CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

Effect of Impeller's Blade Number on The Performance of Mixing Flow in Stirred Tank using CFD Simulation Method

Ahmad Faiq Baba[1], Nor Afzanizam Samiran[1,2,*], Razlin Abd Rashid[1,2], Izuan Amin Ishak[1], Zuliazura Mohd Salleh[1,2], Rais Hanizam Madon[1,2], Muhammad Suhail Sahul Hamid[3]

[1] Department of Mechanical Engineering Technology, Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, Pagoh Johor, 84600, Malaysia
[2] Integrated Engineering Simulation and Design (IESD) Focus Group, Universiti Tun Hussein Onn Malaysia, Pagoh Higher Education Hub, 84600 Pagoh, Muar, Johor, Malaysia
[3] Els Energy and Lab Solutions Sdn Bhd No.11A Tingkat Merpati Dua, Taman Transkrian 14300 Nibong Tebal, Pulau Pinang, Malaysia

ABSTRACT

Article history:
Received 14 March 2022
Received in revised form 29 May 2022
Accepted 30 May 2022
Available online 31 May 2022

The component of an impeller in stirred tank plays an important role in the mixing process for a wide range of industries. The present study conducted the analysis of impeller design specifically on the number of blades to investigate the flow performance in the stirred tank. The analysis was also performed to identify the location of the particle flow dead zone at the bottom of the tank. The grid development and fluid flow study were accomplished using the CFD simulation method via the Ansys Fluent software package. The impeller design with the arrangement of 3,4 and 5 blades number was involved in this study. The rotational speed of the blade was set constant at 90 rpm. The flow characteristic at the vertical position of a tank at 0.01, 0.1, 0.2 and 0.3 m has been investigated. The general governing equation and solution approach used in this simulation were also presented in this paper. The results showed that 5 blades impeller produced a broader region of high-velocity magnitude distribution compared to the 3 and 4 blades. Hence, 5 blades impeller also relatively produced a higher distribution of velocity magnitude compared to the 3 and 4 blades impeller, typically in the upper region of the tank. However, the effect of a number of blades seemed not significant in the region close to the rotating impeller.

Keywords:
Stirred tank; Impeller; Number of blades; velocity magnitude, CFD

1. Introduction

Agitation in cylindrical tanks or stirred tank is a common practice in a variety of sectors, including the food, polymer, paint, pharmaceutical, and petroleum industries. The optimal design of a stirred tank is determined by the required production rate and product characteristics [1, 2]. In mixing systems, the hydrodynamic generated by the impeller has a significant impact on the ultimate product quality [3]. The primary goals of mixing are to increase mass and heat transmission, as well as to create a homogenized mixture to reduce particle settling at the tank's bottom [4].

* *Corresponding author.*
E-mail address: afzanizam@uthm.edu.my (Nor Afzanizam Samiran)

https://doi.org/10.37934/cfdl.14.5.3342

Many factors influence mixing efficiency, but one of the most important is impeller form [5]. The number and form of impellers, as well as the position, shape, and flow rate of a sparger, have a significant impact on the flow parameters in the reactor [6]. Due to the complexity of the three-dimensional hydrodynamics inside the agitated vessels, many problems were yet not addressed. One of the main problems often encountered is how to reduce the power required for achieving the mixing operation. The increase of the impeller's blade curvature was typically capable to greatly reduces the power consumption, for the same blade diameter and the same Reynolds number [7]. Characterization and understanding of the flow pattern and the mixing process of the stirred tank reactors are also important [1]. One of the applicable methods for understanding the flow behaviour and mixing process in stirred tank reactor is Computational Fluid Dynamic (CFD).

Lot of researchers has investigated about the design of the impeller that effect on the mixing parameter via the CFD simulation method. Gu *et al.,* [8] study the hydrodynamics of floating and sinking particle mixing in a stirred tank with four pitched-blade impellers and fractal impellers were compared using CFD simulation, and the results showed that increasing impeller speed enhanced the floating and sinking particle suspension quality. Gu *et al.,* [9] in other study also investigated using the CFD simulation to study the hydrodynamics of solid-liquid mixing in a stirred tank with four pitched-blade impellers and a fractal impeller. The results showed that the fractal impeller reduced the size of the impeller trailing vortex and used less power than the four pitched-blade impellers at the same impeller speed, and the greater the impeller energy consumption rate of fractal impeller, the more fractal iterations there are. Shu *et al.,* [10] investigated the effects of control factors of impeller type, gas flow rate and agitation speed on the gas holdup and power consumption. There are 3 type of impeller design including RT, BBDT and CBDT (should write the full sentences before short form) has been build. The study illustrated that the design of CBDT exhibited higher viscosity value among all the design.

