Optimization of impeller design for stirred tank using computational fluid dynamics

K Priyadi¹, Chin-Tu Lu², H Sutanto¹

¹Department of Mechanical Engineering, Atma Jaya Catholic University, Jakarta 12930, Indonesia
²Department of Mechanical Engineering, Southern Taiwan University of Science and Technology, Tainan 710, Taiwan ROC

Corresponding author: hadi.sutanto@atmajaya.ac.id

Abstract. Stirred tanks are widely used for mixing to pseudo-plastic fluids in chemical, food and many processing industries. Impeller stirred tanks play a very important role throughout process industries. Design of such systems and the effective control of the relevant processes require an understanding and prediction of the flow characteristics. One technique to develop high efficiency of this fluid production such as bioethanol is to reduce the stagnant or dead zones formation phenomena inside the stirred tank using computational fluid dynamics. The results have shown that the new impeller design as dependent variables were able to eliminate the dead zone formation related with the independent variables such as power number $P_0$, pumping number $N_q$, wall shear stress $\tau$, mixing time $\tau$, and average effective viscosity $\mu$.

1. Introduction

Stirred tanks are among the most common process equipment used in chemical and other process industries since their good mixing ability and the characteristics for scale-up. Stirred tanks can be classified in different ways. Firstly, they can be sorted by the way of operation of the units, such as batch, semi batch or continuous. Secondly, by thermal operation as isotherm or adiabatic. Thirdly, by the type of applied impeller, such as turbine, jet, blade, etc. Normally in a stirred tank with the centrally positioned impellers, a rotating motion with a pair of vortices behind each blade, one above and one below the disk is generated. The flow generated in the stirred tanks are predominantly turbulent due to high impeller rotation speeds to achieve necessary process conditions [1].

The most commonly used type of stirred tanks is a fluid stirring reactor type. The problem that often arises in the fluid stirrer reactor is the emergence of stagnant or dead zone or cavern formation phenomenon as shown in Figure 1. In the non-Newtonian fluidizing process, the viscosity value of the fluids are vary depending on the shear stress occurring in the fluid. The higher the shear stress occurs, the lower the viscosity of the fluid as related with the shear-thinning in pseudo-plastic property. Hence, the fluid located close to the slats tend to have lower viscosity, while the fluid away from the blades has a higher viscosity value resulting in a stagnant zone or cavern formation.

The stirred units have many parts, therefore experimental information or data during the production phase are difficult to obtain. Hence, to build an accurate and validated model of the units can require much time which cannot be afford nowadays. Experiments in laboratory scale can support the modeling process but the scale-up of the tanks to industrial scale can be a problem because the developed flow pattern of scale-up levels can be significantly different. Using CFD or Computational Fluid Dynamics, the operation of equipment can be studied in detail without disrupting the production
process. The CFD approach has the possibility to model the entire geometries in three dimensions even multiphase systems [2]. In addition, CFD simulations applicable not only in development [3], safe operation [4] and modeling the dynamic behavior of the experimental system, but also after integration of the mathematical description of considered parameters it can be describe mass and energy with related to transport process [5]. CFD can lead to a better understanding of the behavior of stirred vessel and macro mixing phenomenon in three-dimensional turbulent flow [6].

The purpose of this study is to provide a solution to the stagnant or dead zone or cavern formation phenomenon that occurs in the fermentation stirring process in the fluid stirrer reactor to produce bio-ethanol that caused by the weak propulsion of the blade. A new blade design for a pseudo-plastic type of fluid behavior has to produce a higher propulsion or pumping rate and also maintaining the stability of the power required to rotate the blades, using CFD software with ANSYS Fluent.

2. Stirred Tank Reactor
Stirred tank as biological reactor or bioreactor like a fermenter, which is producing a product, using the work of micro-organisms or enzymes studied by the industrial microbiology domain. Bioreactors always set the condition of their tanks in a controlled environment therefore the micro-organisms can live and work. Hence, there is a classification of bioreactors based on the needs of the desired results. The classification of the bioreactor according to the moisture content contained in the tank can be divided into submerged-culture fermentation and solid-state fermentation. For stirred tank reactor it is categorized into submerged-culture fermentation.

The power number $Po$ to turn the wheel of fluid is:

$$P_0 = \frac{p}{(\rho x N^3 x D^5)}$$

and $N$ is the impeller angular velocity and $D$ is the impeller diameter. This equation becomes the basis for the power or newton number ($Po$) equation, which goes into the list of dimensionless parameter groups [8].

