Turbulent flow in a vessel agitated by side entering inclined blade turbine with different diameter using CFD simulation

N N Fathonah¹, T Nurtono¹, Kusdianto¹, and S Winardi*¹

¹ Laboratory of Fluid Mechanic and Mixing, Department of Chemical Engineering, Institut Teknologi Sepuluh Noemper
Kampus ITS Keputih, Surabaya Indonesia 60111

*Email: swinardi@chem-eng.its.ac.id

Abstract. Single phase turbulent flow in a vessel agitated by side entering inclined blade turbine has simulated using CFD. The aim of this work is to identify the hydrodynamic characteristics of a model vessel, which geometrical configuration is adopted at industrial scale. The laboratory scale model vessel is a flat bottomed cylindrical tank agitated by side entering 4-blade inclined blade turbine with impeller rotational speed N=100-400 rpm. The effect of the impeller diameter on fluid flow pattern has been investigated. The fluid flow patterns in a vessel is essentially characterized by the phenomena of macro-instabilities, i.e. the flow patterns change with large scale in space and low frequency. The intensity of fluid flow in the tank increase with the increase of impeller rotational speed from 100, 200, 300, and 400 rpm. It was accompanied by shifting the position of the core of circulation flow away from impeller discharge stream and approached the front of the tank wall. The intensity of fluid flow in the vessel increase with the increase of the impeller diameter from d=3 cm to d=4 cm.

1. Introduction
One of the mixer widely used is agitated tank by side entering impeller which an agitator is mounted horizontally on the sides of the tank wall, near the bottom of the tank. The main advantages of side entering agitated vessel includes large volume capacity, low capital and operating cost, and simple installation as no mounting support on the top of the tank is needed [1,2]. One of the main drawbacks of side entering agitated vessel is the poorer mixing performance with respect to baffled vessels, due to the smaller pumping rate generated by the impeller. As a consequence, limited attention has been devoted so far to the fundamental mixing characterization of these mixers. Nevertheless, vessel with side entering mixer are gaining interest as they may provide significant advantages in a number of applications including biochemical, oil and pulp and paper processes.

Mixing performance in side entering agitated vessel can be studied experimental and computational approaches. Over the past years a number of works have been carried out to investigate mixing performance by side entering impeller using experimental and simulation approaches [3,4,5,6]. However, almost of all simulation studies has been conducted for large-scale mixing system without quantitative validation with experimental data. This is because due to experimental work is a laborious and expensive process when carried out in large scale side entering agitated vessel. In this study, therefore, the hydrodynamics and characteristics of side entering agitated vessel will be studied by
numerical simulations using CFD software package ANSYS Fluent 17.1. Impeller used are 4-bladed inclined blade turbine (IBT) with diameter 3 and 4 cm. The influence of impeller diameter on turbulent fluid flow will investigated. The simulation results will be validated with experimental data.

2. Geometry of mixing system
A side entering agitated vessel is used in this work includes the flat bottomed cylindrical vessel \((D=40\, \text{cm})\), 4-blade IBT with diameter \(d=3\) and \(4\, \text{cm}\) \((d/D=0.075\) and 0.1). The length of the impeller shaft \((s)\) and the distance of impeller shaft to the tank bottom \((h)\), are equal to the impeller diameter \((d)\) respectively. The horizontal angle of impeller shaft \((\beta)\) is constant at 0o. The geometry and dimension of the vessel and impeller are shown in figures 1 and 2, schematically.

3. Model development
3.1. Assumption
The assumption of operating condition for this system is set as fluid flow is three dimensional and unsteady state. Working fluid in vessel is incompressible, isothermal, and Newtonian fluid. The model is single phase without accounting for multiphase interactions.

3.2. Description of computational model
The computational model is based on solving the conservation equations which describe the flow in the tank [7]. For the mean flow of an incompressible liquid, the equation of mass conservation is expressed as,

\[
\frac{\partial U_i}{\partial x_i} = 0 \quad (1)
\]

\(U_i\) denotes the mean velocity in direction \(x_i\). Under steady-state laminar flow conditions the momentum balance is given by:

\[
\rho U_i \frac{\partial U_i}{\partial x_i} = \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i \quad (2)
\]

The stress tensor \(\tau_{ij}\) is given by:

\[
\tau_{ij} = \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (3)
\]

When the flow is turbulent an additional term appears in the momentum balance for the mean flow:

\[
\rho U_i \frac{\partial u_i}{\partial x_i} = -\frac{\partial p}{\partial x_i} + \frac{\partial \left( \tau_{ij} + \rho u_i u_j \right)}{\partial x_j} + \rho g_i \quad (4)
\]