The reviewed work from the previous study showed that investigation using CFD simulation method were typically comparing between different type of impeller design. Direct comparative study specifically on the blade parameter itself such as number of blades, blade angle, thickness, hub diameter, bottom tank clearance and etc. seem still limited. Hence, the present study aims to conduct a CFD simulation analysis for the effect of number of blades on the mixing flow performance characteristic using impeller type of Pitched Blade. Pitch blade impeller design is an approach of the conventional impeller type in the industry. The angle of the blades is 45-degree. In the present study, the simultaneous solution of continuity and Reynolds-averaged Navier–Stokes (RANS) were used for the numerical method. The standard k-e turbulence model was used to model the fluid flow and turbulence in the stirred tank. A simple curvature-based mesh was used for the meshing method.

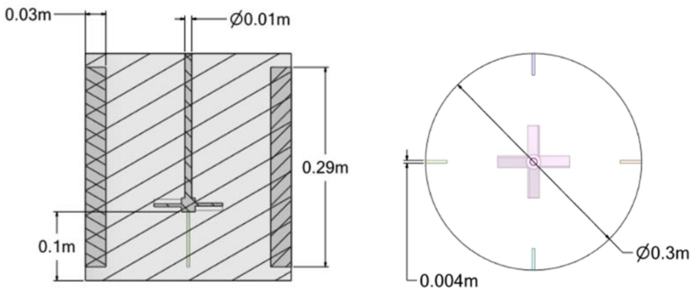
2. Methodology

2.1 Geometry Development

The present study investigated the system in a stirred tank which was equipped with three types of impellers. The stirred tank design was based on the developed model from the previous research by Rasool *et al.,* [4] as shown in Table 1. The system consists of a cylindrical vessel with a flat bottom and a diameter (D_t = 0.3 m) equal to the liquid's height ($H=D_t$). Four baffles with a width of $W=D_t/10$ are evenly positioned around the vessel. The shaft of the impeller is parallel to the vessel's axis. The diameter of the impeller, D_i, is equal to $D_t/3$. $C= D_t/3$ is the distance between the tank bottom and the impeller position C. The stirred tank was created using the usual arrangement depicted in Figure 1.

Table 1
Diameter of stirred tank

Part of stirred tank	Dimension (m)
Tank diameter	0.3
Tank height	0.33
Baffled thickness	0.004
Baffled length	0.29
Baffled width	0.29



(a) (b)

Fig. 1. Stirred tank geometry from (a) cross-sectional view (b) top view

Three different impeller types were used in this study. The conventional 4 Bladed design was set as a baseline. Whereas the modified design involved the Impeller with number of blades of three and five blades. Study by Rasool *et al.,* [4] used an impeller with blade angle of 90^o and 45^o. The study reported that 90^o produced higher velocity magnitude as compared to 45^o. This is also a primary indication that impeller with blade angle lower than 90^o will produce lower velocity magnitude. However, the study also reported that impeller with blade angle of 45^o produced larger velocity distribution in the tank compared to 90^o. This means that, although 90^o produce higher maximum velocity magnitude, but 45^o produce higher mixing zone and less dead zone compared 90^o. Hence 45^o was preferred in terms of mixing flow. The present study is thus used 45^o as an angle blade and different number of blades is implemented to investigate the effect on the performance of the flow velocity. The impeller with different number of blades was shown in the Figure 2 with the specification as listed in Table 2.

Table 2
Specification of impeller

Specification	3 Blade	4 Blade	5 Blade
Blade width (m)	0.02	0.02	0.02
Blade length (m)	0.04	0.04	0.04
Blade thickness (m)	0.004	0.004	0.004
Hub Diameter (m)	0.02	0.02	0.02
Hub height (m)	0.02	0.02	0.02

