid components.The high concentration of solid particles, would increase the amount of solid particles in the tank, they will lead more interaction between the liquid and solid phases. It made turbulence flow more intense to eliminate the influence of vortices in the area beneath of the impeller. This also applies on the C/T 0.17 and 0.33

Particle lift-off from a bed of particles on the bottom of the vessel occurs as a result of the drag and lift forces exerted by the moving fluid. The flow near the base has been described as boundary layer flow which causes particles to be swept across the base of the vessel. Once small fillets of particles have been formed, particle lift-off is usually seen to be caused by sudden turbulent bursts originating in the turbulent bulk flow above. Fluid-particle interactions are dependent on the relative size of eddies and the particles. Eddies relatively large compared to the particles will tend to entrain them and are responsible for their suspension. Consequently, different agitator designs and configurations generate different convective flows and, thus, achieve different levels of solids suspension at the same power input. The complex nature of the flow field in a stirred vessel is such that there is no fundamental theory to describe the process of particle suspension. The CFD model does not incorporate all of the complex physics of particle sedimentation, particle lift-off, and particleparticle interactions and, hence, the difficulty in achieving accurate prediction of the local solid concentration near the base of the vessel.

Furthermore, when reviewing the homogeneity of the particle by particle diameter size, the particles with a diameter of 1 mm is more evenly distributed than that of

X = 40% wt, C/T = 0.17, N = 612 rpm



X = 40% wt, C/T = 0.25, N = 612 rpm

X = 40% wt, C/T = 0.33, N = 612 rpm

Figure. 2. Contour distribution of volume fraction with variation C/T ratio.

larger particle diameter of 1.5 mm - 3 mm. This is because the smaller particle diameter (dp 1 mm) has a higher fluidity and more easily carried away from the bottom of the tank with smaller a force than the particles with a larger diameter. It can be viewed on any variable studied contour of Figure 3 distribution of solids concentration distribution of particles dp 1 mm more evenly.

There are three interesting points to observed. First on the area beneath of the impeller, in this area there are many particles deposited because the particles are not carried away by the current pumping down. Especially for C/T 0:33, where the circulation flow formed on the bottom of the impeller causes down pumping flow to the bottom of the tank is not perfect.

The second part at the base of the tank. Many Solid particles deposited at the base of the tank because there is a dead zone that made particles cannot carried away and deposited in this area. This condition will be greatly minimized on C/T 0.17 because at the base end of the tank area of turbulence is still very dominant.

Third, the area at the center of the macro vortex is formed. In this area, number of solid particle decreased compared with the area around it. This is due to the centrifugal force which resulted solid particles thrown out of center vortek. Overall, the agreement with experimental data is good, except near the base of the vessel and around the impeller. The solid distributions are largely overestimated near the base.

The data in this table to inform you that at the same speed C/T 0.25 has greater power needs compared with C/T 0.17. This is because the C/T 0.17 closer to the bottom of the tank and need more little energy to form a larger looping flow.

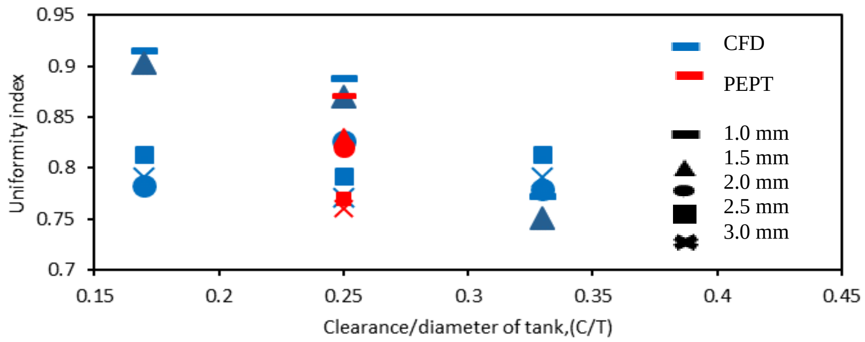


Figure. 3. Uniformiti index with variation C/T ratio.

The last factor that determines the mixing process is the uniformity index of the mixture. This uniformity index is the value that interpret contour profile distribution and volume fraction of the mass distribution profile in the whole volume of the tank Where uniformity index values will be better when approaching

$$\xi = 1 \; or \; \sigma^2 = 0.$$

One of CFD simulation results on polydisperse solid-liquid mixing is uniformity index. Uniformity index values can reinforce data from contour volume fraction of solid particles distribution profile and solid mass distribution profile in the whole volume of the tank. Figure 3. compares uniformity index and the difference ratio of C/T.

Uniformity index values are slightly different for all variabels because C/T ratio in geometry of tank given differences of flow patterns, thus the axial, radial and tangential velocities at every fifty cells are also different. Especially on the axial position of 0 - 0.3 H, in this area for C/T 0.33 axial velocity of down pumping flow cannot reach the the bottom of the tank. This was occurred due to the vortex beneath the impeller zone prevented turbulent flow from impeller. The vortex produce a bulk of solid particle deposited in the bottom of the tank. As consequences the uniformity index C/T 0.33 becomes lower than others. Impeller speed used in this simulation is Njs for C/T 0.25. When it is applied in C/T 0.33, N<Njs, velocity vector of turbulent flow cannot remove solid particle from bottom of the tank.