The pumping number $Nq$ is obtained based on the fluid flowing from the blade to the outer blade of the region at any time, which can be represented by the amount of flow flowing through the surface area. The mass flow out through the surface area can be calculated by the following equation,

$$Q = \frac{\dot{m}_{axial} + \dot{m}_{radial}}{\rho}$$

where, $\dot{m}$ is the mass flow rate. Hence the pumping number is then obtained by using the following equation,
The wall shear stress $\tau$ is obtained related with viscosity that can be modelled with fluids consisting of several layers following both the plates on the top and bottom without slippage between them. The expression of shear stress for Newtonian fluids with the variable of shear rate is \[ \tau = \mu x \dot{\gamma} \]

In the fluid stirrer, the mean specific energy dissipation rate can be calculated by the equation, where all energy occurring in the fluid, as a result of the impulse movement in the vessel, changed or dissipated into heat energy,

\[ \bar{\varepsilon}_T = \frac{P}{\rho xV} = \frac{P_0 x N^3 x D^5}{V} \]

The predictable mixing time or the time required of a mixture to reach the level of homogenization, can be calculated with the following equation,

\[ \tau_n = \left( \frac{\dot{\gamma}}{\bar{\varepsilon}_T} \right)^{1/2} \]

where, $\nu$ is the kinematic viscosity of the fluid and $V$ is the total volume of liquid inside the stirred tank. Mixing time has been used as a key parameter for assessing the performance of a mixing system. For Newtonian fluid, this phenomenon can occur due to the vertical stress drop $\sigma_v$ occurring in the fluid or the horizontal fluid stress increase $\sigma_h$ in the stagnant zones area.

The flow density as the flow decreases, causing the increase of the horizontal stress at a point from the flow to the stagnant zone. Due to the velocity difference at that point the formation of wall shear stress and the vertical stress value in the stagnant area increases that result to a periodical slip. On the same principle for non-Newtonian fluids, that there is a velocity difference between flowing fluids and stagnant zones or cavern formation. In the fluid stirrer reactor, the velocity difference due to the increased concentration of shear stress at the center of the blade region will result for the viscosity difference between the fluid that flows and at the stagnant zones.

### 3. Research Methodology

The stirred tank geometry represents a typical “standard” configuration where the tank diameter $T$ is 0.8 m and other dimensions may be related to $T$ as follows,

- Fluid height $H_l = T$
- Impeller diameter $D = 0.4 T$
- Impeller clearance $C = T/3$
- Blade thickness = 0.001 m
- Tank bottom type = ASME 6% bottom
- Shaft diameter = 0.03 m
- Angular velocity = 60 rpm

This study uses data validation for power number and pumping number from experimental results by Lane and Koh [9, 10] as $Po = 4.67$ and $Nq = 0.73$. Data for effective viscosity $\mu = 0.3$ kg/ms and specific gravity 1.21 from reference [8]. For shear thinning non Newtonian fluids, where the viscosity decrease with increasing shear rate, the data obtained from the study of Fraiha et.al [8].

The standards used to design the geometry of the new blade design is that it should increase the pumping number and shear rate generated by the blade since they allow for a solution to reduce the stagnant or dead zone. The new design also has additional effect to stabilize power consumption of the stirred tank. Numerical simulation set up based on the new design of impeller was specified for turbulence using $k-\epsilon$ model with standard wall function and the simulation run as a transient problem with standard initial condition of absolute reference frame zero velocity at all grid nodes [10][11].
4. Results and Discussions
The study on development of stirred tank reactor did an analysis of causes and effects of the phenomenon from the mechanical approach of blade geometry using numerical analysis. Results from numerical simulation have shown the percentage of error for power number $Po$ and pumping number $Nq$ are small, as seen in Table 1.

| VARIABLES       | SYMBOL | EXPERIMENTAL RESULT | NUMERICAL RESULT | ERROR [%] |
|-----------------|--------|---------------------|------------------|-----------|
| Power Number    | $Po$   | 4.67                | 4.35             | 7.40      |
| Pumping Number  | $Nq$   | 0.73                | 0.70             | 3.94      |

After analyzing a correlation between the angle of attack to the performance of the blade, the $30^\circ$ angle of blade is selected since the angle of blade has the ratio of $Nq$ to $Po$ to the second highest while maintaining the stable wall shear stress. The design of the new blade has a basic principle that the larger surface area of blade to drive fluids, the greater the resulting $Nq$, but the $Po$ value increases. Hence the number of blades was reduced to 3 pieces or AM 30 (Axial-flow Mix-solidity) with addition to two wings or flaps on the top and bottom of blades (Figure 2). The shape of the blade resembles hydrofoil in order to lower $Po$ and increase the number of surfaces that push the fluid to increase $Nq$. Figure 3 shows the relationship between impeller type and the ratio of $Nq/Po$ and average effective viscosity.