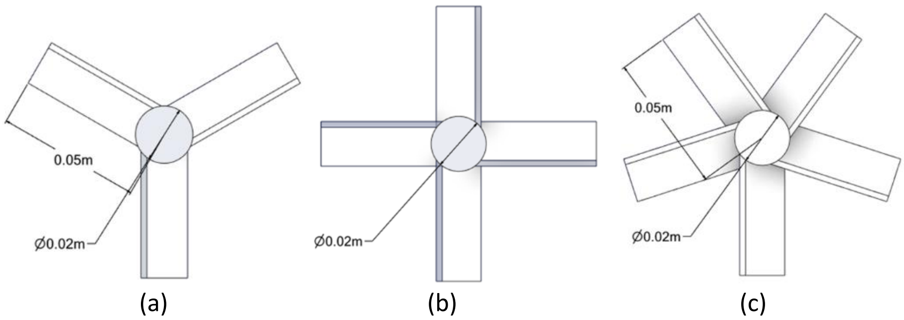


Fig. 2. Type of impeller (a) 3 Blade (b) 4 Blade and (c) 5 Blade

2.2 Governing Equation

The numerical solution of conservation equations is a part of CFD. Using the finite control volume with cylindrical coordinates, the simultaneous solution of the continuity and Reynolds-averaged Navier-Stokes (RANS) equations. The governing equation in the simulation study included mass conservation and conservation of momentum in Z, R and θ-direction as formulated in Eq. (1), Eq. (2) and Eq. (3) respectively [4].

In Z-direction

$$\rho\left(\frac{\partial U}{\partial t}+U\frac{\partial U}{\partial z}+V\frac{\partial U}{\partial r}+\frac{W}{r}\frac{\partial U}{\partial \theta}\right)=-\frac{\partial p}{\partial r}+\left[\frac{1}{r}\frac{\partial}{\partial r}(r\tau_{rz})+\frac{1}{r}\frac{\partial}{\partial \theta}\tau_{\theta z}+\frac{\partial}{\partial z}\tau_{zz}\right]+\rho Fz \tag{1}$$

In R-direction

$$\begin{aligned}\rho\left(\frac{\partial V}{\partial t}+U\frac{\partial V}{\partial z}+V\frac{\partial V}{\partial r}+\frac{W}{r}\frac{\partial V}{\partial \theta}-\frac{w^2}{r}\right)\\ =-\frac{\partial p}{\partial r}+\left[\frac{1}{r}\frac{\partial}{\partial r}(r\tau_{rr})+\frac{1}{r}\frac{\partial}{\partial \theta}\tau_{\theta r}+\frac{\partial}{\partial z}\tau_{zr}-\frac{\tau_{\theta\theta}}{r}\right]+\rho Fr\end{aligned} \tag{2}$$

In θ-direction

$$\begin{aligned}\rho\left(\frac{\partial W}{\partial t}+V\frac{\partial W}{\partial z}+\frac{W}{r}\frac{\partial W}{\partial \theta}+\frac{WV}{r}+U\frac{\partial W}{\partial z}\right)\\ =-\frac{1}{r}\frac{\partial p}{\partial \theta}+\left[\frac{1}{r^2}\frac{\partial}{\partial r}(r^2\tau_{r\theta})+\frac{1}{r}\frac{\partial}{\partial \theta}\tau_{\theta\theta}+\frac{\partial}{\partial z}\tau_{z\theta}+\frac{\tau_{\theta r}-\tau_{r\theta}}{r}\right]+\rho F\theta\end{aligned} \tag{3}$$

2.3 Boundary Condition

The stirred tank with different number of blade impeller design were constructed using Solidwork software package. Geometry of the stirred tank and impeller were shown in Figure 1 and Figure 2 respectively. These designs consist of cylindrically tapered flat bottom tanks with impeller at the centre of the tank. Each impeller is a standard pitched three-blade turbine with a pitch angle of 45°. The impellers are spaced 0.1 m from the bottom of the tank. The baffled tank has a set of four baffles with length of 0.29 m. Impellers was designated as moving walls, whereas other walls were left static by default, and all walls were given no-slip boundary requirements as depicted in Figure 3. A Multiple Reference Frame (MRF) approach was used to represent the movement of the impeller zone in the tank-fluid region, which incorporates the calculation of both fixed and moving frames [11]. On the outside and bottom walls of the stirred-tank reactor, baffles are represented as solid walls with no slip boundary condition, V = 0. The rotating domain was set to be relative with static domain. Impellers and the agitation shaft were set as moving walls. Rotational speed was set at 90 rpm.