TABLE 1.
POWER CONSUMPTION OF VARIOUS TYPE OF RATIO C/T

C/T	N (rpm)	momen	Power (watt)
0.17	612	-3.92	251.18
0.25	612	-4.08	261.38
0.33	612	-4.23	270.65

Estimation of the power consumed on stirred tank is very important as a variable for the industry design. Errors in estimation power consumption at lab scale will be great when scaled up to an industrial scale. Therefore, power consumption estimation and measurement of the dimensions of a stirred tank used is very important.

However this is different on C/T 0.33. C/T 0.33 makes the vortex at the beneath of the impeller. This causes the vortex arising power needed becomes bigger though baffle trailing edge is greater aspect in influencing the value of the power required in mixing a solid suspension system. This is because the impeller further distance from the bottom of the tank then it will require greater power to remove particles from the bottom of the tank.

This resulted in a load factor of the impeller to stir the mixture becomes larger so needs a greater power. A greater burden of the increase in concentration of suspended solids cause the viscosity and density of the mixture becomes larger so the viscosity in the mixing process became larger. The viscosity is greater this cause needs a larger power.

IV. CONCLUSION

The objective of this study was to propose and test three parameters C/T ratio on geometry of the tank in high solid loading polydisperse solid liquid mixing. The influence of the ratio of C/T gave differences flow patterns of the hydrodynamic flow in the tank. The spatial distribution of the solid phase is well predicted except close to the base of the vessel and around the impeller where it is largely overestimated. Generally, the smaller particles are better distributed than the larger ones except when N> NJS. it becomes difficult to reach 100% homogeneity, as it is hard to distribute particles below the impeller and near the free surface That C/T 0.17 have the highest uniformity index was performed for 0.88. The rotational speed of the impeller affected homogeneity. The speed N> NJS generated greater velocity vector when reached the bottom the tank resulting uniformity index about 0.8409 and need a bit power consumption about 251.18 watt

REFERENCES

[1] Virdung T. dan Rasmuson A. (2007), " Measurement of Continuous Phase Velocities in Solid-Liquid Flow at Elevated Concentrations in A Stirred Vessel Using LDV". *Chemical Engineering Research and Design,* Vol 85 (A2) hal. 193-200.

[2] Tamburini A., Cipollina A., Micale G., Brucato A., Ciofalo M. (2011)," CFD Simulation of Dense Solid-Liquid Suspensions in Baffled Stirred Tank: Prediction of Suspensions Curves". *Chemical Engineering Journal.* Vol. 178 hal. 324-341

[3] Li Liu dan Barigou M. (2015)," Lagrangian Partickel in Mechanically Agitated Polydispersed Suspensions: Multi-Component Hydrodynamics And Spatial Distribution". International Journal of Multiphase Flow. Vol. 73, hal. 80-89.

[4] Guida A., Alvin W., Nienow, Barigou M. (2010), "PEPT Measurements of Solid-Liquid Flow Field and Spatial Phase Distribution in Concentrated Monodisperse Stirred Suspensions", *Chemical Engineering Science,* No. 65, hal.1905-1914.

[5] Ayranci I., Machado M.,Madej A.,Derksen J., Nobes D., Kresta S. (2012), " Effect of Geometry on Mechanisms for Off-Bottom Solids Suspension in a [Stirred Tank". Chemical Engineering Science. Vol 79, hal 163

[6] Ayranci I., dan Kresta S. (2014) " Critical Analysis of Zwietering Correlation for Solids Suspension in Stirred Tanks". Chemical Engineering Research and Design. Vol. 92, hal. 413-422

[7] Tamburini A., Cipollina A., Micale G., Brucato A., Ciofalo M. (2013), "CFD Simulation of Dense Solid-Liquid Suspensions in Baffled Stirred Tank: Prediction of of Solid Particle Distribution". Chemical Engineering Journal. Vol. 223 hal. 875-890

[8] Tamburini A., Cipollina A., Micale G., Brucato A., Ciofalo M. (2012)," CFD Simulation of Dense Solid-Liquid Suspensions in Baffled Stirred Tank: Prediction of The Minimum Impeller Speed For Complete Suspension". Chemical Engineering Journal. Vol.194, hal. 234-255.

[9] Brucato A., Grisafi F., Montante G.(1998)," Particle Drag Coeficient in Turbulent Fluids". Chemical Engineering Science, Vol. 53, No.18, hal. 3295-3314.

[10] Altway A., Setyawan H., Margono, Winardi S. (2001), " Effect of Particle Size on Simulation of Three-Dimensional Solid Dispersion in Stirred Tank". *Chemical Engineering Research and Design.*Vol 79, No. A8, hal. 1011-1016

[11] Li Liu dan Barigou M. (2013)." Numerical Modelling of Veocity Field and Phase Distribution in Dense Monodispersed Solid-Liquid Suspensions Under Regimes of Agitation: CFD and Experiment". *International Journal of Multiphase Flow*. Vol. 60, hal. 100-119.