![ISO view of new blade design 3 AM 30](image)

**Figure 2.** ISO view of new blade design 3 AM 30

![Graph of impeller type vs (a) Nq/Po and (b) average effective viscosity](image)

**Figure 3.** Graph of impeller type vs (a) $Nq/Po$ and (b) average effective viscosity
The new blade design 3 AM 30 has the ratio of $Nq/Po$ is 0.229 but it has the lowest effective viscosity which is useful in reducing the stagnant or dead zone phenomenon in stirred tank. This phenomenon is possible to occur only in a fluid stirrer reactor for non-Newtonian fluids aggregation processes. The fluid viscosity may change depend on the amount of shear stress that occurs in the fluids, as seen in Figure 4.

![Figure 4. Average effective viscosity vs flow time](image)

The stagnant or dead zone phenomenon has a negative impact on the fermentation process in the stirred tank reactor as it decreases the homogeneity level of the fermentation mixture.

A comparison between two contours for the tendency to form a cavern formation is shown in figure 5, where the blades 3 AM 30 has areas of less effective viscosity than stirring results using blades 4 PBT 45 type. The average effective viscosity of blade design 3 AM 30 is 0.026 kg/ms which is smaller than the average effective viscosity of blade 4 PBT 45 as 2.889 kg/ms. The maximum effective viscosity of blade design 3 AM 30 is 43.173 which is smaller than the maximum effective viscosity of blade 4 PBT 45 as 75.838 kg/ms.

![Figure 5. Contour of effective viscosity in cross section of stirred tank for impeller type (a) 4 PBT 45 and (b) 3 AM 30](image)

5. Conclusions

CFD analysis on the optimization of high-efficiency stirred tank reactor using mechanical approach from blades can minimize or eliminate stagnant or dead one or cavern formation using a new design.
blades. Independent variables such as power number, pumping number and wall shear stress are also compatible variables to develop high efficiency stirred tank. Some dependent variables such as impeller design and the angle of attack have a correlation with high efficiency stirred tank reactor. The new design of the impeller to produce high efficiency stirred tank can be developed to produce high efficiency stirred tank can be developed by reducing the number of blades, increasing the surface area of the impeller blade and make it similar to hydrofoil.

The new impeller design as 3 AM 30 is able to reduce the power consumption but maintain the pumping number at the similar value as 4 PBT 45. Optimization of impeller design to 3 AM 30 can prevent the phenomenon of stagnant or dead zone in stirred tank reactor with “standard” configuration of $T$ equal to 0.8 m.

References
[1] Joshi JB, Nere NK, Rane CV, Murthy BN, Mathpati CS, Patwardhan AW and Ranade VL 2011 CFD simulation of stirred tanks: Comparison of turbulence models Part I : Radial flow impellers Can. J. Chem Eng 89 (1) pp 23-82
[2] Tamburini A, Cipollina A, Micale G, Brucato A and Ciofalo M 2013 CFD prediction of solid particle distribution in baffled stirred vessels under partial to complete suspension conditions Chemical Eng Transaction 32 pp 1447-1452
[3] Wei H 2010 Computer-aided design and scale-up of crystallization processes: integrating approaches and case studies Chemical Eng Research and Design 88 (10) pp 1377-1380
[4] Rudniak L, Machniewski PM, Milewska A and Molga E 2004 CFD modelling of stirred tank chemical reactors_ homogenuous and heterogeneous reaction systems Chemical Engineering Science 59 (22-23) pp 5233-5239
[5] Sommerfield M and Decker S 2004 State of the art and future trends in CFD simulation of stirred vessels hydrodynamics Chem. Eng. Technology 27, pp 215-224
[6] Ranade VV, Tayalia Y and Krishnan H 2015 CFD prediction of flow near impeller blades in baffled stirred vessels: Assessment of computational shapshot approach Chemical Engineering Communications 189:7 pp 895-922
[7] Edward PL, Atiemo-Obeng VA and Kresta SM 2004 Handbook of Industrial Mixing-Science and Practice John Wiley and Sons, NJ
[8] Fraiha M, Biagi JD and Ferraz CO 2011 Rheological behavior of corn and soy mix as feed ingredients, Ciencia e Tecnologia de Alimentos 31 (1), Campinas pp 129-134
[9] Lane GL and Koh PLT 1997 CFD simulation of Rushton turbine in a baffled tank Int. Conf. on CFD in Mineral & Metal Processing and Power Generations, CSIRO Division of Minerals, Clayton South, pp 377-386
[10] Lane GL, Schwarz MP and Evans GM 2015 Comparison of CFD methods for modeling of stirred tanks in Proceedings of 10th European Conference on Mixing, Elsevier, pp 273-280
[11] ANSYS Fluent 2013 Theory Guide 15 Ansys, Inc., Canonsburg, PA
[12] Gerasimov A 2006 Modeling Turbulent Flows with FLUENT Ansys, Inc., PA.