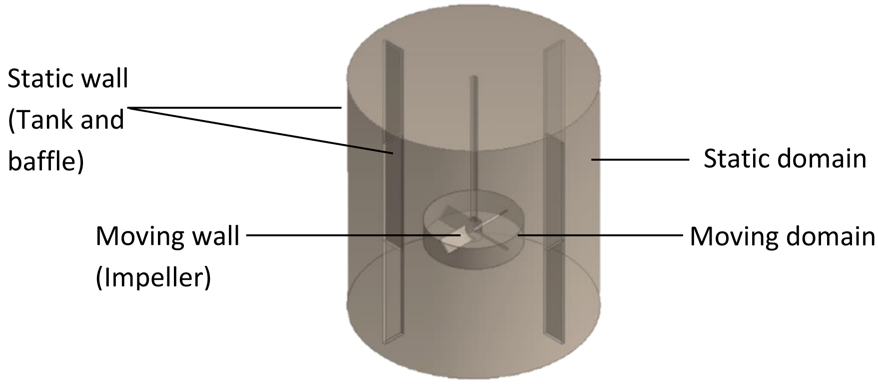


Fig. 3. Boundary condition

2.4 Grid Development

The meshing construction involved the breakdown process of domain into thousand or more shapes in which each is representing an element of the component. Meshing process for the present study was developed for each component domain in the stirred tank including the impeller, baffle and tank as depicted in Figure 4. The mesh type of tetrahedron has been applied and every core surface and volume meshing were used in this meshing model. The element size for a mesh-based curvature is numerically approximated by the average number of elements that fit inside a hypothetical circle, while taking into account the user specified minimum and maximum element sizes [12]. Table 3 showed the generated meshing properties of the present model. The total number of elements generated was 523636 with maximum aspect ratio of 39.13. The aforementioned aspect ratio was closed with the recommended value which is 35 as regard to the stability of the energy solution [13, 14].

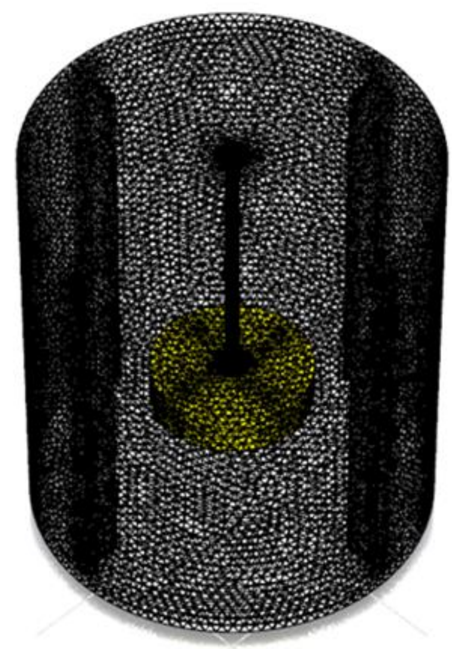


Fig. 4. The result of generated mesh model

Table 3
Mesh properties

Mesh type	Solid Mesh
Mesher used	Tetrahedron
Total nodes	134726
Total element	523636
Maximum aspect ratio	39.1325
Time to complete mesh (hh:mm:ss)	00.00.08

3. Results

3.1 Model Validation

Figure 5, showed the distribution of velocity magnitude in mixing tank and vertical position of tank at 0.01, 0.1, 0.2 and 0.3m from the bottom of the tank. The present results were validated with the previous study as shown in Figure 6. Figure 6, showed that the present study is in good agreement with the Rasool *et al.,* [4] specifically at position of 0.01, 0.2 and 0.3 as the errors were below 15%. However, the present results showed a significant deviated value with Rasool *et al.,* [4] at the position of 0.2m as the error was high which is 32%. The deviation seems to cause by the variation in the usage of turbulence model. Flow in vertical tank position of 0.2m was a lower velocity region hence low Reynold number based on the visual colour scheme contour appearance as in Figure 5.

Thus, the pressure gradient was large for the flow transition from the impeller region to the top of the tank. K-e turbulent model is typically formulated for free-shear layer flows with small pressure gradients and confined flows where Reynold's shear stresses are the most critical [14-16]. The K-e turbulence hence produced a deviated result as the accuracy of prediction for low Reynold number using this model was insufficient. This problem can be solved by alternatively using the improved k-e model formulation which is so call a realizable K-e turbulence model [17]. The mentioned model was not covered in this present study scope.

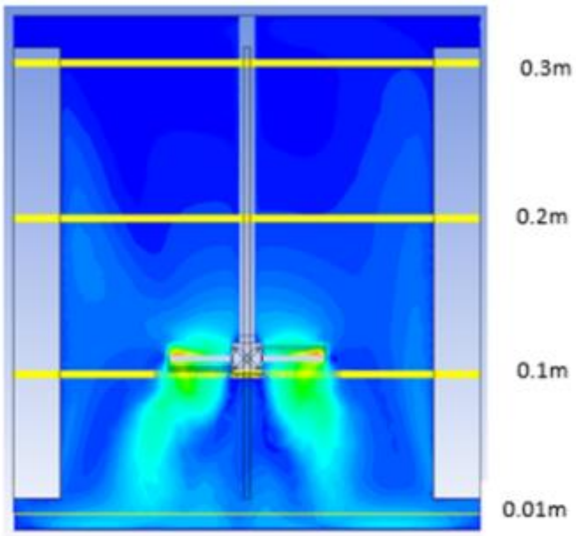


Fig. 5. The distribution of velocity in mixing tank at different horizontal position; 0.01m, 0.1m, 0.2m and 0.3m