\(u_i\) is the fluctuating, turbulent, velocity component in direction \(x_i\). The cross correlation term denotes the Reynolds stress tensor. Although analytical equations for the Reynolds stress tensor can be
derived, these equations again contain higher order cross correlation terms, and the set of equations is not closed. For modeling the Reynolds stresses various so-called turbulence models are available. An often used model is the $k$-$\varepsilon$ model which models the Reynolds stresses as being proportional to gradients in the mean flow.

$$\rho \overline{u_i u_j} = -\mu_t \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) + \frac{2}{3} \delta_{ij} k$$

(5)

The turbulent viscosity $\mu_t$ is given by:

$$\mu_t = c_\mu \rho \frac{k^2}{\varepsilon}$$

(6)

The model equations for the conservation of the turbulent kinetic energy density $k$ and the turbulent energy dissipation rate density $\varepsilon$ are:

$$U_k \frac{\partial k}{\partial x_k} = \frac{\partial}{\partial x_k} \left( (v + \nu_{\varepsilon}) \frac{\partial k}{\partial x_k} \right) + P_k - \varepsilon$$

(7)

$$U_k \frac{\partial \varepsilon}{\partial x_k} = \frac{\partial}{\partial x_k} \left( (v + \nu_{\varepsilon}) \frac{\partial \varepsilon}{\partial x_k} \right) + C_{\varepsilon} \frac{\varepsilon}{k} P_k - \varepsilon$$

(8)

Equation (5) for the Reynolds stresses is similar to equation (3) for the laminar shear stresses. The turbulent viscosity $\mu_t$ plays the same role as the molecular viscosity $\mu$. As a result the form of the turbulent momentum equations is similar to the form of the laminar momentum equation except that $\mu$ is replaced by an effective viscosity $\mu_{eff}$:

$$\mu_{eff} = (\mu + \mu_t)$$

(9)

3.3. CFD simulation

The simulation were performed using software ANSYS® ver. 17.1 that included Design Modeler as geometry modeler, ANSYS Meshing as grid generator, and ANSYS Fluent as the CFD solver. The simulations were started from steady-state condition using the combination of $k$-$\varepsilon$ as turbulence model and multiple reference frame (MRF) as impeller motion model. The calculation domain were divided into two zones that are depicted in Figure 3 below.

![Figure 3. The calculation domain with two zones](image)

The display of the flow field vector in each plane inside the agitated vessel was recorded every time step to make flow pattern animations. The flow pattern recognition and characterization were obtained from vertical and horizontal planes passing through the impeller shaft.

4. Result and discussion

4.1. Influence of impeller rotational speed

Fluid flow patterns in side entering agitated vessel is essentially characterized by the phenomena of macro-instabilities, i.e. the flow patterns change in space and time with large scale. These phenomena always accompanied with strong velocity fluctuations in the entire of the vessel. In this study, for
simplification, changes in fluid flow patterns only observed in a vertical and horizontal plane passing through the impeller shaft. This phenomena of macro-instabilities shown in figure 4 below. This flow patterns are recorded in 60, 100, 120, 300, 400 and 600 second. From these figures, it can be seen that macro-instabilities phenomena occur throughout the mixing process runs at rotational speed 300 rpm and agitated with IBT 3 cm of diameter. Flow patterns formed are different every time.

Figure 4. Macro-instabilities phenomena in tank stirred by IBT with diameter of $d=3$ cm

Effect of impeller rotational speed to fluid flow pattern in tank stirred by IBT with diameter of $d=3$ cm are shown in Figure 5. From side view observation, at impeller rotational speed $N=100$ rpm, discharge stream from impeller will flow toward the wall of tank, then most discharge stream impingement wall, and along the wall rises up towards the liquid surface. Another part of the discharge stream turned upward to the liquid surface then flow return back to the impeller in short time. It can be seen that the large circulation flow is formed with the core located close to the impeller discharge stream. On the right side of the area near the liquid surface formed a stagnant region. From the top view observation, it can be found that the circulation flow also occurs horizontally on the area on the left side and the right side of the impeller.

When the impeller rotational speed is increased in the range $N=200-400$ rpm, from the side view observation shows that the intensity of circulation flow increase with the increase of rotational speed of the impeller. It was accompanied by shifting the position of the core of circulation flow away from impeller discharge stream and approached the front of the tank wall. From the top view observation, it
can be seen that the circulation flow position shifted away from the impeller area and approached the tank wall.