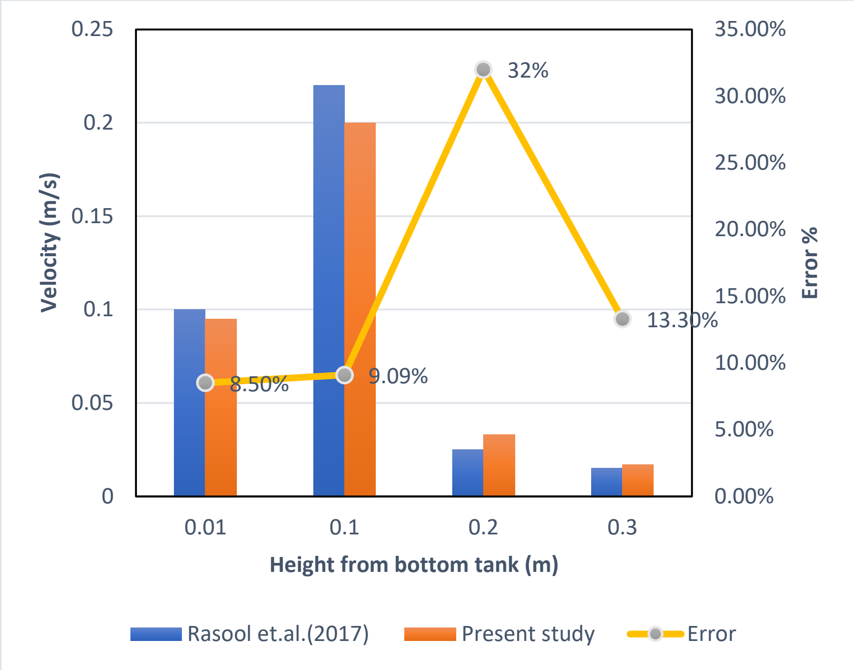
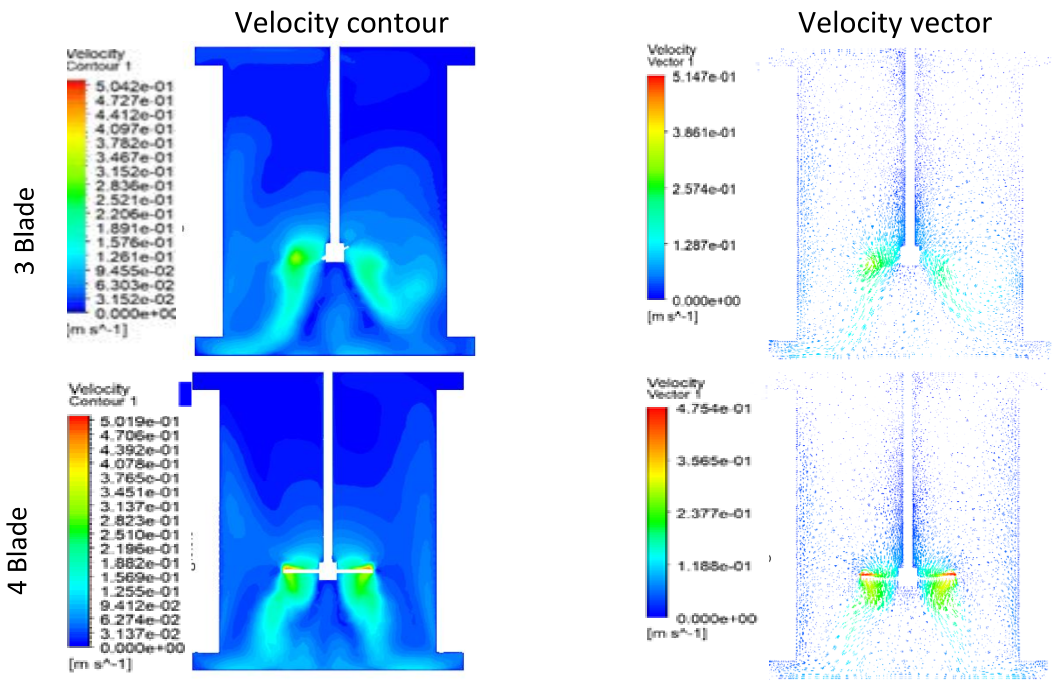


Fig. 6. The comparison of simulation result between previous study by Rasool *et al.,* [4] and present study

3.2 Distribution of Velocity Magnitude

Figure 7 depicts the differences in flow patterns caused by impellers with three blades, four blades, and five blades with the same rotating speed at 90 rpm. At the same rotational speed, the 5 Blade impeller produced wider region of distributed higher velocity flow which ranged between 0.1 - 0.5m/s. The distribution of high velocity flow produced by 3 bladed impeller was only covered some of the internal tank region. It is thus showed that the velocity of fluid flow in a tank, agitated by the 3 Blade impeller is higher than that produced by a 4 or 5 Blades impeller.



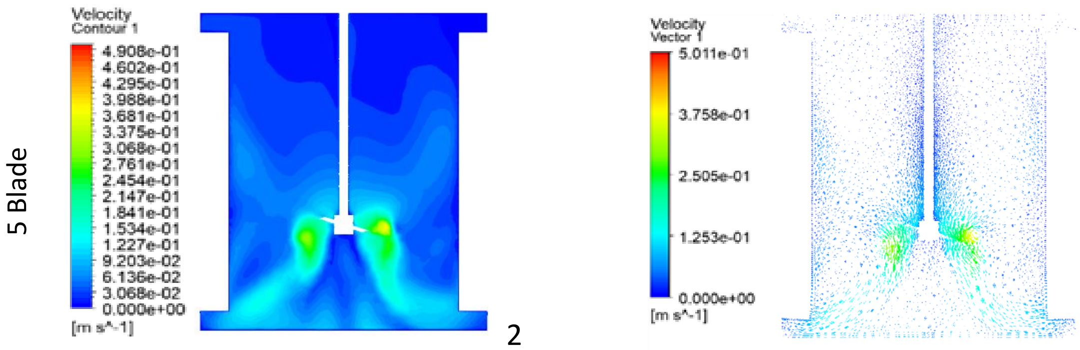
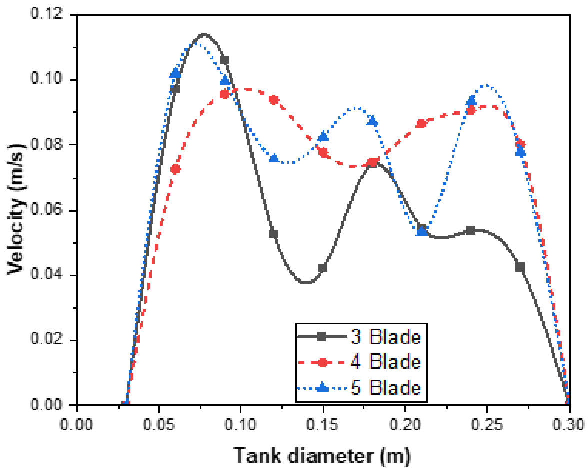


Fig. 7. Velocity Contour for 3 Blade, 4 Blade and 5 Blade impeller rotation speed at 90 rpm from Z-R plane at the height of 0.01m, 0.1m, 0.2m and 0.3m.

Figure 8 (a) and (b) showed the distribution of flow velocity in stirred tank for different number of blade impeller with rotational speed of 90rpm and at vertical height of 0.01m and 0.3m from the bottom of the tank. The velocity magnitude was generally lower as the flow is approaching tank wall for all three types of blades. The velocity produced as approaching wall was below 0.001 m/s which relative to the dead zone condition. Caillet *et al.,* [18] reported that the dead zones are defined as a velocity magnitude with the value less than 0.001 m/s. At any height of 0.01m and 0.3m, all impeller also exhibited poor mixing as the flow velocity approaching the tank wall hence dead zone occur in that region which is at diameter less than 0.05m and more than 0.25m of the tank. The effect of baffles is obvious, as they create poor mixing zones between them.

Figure 8 (a) shows that the 3 Blade impeller generated higher peak velocity compared to other impeller type at the diameter distance between 0.05m to 0.1m of the tank. However, the velocity produced by 3 bladed impeller was sharply decrease and lower compared to others impeller at the other side of tank which between the diameter of 0.1 to 0.3m. The 3-blade impeller also produced lower velocity at the centre of the tank. This fluid motion somewhat created a circular mixing zone (eddy) region in which occurred around certain location positioned in the middle of the tank from the base surface. The middle circular motion also significant at almost half of the clearance between the impeller and the tank's bottom [4]. Rasool *et al.,* [4] reported that the high mixing zone shrinks, notably in the tank's centre which caused the velocity in the middle region of the tank become lower. Figure 8 (b) showed the 4 Blade impeller produce lower distribution of flow velocity which demonstrates a poor performance flow compared to the other impellers. The 5-blade impeller seems generally exhibited higher flow velocity distribution typically by 3 to 100% and 2 to 600% for both in Figure 8 (a) and (b) respectively. Therefore 5 blades impeller seem to demonstrate effective motion of flow velocity compared to other blade arrangement. It shows that as the number of blade increase, the flow velocity also increases especially at the vertical tank location of 0.3m. Adrian *et al.,* [19] also reported that impeller with higher number of blades produced homogenous mixture due to increased velocity and reduce sedimentation. However, 4 blades impeller produced higher velocity compared to 5 blades at some region of 0.01m vertical tank position. It shows that the effect of number of blades at some point was not significant on the fluid flow in the region close to rotating impeller.