Figure 5. Fluid flow patterns in tank stirred by IBT with $d=3$ cm in different impeller rotational speed
Figure 6. Fluid flow patterns in tank stirred by IBT with $d=4$ cm in different impeller rotational speed

Effect of the impeller speed to the fluid flow patterns in the tank agitated by IBT with a diameter $d=4$ cm are shown in Figure 6. From the side view and top view observations, it can be seen that the circulation flow intensity increase with the increase in impeller rotational speed from $N=100-400$ rpm as seen with impeller diameter of 3 cm. This was indicated by a shift in the position of the core of circulation flow, which is further away from the impeller discharge stream and approached the tank wall in front of them.
4.2. Influence of impeller diameter

As shown in Figures 5 and 6 above, the fluid flow patterns in the tank produced by IBT showed the same tendency when the impeller rotational speed increases. However, the intensity of fluid flow increases with the increase of the impeller diameter from \(d=3\) cm to \(d=4\) cm as shown in Figure 6.

![Fluid flow patterns](image)

Figure 7. Influence of impeller diameter to fluid flow pattern in side entering agitated vessel

At a rotational speed \(N=100\) rpm, the intensity of the fluid flow generated by the impeller \(d=4\) cm is relatively large compared with those produced by the impeller with \(d=3\) cm. This is shown by the size
of the core of a larger circulation flow and stagnation region in the top right corner regions near the surface of the liquid does not appear.

4.3. Comparison with experimental data
Computational data needs to be validated with experimental data. Because the simulation results depend on several factors such as geometry of the system used, meshing structure, definition and properties of the fluid used, the turbulence model and impeller motion model selected [8].

![Simulation and Experiment Comparison](image)

**Figure 8.** Flow pattern comparison between simulation and experimental results
Figure 8 above shows the comparison of flow pattern produced by simulation and experimental results. Flow patterns produced by axial flow impeller has main characteristic one loop circulation. Additional flow pattern might arise due to the interaction of the structures and the vessel wall and have been explained in macro instabilities phenomena section 4.1 [9]. From this figure we can conclude that simulation results has a good agreement with experimental data, that can be seen from flow pattern produced tend to be similar.

5. Conclusion
The fluid flow patterns in the vessel equipped with side entering 4-blade IBT is essentially characterized by the phenomena of macro-instabilities, i.e. the flow patterns change with large scale in space and low frequency. The intensity of fluid flow in the tank with IBT increase with the increase of impeller rotational speed from 100, 200, 300, 400 rpm. It was accompanied by shifting the core position of circulation flow away from impeller discharge stream and approached the front of the tank wall. Also the intensity of fluid flow increase with the increase of impeller diameter from d=3 cm to d=4 cm. Simulation results has a good agreement with experimental data, that can be seen from flow pattern produced tend to be similar.

Acknowledgment
The authors are grateful for financial support provided by a grant from Directorate of Research and Public Service, Directorate General of Research Strengthening and Development, Ministry of Research, Technology and Higher Education of the Republic of Indonesia.

References
[1] Vincent W. Uhl Joseph B. Gray 1966 Mixing Theory and Practice I (New York: Academic Press)
[2] James W. Oldshue 1983 Fluid Mixing Technology (New York: McGraw-Hill Publications Co.)
[3] Johannes A. Wesselingh 1975 Mixing of Liquids in Cylindrical Storage Tanks With Side-Entering Propellers (Chemical Engineering Science)
[4] Klausdieter K. 1984 Suspension by Side Entering Agitators (Chemical Engineering Process) vol. 18 p 233
[5] Asghar A. D. Masoud R. 2004 CFD Simulation of Homogenization in Large Scale Crude Oil Storage Tanks (Petroleum Science & Engineering/ Elsevier) vol. 43 p 151-161
[6] Masoud R. 2005 The Effect of Impeller Layout on Mixing Time in Large-Scale Crude Oil Storage Tank (Petroleum Science & Engineering/ Elsevier) vol. 48 p 161-170
[7] H K Versteeg W. Malalasekera 1995 An Introduction to Computational Fluid Dynamics: The Finite Volume Method (London: Prentice Hall)
[8] C. Gomez C.P.J. Benington F. Taghipour 2010 Investigation of the Flow Field in a Rectangular Vessel Equipped with a Side-entering Agitator (Journal of Fluid engineering/ ASME) vol. 132
[9] Jaime S.E Fariborz T. 2015 Computational Simulation of Mixing Flow of Shear Thinning non-Newtonian Fluids with Various Impellers in a Stirred Tank (Chemical Engineering and Processing: Process Intensification/ Elsevier)