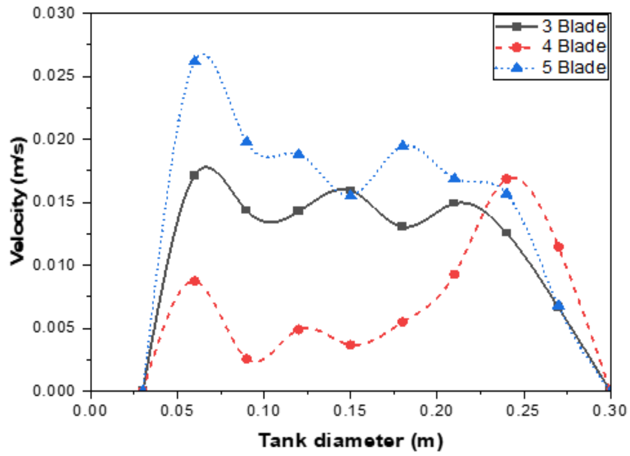


Fig. 8. The distribution of velocity magnitude with rotational speed of 90rpm at the height (a) 0.01m and (b) 0.3m

4. Conclusions

The analysis of three dimensional CFD simulation of the mixing tank with various number of blades were carried out. The model was based on RANS equation and standard k-epsilon turbulence model with the water as a fluid base medium. The moving reference frame meshing method was implemented to model the rotational motion of the impeller. The outcomes of velocity magnitude distribution profile were critically analyzed and discussed. The focused region for the flow in the mixing tank was around the rotational impeller zone.

The present data is useful in describing the flow characteristic between different type of impellers with different number of blade arrangement. The simulation results showed that the impeller blades demonstrate a significant effect to eliminate the formation of a vortex. When the number of blades increased it caused the distribution of velocity become larger hence increase the high mixing region. The results showed that 5 blades arrangement produced wider region of higher velocity magnitude distribution compared to 3 and 4 blades. The 5 blades impeller also exhibited higher velocity distribution value by up to 600% compared to 3 and 4 blades arrangement typically at the position of 0.3m of the tank. The 5 blades impeller also produced more peak value velocity magnitude at the tank location of 0.01m. However, some locations show that 3 and 4 blades produced higher velocity than 5 blades. It shows that the effect of number of blades was not significant at the region close to impeller.

Acknowledgement
The authors would like to thank the Ministry of Higher Education Malaysia for supporting this research under Fundamental Research Grant Scheme Vot No. FRGS/1/2019/TK07/UTHM/03/2 and partially sponsored by Universiti Tun Hussein Onn Malaysia.

References
[1] Hoseini, S. S., G. Najafi, B. Ghobadian, and A. H. Akbarzadeh. "Impeller shape-optimization of stirred-tank reactor: CFD and fluid structure interaction analyses." *Chemical Engineering Journal* 413 (2021): 127497. https://doi.org/10.1016/j.cej.2020.127497

[2] Shu, Lei, Mingjin Yang, Hang Zhao, Tianfu Li, Ling Yang, Xiang Zou, and Yunwu Li. "Process optimization in a stirred tank bioreactor based on CFD-Taguchi method: A case study." *Journal of Cleaner Production* 230 (2019): 1074-1084. https://doi.org/10.1016/j.jclepro.2019.05.083

[3] Ameur, Houari, and Youcef Kamla. "Geometrical modifications of the anchor impeller to enhance the overall performances in stirred tanks." *Instal* 6 (2020): 42-45. https://doi.org/10.36119/15.2020.6.5
[4] Rasool, Adnan AAR, Safaa S. Ahmad, and Faik Hamad. "Effect of impeller type and rotational speed on flow behavior in fully baffled mixing tank." *International Journal of Advanced Research (IJAR)* 5, no. 1 (2017): 1195-1208. https://doi.org/10.21474/IJAR01/2871
[5] Jaszczur, Marek, Anna Młynarczykowska, and Luana Demurtas. "Effect of impeller design on power characteristics and Newtonian fluids mixing efficiency in a mechanically agitated vessel at low Reynolds numbers." *Energies* 13, no. 3 (2020): 640. https://doi.org/10.3390/en13030640
[6] Agarwal, Alankar, Gurveer Singh, and Akshay Prakash. "Numerical investigation of flow behavior in double-rushton turbine stirred tank bioreactor." *Materials today: proceedings* 43 (2021): 51-57. https://doi.org/10.1016/j.matpr.2020.11.208
[7] Ameur, Houari, Youcef Kamla, and Djamel Sahel. "Optimization of the operating and design conditions to reduce the power consumption in a vessel stirred by a paddle impeller." *Periodica Polytechnica Mechanical Engineering* 62, no. 4 (2018): 312-319. https://doi.org/10.3311/PPme.12372
[8] Joanna Pyzel, "Change and Continuity in the Danubian Longhouses of Lowland Poland," in *Tracking the Neolithic House in Europe*, eds. Hofmann D., Smyth J. (New York: Springer, 2013), 183-196. https://doi.org/10.1016/j.jtice.2020.11.013
[9] Gu, Deyin, Mei Ye, Xingming Wang, and Zuohua Liu. "Numerical investigation on mixing characteristics of floating and sinking particles in a stirred tank with fractal impellers." *Journal of the Taiwan Institute of Chemical Engineers* 116 (2020): 51-61. https://doi.org/10.1016/j.apt.2019.06.028
[10] Hu, Yinyu, Zhe Liu, Jichu Yang, Yong Jin, and Yi Cheng. "Study on the reactive mixing process in an unbaffled stirred tank using planar laser-induced fluorescence (PLIF) technique." *Chemical engineering science* 65, no. 15 (2010): 4511-4518. https://doi.org/10.1016/j.ces.2010.04.033
[11] Mahrous, Abdel-Fattah. "Computational Fluid Dynamics Study of a Modified Savonius Rotor Blade by Universal Consideration of Blade Shape Factor Concept." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 85, no. 1 (2021): 22-39. https://doi.org/10.37934/arfmts.85.1.2239
[12] Ahmad, Mohd Nazri, Mohammad Khalid Wahid, Nurul Ain Maidin, Mohd Hidayat Ab Rahman, Mohd Hairizal Osman, Ridhwan Jumaidin, and Muhammad Afiq Abu Hassan. "Flow Analysis of Five-Axis Impeller in Vacuum Casting by Computer Simulation." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 61, no. 2 (2019): 181-189.
[13] Gradov, Dmitry, Arto Laari, and Tuomas Koiranen. "Assessment of Interaction between Baffles and Impeller Blades in Stirred Tanks." In *MATEC Web of Conferences*, vol. 49, p. 04002. EDP Sciences, 2016. https://doi.org/10.1051/matecconf/20164904002
[14] Fluent, A. N. S. Y. S. "Ansys fluent theory guide." *Ansys Inc., USA* 15317 (2011): 724-746.
[15] Torotwa, Ian, and Changying Ji. "A study of the mixing performance of different impeller designs in stirred vessels using computational fluid dynamics." *Designs* 2, no. 1 (2018): 10. https://doi.org/10.3390/designs2010010
[16] Qing, Nelvin Kaw Chee, Nor Afzanizam Samiran, and Razlin Abd Rashid. "CFD Simulation analysis of Sub-Component in Municipal Solid Waste Gasification using Plasma Downdraft Technique." *Journal of Advanced Research in Numerical Heat Transfer* 8, no. 1 (2022): 36-43.
[17] Subramaniam, Thineshwaran, and Mohammad Rasidi Rasani. "Pulsatile CFD Numerical Simulation to investigate the effect of various degree and position of stenosis on carotid artery hemodynamics." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 26, no. 2 (2022): 29-40. https://doi.org/10.37934/araset.26.2.2940
[18] Caillet, Hélène, Alain Bastide, and Laetitia Adelard. "CFD Simulations in Mechanically Stirred Tank and Flow Field Analysis: Application to the Wastewater (Sugarcane Vinasse) Anaerobic Digestion." *Promising Techniques for Wastewater Treatment and Water Quality Assessment* (2021): 179. https://doi.org/10.5772/intechopen.93926
[19] Stuparu, Adrian, Romeo Susan-Resiga, and Constantin Tanasa. "CFD assessment of the hydrodynamic performance of two impellers for a baffled stirred reactor." *Applied Sciences* 11, no. 11 (2021): 4949. https://doi.org/10.3390/app11114949